Aerodynamics of Race Car Wings: A CFD Study

Simon Durrer

Grand Valley State University

Follow this and additional works at: http://scholarworks.gvsu.edu/theses

Part of the Engineering Commons

Recommended Citation

http://scholarworks.gvsu.edu/theses/798

This Thesis is brought to you for free and open access by the Graduate Research and Creative Practice at ScholarWorks@GVSU. It has been accepted for inclusion in Masters Theses by an authorized administrator of ScholarWorks@GVSU. For more information, please contact scholarworks@gvsu.edu.
Aerodynamics of Race Car Wings, a CFD Study

Simon Durrer

A Thesis Submitted to the Graduate Faculty of

GRAND VALLEY STATE UNIVERSITY

In

Partial Fulfillment of the Requirements

For the Degree of

Master of Science in Engineering

Padnos Collage of Engineering and Computing

April 2016
Acknowledgments

I would like to thank my advising professor Dr. Wael Mokhtar for all the support and encouragement throughout the work of this thesis. Further, I would like to thank my committee members Dr. Wendy Reffeor and Dr. Mehmet Sözen for their support and inputs while reviewing my work. I would also like to thank Carl Strebel, who provided help and assistance using the computational resources at Grand Valley State University.
Abstract

Formula 1 racing is one of the most advanced technological sports. The aerodynamic on open wheel race cars is essential for the performance during a race. The front wing on a race car produces about 30 percent of the entire downforce of a race car. Several studies on front wings for open wheel race cars are conducted by various authors. A number of research studies include single element airfoils in ground effect and undisturbed flow. Numerical and experimental studies show that by decreasing the ground clearance, the downforce increases. The most efficient ground clearance is reported to be approximately 10 percent of the chord length. Another effective parameter to increase the downforce is the increase of angle of attack. Both increase of angle of attack and decrease of ground clearance result in an increasing of drag.

Experimental studies on race car front wings have been carried out in disturbed flow. As soon as a wing operates in a wake, a significant change on the aerodynamic forces can be found.

This aerodynamic study of race car wings will focus on a wing operating in a wake. The wing model is analyzed prior in freestream and ground effect only. The study in ground effect shows a maximum downforce at a ground clearance of 22 percent of the chord length. The study in a wake consists of different ground clearance levels and different distances between a bluff body and the analyzed wing. At a distance of 10 percent of a car length, both downforce and drag experience a significant decrease compared to undisturbed flow. While moving the wing further downstream, the lift and drag coefficient recover towards the values of a wing operating in ground effect only. The most efficient ground clearance point moves from 22 percent to 25 percent of the chord length at a
distance of 30, respectively 50 percent of a car length. The flow structure analysis clearly showed a positive impact of the wing tip vortices coming from the bluff body. All studies are performed using Star CCM+, a commercial CFD code developed by CD Adapco.
# Table of Contents

Acknowledgments ................................................................................................................. 3

Abstract ................................................................................................................................. 4

Table of Contents ................................................................................................................. 6

List of Figures ....................................................................................................................... 11

List of Tables ....................................................................................................................... 25

Key to Symbols .................................................................................................................... 26

Abbreviations ...................................................................................................................... 28

1 Introduction ....................................................................................................................... 29

1.1 Background ................................................................................................................... 29

1.1.1 Function of Race Car Wings .................................................................................. 30

1.1.2 History of Race Car Wings .................................................................................... 33

1.2 Goal of Study ............................................................................................................... 34

1.2.1 Phase 1 – A Wing in Freestream ......................................................................... 35

1.2.2 Phase 2 – Creation of a Bluff Body ...................................................................... 36

1.2.3 Phase 3 – A Wing Operating in Ground Effect ....................................................... 37

1.2.4 Phase 4 – A Wing Operating in a Wake ................................................................. 39

2 Literature Review ............................................................................................................. 42

2.1 Wing Profiles ............................................................................................................... 42

2.2 Single Element Front Wings ...................................................................................... 43
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.2.1 Numerical Studies in Undisturbed Flow</td>
<td>43</td>
</tr>
<tr>
<td>2.2.2 Numerical Studies in Disturbed Flow</td>
<td>46</td>
</tr>
<tr>
<td>2.2.3 Experimental Studies in Undisturbed Flow with Ground Influence</td>
<td>47</td>
</tr>
<tr>
<td>2.2.4 Experimental Studies in a Wake</td>
<td>49</td>
</tr>
<tr>
<td>2.3 Multiple Element Front Wings</td>
<td>51</td>
</tr>
<tr>
<td>2.3.1 Numerical Study in Undisturbed Flow</td>
<td>52</td>
</tr>
<tr>
<td>2.3.2 Experimental Studies with moving Ground Influence</td>
<td>52</td>
</tr>
<tr>
<td>2.3.3 Experimental Studies with fixed Ground Influence</td>
<td>54</td>
</tr>
<tr>
<td>2.4 Influence of Endplates</td>
<td>55</td>
</tr>
<tr>
<td>2.5 Gurney Wing Flap</td>
<td>56</td>
</tr>
<tr>
<td>2.6 Studies of Rear Wing</td>
<td>57</td>
</tr>
<tr>
<td>3 Methodology</td>
<td>58</td>
</tr>
<tr>
<td>3.1 Theoretical Equations</td>
<td>58</td>
</tr>
<tr>
<td>3.1.1 Reynolds Number</td>
<td>58</td>
</tr>
<tr>
<td>3.1.2 Lift or Downforce Coefficient</td>
<td>59</td>
</tr>
<tr>
<td>3.1.3 Drag Coefficient</td>
<td>59</td>
</tr>
<tr>
<td>3.1.4 Percentage Change of a Quantity</td>
<td>60</td>
</tr>
<tr>
<td>3.2 CFD Modeling</td>
<td>60</td>
</tr>
<tr>
<td>3.2.1 Pre-Processor</td>
<td>61</td>
</tr>
<tr>
<td>3.2.2 Errors and Uncertainties in CFD</td>
<td>80</td>
</tr>
</tbody>
</table>
4 Phase 1 – Study of a Wing in Freestream .......................................................... 87

4.1 Model .................................................................................................................. 87

4.1.1 Meshing ........................................................................................................... 88

4.1.2 Physics ........................................................................................................... 90

4.1.3 Error Analysis ............................................................................................... 91

4.2 Results ................................................................................................................ 94

4.2.1 Lift Coefficient Study in Freestream .............................................................. 94

4.2.2 Drag Coefficient Study in Freestream ........................................................... 96

4.2.3 Flow Structure Analysis of a Wing in Freestream ........................................ 97

4.3 Conclusion of Phase 1 – Study of a Wing in Freestream ............................... 108

5 Phase 2 – Creation of a Bluff Body ..................................................................... 109

5.1 Model .................................................................................................................. 109

5.1.1 Meshing ......................................................................................................... 110

5.1.2 Error Analysis ............................................................................................... 112

5.2 Results ................................................................................................................ 114

5.3 Phase 2 Conclusion – Creation of a Bluff Body .............................................. 122

6 Phase 3 - A Wing Operating in Ground Effect ............................................... 123

6.1 Model .................................................................................................................. 123

6.1.1 Meshing ......................................................................................................... 124

6.1.2 Physics .......................................................................................................... 126
6.1.3 Error Analysis ................................................................. 127

6.2 Results .................................................................................. 129

6.2.1 Lift Coefficient Study of a Wing Operating in Ground Effect ........ 129
6.2.2 Drag Coefficient Study of a Wing Operating in Ground Effect ........ 131
6.2.3 Flow Structure Analysis of a Wing Operating in Ground Effect .... 132

6.3 Conclusion of Phase 3 - A Wing Operating in Ground Effect ........ 143

7 Phase 4 – A Wing Operating in a Wake ....................................... 143

7.1 Model ..................................................................................... 144

7.1.1 Meshing .............................................................................. 145
7.1.2 Physics ............................................................................... 148
7.1.3 Error Analysis ................................................................. 148

7.2 Results of a Wing Operating in a Wake ................................... 151

7.2.1 Lift Coefficient Study of a Wing Operating in a Wake .............. 151
7.2.2 Drag Coefficient Study of a Wing Operating in a Wake ............ 155
7.2.3 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.1 157
7.2.4 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.3 172
7.2.5 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.5 193

7.3 The Effect of Velocity on a Wing Operating in a Wake ............... 212

7.3.1 Lift Coefficient Study on Velocity Effect ................................. 213
7.3.2 Drag Coefficient Study on Velocity Effect .............................. 215
7.3.3 Flow Structure Analysis ................................................................. 217
7.4 Conclusion of Phase 4 - A Wing operating in a Wake .................... 240
8 Discussion ................................................................................................. 243
9 Conclusion ................................................................................................. 246
10 Future Work ............................................................................................ 248
11 Appendix ................................................................................................. 249
12 References ............................................................................................... 251
List of Figures

Figure 1: Phase 1 model ........................................................................................................ 36
Figure 2: Bluff body ................................................................................................................ 37
Figure 3: Phase 3 model ........................................................................................................ 38
Figure 4: Phase 2 wake size result ........................................................................................ 40
Figure 5: Phase 4 model ........................................................................................................ 40
Figure 6: S1223 airfoil geometry .......................................................................................... 42
Figure 7: Force coefficient of transition free vs. transition fixed [20]................................. 49
Figure 8: Experimental downforce coefficient of LS(1)0417 in freestream (FC1), small wake (FC2), and disturbed air (FC3) ................................................................. 50
Figure 9: Downforce vs. ground clearance for double-element wing..................................... 53
Figure 10: (a) $C_L$ vs. angle of attack and (b) $C_L$ vs. $C_D$, for UIUC700I at $Re=1.1 \times 10^6$ and $H/c = 0.3$ ............................................................................................................................................. 55
Figure 11: Mass flow in and out of a fluid element ................................................................ 68
Figure 12: Stress components in x-direction ......................................................................... 70
Figure 13: Components of the heat flux vector .................................................................... 72
Figure 14: Freestream model AOA = 6° .............................................................................. 88
Figure 15: Mesh refinement blocks (a) wing refinement (b) trailing edge refinement .... 88
Figure 16: Mesh view of center plane, freestream at AOA=6° ........................................... 89
Figure 17: Mesh around the wing, freestream at AOA=6° .................................................. 89
Figure 18: Residuals of freestream case with AOA=6° ......................................................... 92
Figure 19: Drag coefficient tracker for freestream case with AOA=6° ............................... 92
Figure 20: Lift coefficient tracker freestream case with AOA=6° ...................................... 93
Figure 21: Boundary layer at 2/3 of chord length on top surface for freestream with AOA=6° .............................................................................................................................................. 93
Figure 22: Velocity distribution including far field for freestream at AOA=6°................. 94
Figure 23: Lift coefficient vs. angle of attack in free stream........................................ 95
Figure 24: Drag coefficient vs. angle of attack in free stream........................................ 97
Figure 25: Velocity distribution on center plane, freestream with AOA=0°..................... 99
Figure 26: Velocity distribution on center plane, freestream with AOA=6°..................... 99
Figure 27: Velocity distribution on center plane, freestream with AOA=8°..................... 99
Figure 28: Velocity distribution on center plane, freestream with AOA=14°................... 100
Figure 29: Pressure distribution on center plane, freestream with AOA=0°................... 101
Figure 30: Pressure distribution on center plane, freestream with AOA=6°................... 101
Figure 31: Pressure distribution on center plane, freestream with AOA=8°................... 102
Figure 32: Pressure distribution on center plane, freestream with AOA=14°................... 102
Figure 33: Velocity vectors at separation region on center plane, freestream with AOA=0°...................................................................................................................... 103
Figure 34: Velocity vectors at separation region on center plane, freestream with AOA=6°...................................................................................................................... 103
Figure 35: Velocity vectors at separation region on center plane, freestream with AOA=8°...................................................................................................................... 103
Figure 36: Velocity vectors at separation region on center plane, freestream with AOA=14°..................................................................................................................... 104
Figure 37: Velocity distribution at a distance D/c=0.66 for freestream at AOA=0° ......... 104
Figure 38: Velocity distribution at a distance D/c=0.66 for freestream at AOA=6° ......... 105
Figure 39: Velocity distribution at a distance D/c=0.66 for freestream at AOA=8° ......... 105
Figure 40: Velocity distribution at a distance D/c=0.66 for freestream at AOA=14° ....... 105
Figure 41: Wing tip vortex for freestream at AOA =0°..................................................... 106
Figure 42: Wing tip vortex for freestream at AOA =6°..................................................... 106
Figure 43: Wing tip vortex for freestream at AOA =8°..................................................... 107
Figure 44: Wing tip vortex for freestream at AOA =14° .......................................................... 107
Figure 45: Bluff body dimensions ......................................................................................... 109
Figure 46: (a) CFD model of bluff body simulation (b) 3D bluff body model .............. 110
Figure 47: Mesh refinement blocks for bluff body simulation ........................................ 111
Figure 48: Mesh view of center plane, bluff body simulation ........................................... 111
Figure 49: Refined mesh area, bluff body simulation .......................................................... 112
Figure 50: Residuals of bluff body simulation ...................................................................... 112
Figure 51: Velocity distribution of bluff body wake after 600 iterations .................... 113
Figure 52: Velocity distribution of bluff body wake after 900 iterations ....................... 113
Figure 53: Velocity distribution of bluff body wake after 1000 iterations ...................... 114
Figure 54: Velocity distribution of bluff body wake after 1225 iterations ...................... 114
Figure 55: Velocity distribution of bluff body wake after 1350 iterations ...................... 114
Figure 56: Velocity distribution on center plane of the bluff body .................................. 115
Figure 57: Velocity distribution for velocities higher than 30 m/s ................................. 116
Figure 58: Cross-section plane placing in bluff body wake ............................................. 116
Figure 59: Velocity on cross-section view of bluff body wake at D/L=0.1 .................. 117
Figure 60: Wake vectors on cross-section D/L=0.1 .......................................................... 118
Figure 61: Velocity on cross-section view of bluff body wake at D/L=0.3 .................... 119
Figure 62: Velocity on cross-section view of bluff body wake at D/L=0.5 ................. 119
Figure 63: Wake vectors on cross-section D/L=0.3 .......................................................... 120
Figure 64: Wake vectors on cross-section D/L=0.5 .......................................................... 120
Figure 65: Pressure distribution on bluff body ................................................................. 121
Figure 66: Velocity streamlines around bluff body with visual pressure distribution .... 121
Figure 67: Streamline tubes around the bluff body .......................................................... 122
Figure 68: CFD model phase 3 at H/c = 0.3 ................................................................. 124
Figure 69: Refinement blocks phase 3 model at H/c=0.3 ............................................ 125
Figure 70: Mesh view of center plane, phase 3 at H/c = 0.3 ........................................... 126
Figure 71: Refined mesh area, phase 3 at H/c = 0.3 ..................................................... 126
Figure 72: Residuals of governing equation, phase 3 at H/c=0.3 ............................... 127
Figure 73: Lift coefficient tracker, phase 3 at H/c=0.3 .................................................. 128
Figure 74: Drag Coefficient Tracker, phase 3 at H/c=0.3 ............................................ 128
Figure 75: Velocity distribution including far field, phase 3 at H/c=0.3 ...................... 129
Figure 76: Change in lift coefficient vs. freestream in ground effect ......................... 130
Figure 77: Change in drag coefficient vs. freestream in ground effect ....................... 132
Figure 78: Velocity distribution on center plane at H/c=0.3 in ground effect ............. 133
Figure 79: Velocity distribution on center plane at H/c=0.24 in ground effect .......... 134
Figure 80: Velocity distribution on center plane at H/c=0.2333 in ground effect ...... 134
Figure 81: Velocity distribution on center plane at H/c=0.2266 in ground effect ..... 134
Figure 82: Velocity distribution on center plane at H/c=0.22 in ground effect ........... 135
Figure 83: Velocity distribution on center plane at H/c=0.17 in ground effect .......... 135
Figure 84: Pressure distribution on center plane at H/c=0.3 in ground effect ......... 136
Figure 85: Pressure distribution on center plane at H/c=0.24 in ground effect ......... 136
Figure 86: Pressure distribution on center plane at H/c=0.22 in ground effect ......... 137
Figure 87: Pressure distribution on center plane at H/c=0.17 in ground effect ......... 137
Figure 88: Streamlines on center plane at H/c=0.3 in ground effect ......................... 138
Figure 89: Streamlines on center plane at H/c=0.24 in ground effect ......................... 138
Figure 90: Streamlines on center plane at H/c=0.2266 in ground effect

Figure 91: Streamlines on center plane at H/c=0.22 in ground effect

Figure 92: Streamlines on center plane at H/c=0.2166 in ground effect

Figure 93: Streamlines on center plane at H/c=0.17 in ground effect

Figure 94: Wake velocity distribution at a distance D/c=0.66 for freestream

Figure 95: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.3

Figure 96: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.24

Figure 97: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.22

Figure 98: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.17

Figure 99: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.3

Figure 100: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.24

Figure 101: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.22

Figure 102: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.17

Figure 103: Model setup phase 4

Figure 104: Star CCM+ symmetric model at H/c=0.3 and D/L=0.3

Figure 105: Mesh refinement blocks (a) for the wake (b) around the wing

Figure 106: Center plane view of the mesh including far field at H/c=0.3 and D/L=0.3

Figure 107: Center plane view of the mesh refinements at H/c=0.3 and D/L=0.3

Figure 108: Mesh around the wing at H/c=0.3 and D/L=0.3

Figure 109: Residuals of governing equation, phase 4 at H/c=0.21 and D/L=0.5

Figure 110: Lift coefficient tracker, phase 4 at H/c=0.21 and D/L=0.5

Figure 111: Drag coefficient tracker, phase 4 at H/c=0.21 and D/L=0.5
Figure 112: Velocity distribution including far field, phase 4 at H/c=0.21 and D/L=0.5 151

Figure 113: Change of lift coefficient vs freestream 1 ................................................. 153

Figure 114: Comparison of change of lift coefficient between D/L=0.5 and undisturbed flow ....................................................................................................................... 154

Figure 115: Comparison of change of lift coefficient between D/L=0.5 and 0.3 ......... 155

Figure 116: Change in drag coefficient vs. freestream for a wing operating in a wake 156

Figure 117: Comparison of change of drag coefficient for D/L=0.3, 0.5 and undisturbed ................................................................. ....................................................................... 157

Figure 118: Velocity distribution on center plane at H/c=0.25 and D/L=0.1 .......... 158

Figure 119: Velocity distribution on center plane (a) Bluff body (b) Wing placed at D/L=0.1 .......................................................................................................................... 159

Figure 120: Velocity distribution on center plane at H/c=0.25 and D/L=0.1 ............ 160

Figure 121: Velocity distribution on center plane at H/c=0.23 and D/L=0.1 .......... 160

Figure 122: Velocity distribution on center plane at H/c=0.22 and D/L=0.1 .......... 160

Figure 123: Velocity distribution on center plane at H/c=0.21 and D/L=0.1 .......... 161

Figure 124: Velocity distribution on center plane at H/c=0.2 and D/L=0.1 .......... 161

Figure 125: Velocity vectors on center plane at H/c=0.25 and D/L=0.1 ............... 162

Figure 126: Velocity vectors on center plane at H/c=0.22 and D/L=0.1 ............... 162

Figure 127: Velocity vectors on center plane at H/c=0.21 and D/L=0.1 ............... 163

Figure 128: Pressure distribution on center plane of complete model at H/c=0.25 and D/L=0.1 ................................................................................................................. 163

Figure 129: Pressure distribution on center plane at H/c=0.25 and D/L=0.1 .......... 164

Figure 130: Pressure distribution on center plane at H/c=0.23 and D/L=0.1 .......... 164

Figure 131: Pressure distribution on center plane at H/c=0.22 and D/L=0.1 .......... 165

Figure 132: Pressure distribution on center plane at H/c=0.2 and D/L=0.1 .......... 165
Figure 133: Center plane streamlines on velocity distribution at H/c=0.25 and D/L=0.1 ........................................................................................................................................................................166

Figure 134: Center plane streamlines on velocity distribution at H/c=0.23 and D/L=0.1 ........................................................................................................................................................................166

Figure 135: Center plane streamlines on velocity distribution at H/c=0.22 and D/L=0.1 ........................................................................................................................................................................166

Figure 136: Center plane streamlines on velocity distribution at H/c=0.21 and D/L=0.1 ........................................................................................................................................................................167

Figure 137: Center plane streamlines on velocity distribution at H/c=0.2 and D/L=0.1 ........................................................................................................................................................................167

Figure 138: Wake velocity distribution (a) Bluff body only at D/L=0.1 (b) Behind a wing, wing placed at H/c=0.25 and D/L=0.1 ........................................................................................................................................................................168

Figure 139: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.1 ........................................................................................................................................................................169

Figure 140: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.23 and D/L=0.1 ........................................................................................................................................................................169

Figure 141: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.1 ........................................................................................................................................................................169

Figure 142: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.21 and D/L=0.1 ........................................................................................................................................................................169

Figure 143: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.2 and D/L=0.1 ........................................................................................................................................................................169

Figure 144: Wing tip vortices (a) bluff body only (b) wing Behind a wing, wing placed at H/c=0.25 and D/L=0.1 ........................................................................................................................................................................170

Figure 145: Streamlines around the model with shown pressure distribution with wing at H/c=0.25 and D/L=0.1 (a) top view (b) 3D view ........................................................................................................................................................................171

Figure 146: Streamlines around the model with shown pressure distribution with wing at H/c=0.23 and D/L=0.1 (a) top view (b) 3D view ........................................................................................................................................................................171

Figure 147: Streamlines around the model with shown pressure distribution with wing at H/c=0.22 and D/L=0.1 (a) top view (b) 3D view ........................................................................................................................................................................172

Figure 148: Streamlines around the model with shown pressure distribution with wing at H/c=0.2 and D/L=0.1 (a) top view (b) 3D view ........................................................................................................................................................................172
Figure 149: Velocity distribution on center plane of complete model at H/c=0.27 and D/L=0.3 .......................................................... 173
Figure 150: Velocity distribution on center plane at H/c=0.27 and D/L=0.3 ............ 174
Figure 151: Velocity distribution on center plane at H/c=0.25 and D/L=0.3 .......... 175
Figure 152: Velocity distribution on center plane at H/c=0.24 and D/L=0.3 .......... 175
Figure 153: Velocity distribution on center plane at H/c=0.23 and D/L=0.3 .......... 175
Figure 154: Velocity distribution on center plane at H/c=0.22 and D/L=0.3 .......... 175
Figure 155: Velocity vectors of bluff body wake on center plane at H/c=0.27 and D/L=0.3 ............................................................................................................ 176
Figure 156: Velocity vectors around the wing on center plane at H/c=0.27 and D/L=0.3 ............................................................................................................ 177
Figure 157: Velocity vectors around the wing on center plane at H/c=0.26 and D/L=0.3 ............................................................................................................ 177
Figure 158: Velocity vectors around the wing on center plane at H/c=0.25 and D/L=0.3 ............................................................................................................ 178
Figure 159: Velocity vectors around the wing on center plane at H/c=0.24 and D/L=0.3 ............................................................................................................ 178
Figure 160: Velocity vectors around the wing on center plane at H/c=0.23 and D/L=0.3 ............................................................................................................ 178
Figure 161: Pressure distribution on center plane of complete model at H/c=0.27 and D/L=0.3 ............................................................................................................ 179
Figure 162: Pressure distribution on center plane at H/c=0.27 and D/L=0.3 .......... 180
Figure 163: Pressure distribution on center plane at H/c=0.26 and D/L=0.3 .......... 181
Figure 164: Pressure distribution on center plane at H/c=0.25 and D/L=0.3 .......... 181
Figure 165: Pressure distribution on center plane at H/c=0.24 and D/L=0.3 .......... 181
Figure 166: Pressure distribution on center plane at H/c=0.23 and D/L=0.3 .......... 182
Figure 167: Center plane streamlines on velocity distribution at H/c=0.27 and D/L=0.3 ............................................................................................................ 182
Figure 168: Center plane streamlines around the wing at H/c=0.27 and D/L=0.3 ...... 183
Figure 169: Center plane streamlines around the wing at H/c=0.26 and D/L=0.3 ...... 183
Figure 170: Center plane streamlines around the wing at H/c=0.25 and D/L=0.3 ...... 184
Figure 171: Center plane streamlines around the wing at H/c=0.24 and D/L=0.3 ...... 184
Figure 172: Center plane streamlines around the wing at H/c=0.23 and D/L=0.3 ...... 184
Figure 173: Center plane streamlines around the wing at H/c=0.22 and D/L=0.3 ...... 184
Figure 174: Wake velocity distribution (a) Bluff body only at D/L=0.3 (b) Behind a wing, wing placed at H/c=0.27 and D/L=0.3 ................................................................. 185
Figure 175: Wake vortices (a) Bluff body only at D/L=0.3 (b) Behind a wing, wing placed at H/c=0.27 and D/L=0.3 ................................................................. 185
Figure 176: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.27 and D/L=0.3 ................................................................. 186
Figure 177: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.3 ................................................................. 187
Figure 178: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.3 ................................................................. 187
Figure 179: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.3 ................................................................. 187
Figure 180: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.23 and D/L=0.3 ................................................................. 187
Figure 181: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.3 ................................................................. 188
Figure 182: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.2 and D/L=0.3 ................................................................. 188
Figure 183: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.3 ................................................................. 188
Figure 184: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.3 ................................................................. 189
Figure 185: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.3 ................................................................. 189
Figure 186: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.3 ................................................................. 189
Figure 187: Streamlines around the model (a) bluff body only (b) a wing placed at H/c=0.22 and D/L=0.1 ................................................................. 190

Figure 188: Streamlines around the model with shown pressure distribution with wing at H/c=0.27 and D/L=0.3 (a) top view (b) 3D view ................................................................. 191

Figure 189: Streamlines around the model with shown pressure distribution with wing at H/c=0.25 and D/L=0.3 (a) top view (b) 3D view ................................................................. 191

Figure 190: Streamlines around the model with shown pressure distribution with wing at H/c=0.24 and D/L=0.3 (a) top view (b) 3D view ................................................................. 192

Figure 191: Streamlines around the model with shown pressure distribution with wing at H/c=0.22 and D/L=0.3 (a) top view (b) 3D view ................................................................. 192

Figure 192: Velocity distribution on center plane of the model at H/c=0.26 and D/L=0.5 .................................................................................................................. 194

Figure 193: Velocity distribution on center plane at H/c=0.26 and D/L=0.5 .................. 195

Figure 194: Velocity distribution on center plane at H/c=0.25 and D/L=0.5 .................. 195

Figure 195: Velocity distribution on center plane at H/c=0.24 and D/L=0.5 .................. 195

Figure 196: Velocity distribution on center plane at H/c=0.23 and D/L=0.5 .................. 195

Figure 197: Velocity distribution on center plane at H/c=0.22 and D/L=0.5 .................. 195

Figure 198: Velocity distribution on center plane at H/c=0.21 and D/L=0.5 .................. 196

Figure 199: Velocity vectors of bluff body wake on center plane at H/c=0.26 and D/L=0.5 .................................................................................................................. 197

Figure 200: Velocity vectors around the wing on center plane at H/c=0.26 and D/L=0.5 .................................................................................................................. 198

Figure 201: Velocity vectors around the wing on center plane at H/c=0.25 and D/L=0.5 .................................................................................................................. 198

Figure 202: Velocity vectors around the wing on center plane at H/c=0.24 and D/L=0.5 .................................................................................................................. 198

Figure 203: Velocity vectors around the wing on center plane at H/c=0.23 and D/L=0.5 .................................................................................................................. 198

Figure 204: Velocity vectors around the wing on center plane at H/c=0.21 and D/L=0.5 .................................................................................................................. 198
Figure 205: Pressure distribution on center plane of complete model at H/c=0.26 and D/L=0.5 ................................................................. 199

Figure 206: Pressure distribution on center plane at H/c=0.26 and D/L=0.5................... 200

Figure 207: Pressure distribution on center plane at H/c=0.25 and D/L=0.5................. 200

Figure 208: Pressure distribution on center plane at H/c=0.24 and D/L=0.5............. 201

Figure 209: Pressure distribution on center plane at H/c=0.23 and D/L=0.5............. 201

Figure 210: Pressure distribution on center plane at H/c=0.22 and D/L=0.5............. 201

Figure 211: Center plane streamlines on velocity distribution at H/c=0.26 and D/L=0.5 .................................................................................................................. 202

Figure 212: Center plane streamlines around wing (a) at H/c=0.24 and D/L=0.5 (b) at H/c=0.24 in undisturbed flow................................................................. 203

Figure 213: Center plane streamlines around the wing at H/c=0.25 and D/L=0.5 ...... 203

Figure 214: Center plane streamlines around the wing at H/c=0.23 and D/L=0.5 ...... 204

Figure 215: Center plane streamlines around the wing at H/c=0.22 and D/L=0.5 ...... 204

Figure 216: Wake velocity distribution (a) Behind a wing, wing placed at H/c=0.25 and D/L=0.5 (b) Bluff body only at D/L=0.5 ................................................................. 205

Figure 217: Wake vortices (a) Behind a wing, wing placed at H/c=0.25 and D/L=0.5 (b) Bluff body only at D/L=0.5................................................................. 205

Figure 218: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.5................................................................. 206

Figure 219: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.5................................................................. 206

Figure 220: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.5................................................................. 207

Figure 221: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.5................................................................. 207

Figure 222: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.5................................................................. 208

Figure 223: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.5................................................................. 208
Figure 224: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.5 ......................................................... 208

Figure 225: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.5 ......................................................... 208

Figure 226: Streamlines around the model (a) bluff body only (b) a wing placed at H/c=0.22 and D/L=0.5 ................................................................. 209

Figure 227: Streamlines around the model with shown pressure distribution with wing at H/c=0.26 and D/L=0.5 (a) top view (b) 3D view ........................................... 210

Figure 228: Streamlines around the model with shown pressure distribution with wing at H/c=0.25 and D/L=0.5 (a) top view (b) 3D view ........................................... 211

Figure 229: Streamlines around the model with shown pressure distribution with wing at H/c=0.24 and D/L=0.5 (a) top view (b) 3D view ........................................... 211

Figure 230: Streamlines around the model with shown pressure distribution with wing at H/c=0.22 and D/L=0.5 (a) top view (b) 3D view ........................................... 212

Figure 231: Change in lift coefficient vs freestream case at various velocities .......... 215

Figure 232: Change in drag coefficient vs freestream case at various velocities ...... 217

Figure 233: Velocity distribution on center plane in wake region at D/L=0.1, H/c=0.22, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000 ................................................ 218

Figure 234: Velocity distribution on center plane around the wing at H/c=0.22, D/L=0.1, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000 ................................................ 219

Figure 235: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=600,000 .......... 220

Figure 236: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=1,200,000 .......... 220

Figure 237: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=1,200,000 .......... 220

Figure 238: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22, D/L=0.1 and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000 ........................................................................................................ 221

Figure 239: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22, D/L=0.1, and Re=600,000 .................................................. 222

Figure 240: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22, D/L=0.1, and Re=1,200,000 .................................................. 222
Figure 241: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.22 \), \( D/L = 0.1 \), and \( \text{Re} = 1,800,000 \) ............................................. 222

Figure 242: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.22 \), \( D/L = 0.1 \), and \( \text{Re} = 600,000 \) (a) top view (b) 3D view ................................. 223

Figure 243: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.22 \), \( D/L = 0.1 \), and \( \text{Re} = 1,200,000 \) (a) top view (b) 3D view ................................. 224

Figure 244: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.22 \), \( D/L = 0.1 \), and \( \text{Re} = 1,800,000 \) (a) top view (b) 3D view ................................. 224

Figure 245: Velocity distribution on center plane in wake region at \( D/L = 0.3 \), \( H/c = 0.25 \), and (a) \( \text{Re} = 600,000 \) (b) \( \text{Re} = 1,200,000 \) (c) \( \text{Re} = 1,800,000 \) .................................................. 226

Figure 246: Velocity distribution on center plane around the wing at \( H/c = 0.25 \), \( D/L = 0.3 \), and (a) \( \text{Re} = 600,000 \) (b) \( \text{Re} = 1,200,000 \) (c) \( \text{Re} = 1,800,000 \) .................................................. 227

Figure 247: Center plane streamlines at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 600,000 \) (a) Complete model (b) Around the wing.................................................. 228

Figure 248: Center plane streamlines at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,200,000 \) (a) Complete model (b) Around the wing.................................................. 228

Figure 249: Center plane streamlines at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,800,000 \) (a) Complete model (b) Around the wing.................................................. 228

Figure 250: Wake velocity distribution at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25 \), \( D/L = 0.3 \) and (a) \( \text{Re} = 600,000 \) (b) \( \text{Re} = 1,200,000 \) (c) \( \text{Re} = 1,800,000 \) .................................................. 229

Figure 251: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 600,000 \) .................................................. 230

Figure 252: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,200,000 \) .................................................. 230

Figure 253: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,800,000 \) .................................................. 230

Figure 254: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 600,000 \) (a) top view (b) 3D view ................................. 231

Figure 255: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,200,000 \) (a) top view (b) 3D view ................................. 232

Figure 256: Streamlines around the model with shown pressure distribution with wing at \( H/c = 0.25 \), \( D/L = 0.3 \), and \( \text{Re} = 1,800,000 \) (a) top view (b) 3D view ................................. 232
Figure 257: Velocity distribution on center plane in wake region at D/L=0.5, H/c=0.25, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

Figure 258: Velocity distribution on center plane around the wing at H/c=0.25, D/L=0.5, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

Figure 259: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=600,000
(a) Complete model (b) Around the wing

Figure 260: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=1,200,000
(a) Complete model (b) Around the wing

Figure 261: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=1,800,000
(a) Complete model (b) Around the wing

Figure 262: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.5 and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

Figure 263: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.5, and Re=600,000

Figure 264: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.5, and Re=1,200,000

Figure 265: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.5, and Re=1,800,000

Figure 266: Streamlines around the model with shown pressure distribution with wing at H/c=0.25, D/L=0.5, and Re=600,000 (a) top view (b) 3D view

Figure 267: Streamlines around the model with shown pressure distribution with wing at H/c=0.25, D/L=0.5, and Re=1,200,000 (a) top view (b) 3D view

Figure 268: Streamlines around the model with shown pressure distribution with wing at H/c=0.25, D/L=0.5, and Re=1,800,000 (a) top view (b) 3D view
List of Tables

Table 1: Phase 1 study parameters ................................................................. 36
Table 2: Phase 3 study parameters ................................................................. 38
Table 3: Study parameters phase 4 ................................................................. 40
Table 4: Physical models for freestream analysis ......................................... 91
Table 5: Parameter for velocity study ............................................................ 213
Key to Symbols

A – Planform area of the wing [m²]

AOA – Angle of Attack [degree°]

c – Chord length [mm]

Cₗ – Lift or downforce coefficient, measured in negative z direction, downforce direction

C₅ – Drag coefficient

D/c – A distance normalized by the wings chord length

D/L – Distance ratio to bluff body normalized by a car length

E – Energy

Fₜₙₗ₉ – Drag force [N]

Fₗᵢₙ – Lift or downforce [N]

g – Gravity constant [m×s²]

H/c – Ground clearance distance ratio normalized by the wings chord length

L – Car length [mm]

p – Pressure [Pa]

q – Heat flux [W/m²]

Re – Reynolds number

s – Wing span [mm]

t – time

T – Temperature [degree Celsius]

u – Velocity in x-direction [m/s]

Uᵢ – Freestream velocity [m/s]

v – Velocity in y-direction [m/s]
\( w \) – Velocity in z-direction [m/s]

\( \mu \) – Dynamic viscosity of a fluid [kg/m×s]

\( \rho \) – Density [kg/m\(^3\)]

\( \tau_{xx} \) – Stress component in x-direction

\( \tau_{yy} \) – Stress component in y-direction

\( \tau_{zz} \) – Stress component in z-direction

\( S_M \) – Sum of body forces
**Abbreviations**

AIAA – American Institute of Aeronautics and Astronautics

CFD – Computational Fluid Dynamic

EOS – Equation of State

ERCOFTAC – European Research Community On Flow, Turbulence And Combustion

FEA – Finite Element Analysis

FIA – Fédération Internationale de l’Automobile

RANS – Reynolds Averaged Navier Stokes

SAE – Society of Automotive Engineers

sdr – scalar dissipation rate

Tke – Turbulent kinetic energy
1 Introduction

Aerodynamics has become an important factor in recent race car design. The large amount of downforce produced by race cars allows higher cornering speeds. Despite recent gains in aerodynamics, still little knowledge is available in the literature. Formula 1 or Indy Car Teams may have great knowledge about the influence and behavior of race car wings; however, the aerodynamics of their car is a well-guarded secret. Small changes in the aerodynamics of the race car front wing can lead to a significant change in performance of a wing. In order to understand the effect of the aerodynamics on race car wings, it is important to understand what exactly a race car and its wings are designed to do. In the most basic way, a race car must exhibit maximum performance in the categories of acceleration, speed, deceleration, and cornering speeds (lateral acceleration), as these factors determine how quickly a car can race through a track. The wings have the function to improve the mentioned categories of the race car. Due to the lack of provided knowledge, this work should help to understand the behavior of a race car wing in disturbed flow (wake). This is mainly the case where one car is following another car. In recent years, the research in this field grew. Nevertheless, only few studies of race car wings with ground influence in disturbed flow have been carried out. Therefore, the understanding of the wing in a wake near ground is not completely provided.

1.1 Background

The principle of race car wings was borrowed from successful airplane wing designs from the mid-twentieth century. Due to the different nature of race cars and airplanes,
this approach was not very successful. Katz [1] summarizes his own findings of the difficulties as follows:

“A race car lifting surface design is different from a typical airplane wing design because (a) a race car’s front wing operates within strong ground effect, (b) open-wheel race car rear wings have very small aspect ratio, and (c) there are strong interactions between the wings and other vehicle components.” [1]

The term, race car wing, is related to the actual front and rear wing of a race car. In technical terms of race car regulations, the term, wing, is not used to describe the actual dimensions in the Fédération Internationale de l’Automobile (FIA) regulations of Formula 1. The front wing is specified within the chapters of “Bodywork around the front wheels” and “Front bodywork” [2]. However, Formula 1, or FIA racing categories are not the only race cars which use wings. Nevertheless, Formula 1, Indy Cars, and other FIA race categories are probably the most commonly known cars with front and rear wings. The actual wings have turned into very important aerodynamic features.

1.1.1 Function of Race Car Wings

The simplest description of the function of a race car wing is to improve the car’s performance. The main part is to produce downforce for increasing cornering speeds. According to Seljak [3], each wing produces about a third of the total car’s downforce. The function of race car wings changed over the years. During the time of introduction of the wings on race cars, the desired function of a wing was the principle of minimizing the drag while maximizing the downforce [4]. Today, both the front and rear wing exist by themselves, and additional aerodynamic features such as endplates and flaps
improve the performance of the wings. A front wing experiences normally more modifications to improve the race car’s performance than a rear wing. Since the front wing is the leading element of a race car, its commission is also to guide the incoming flow towards the body. Therefore, a variety of additional features such as Wing-Gurney flaps and endplates control the airflow around it [3].

**Formula 1 Technical Regulations**

As an example of the complex regulations for race car wings, or aerodynamics, a few examples of Fédération Internationale de l’Automobile regulations for 2015 are given here [2].

“**3.15 Aerodynamic influence:**

*With the exception of the ducts described in Article 11.4, any specific part of the car influence its aerodynamic performance:*

a) *Must comply with the rules relating to bodywork*

b) *Must be rigidly secured to the entirely sprung part of the car (rigidly secured means not having any degree of freedom).”*

“**3.11 Bodywork around the front wheels:**

**3.11.1** *With the exception of the air ducts described in Article 11.4 and the mirrors described in Article 3.8.1, in plain view, there must be no bodywork in the area formed by the intersection of the following lines:*

a) *A longitudinal line parallel to and 900mm from the car centre line.*

b) *A transverse line 450mm forward of the front wheel centre line.*
c) A diagonal line from 450mm forward of the front wheel centre line and 400mm from the car centre line to 750mm forward of the front wheel centre line and 250mm from the car centre line.

d) A transverse line 750mm forward of the front wheel centre line.

e) A longitudinal line parallel to and 165mm from the car centre line.

f) A diagonal line running forwards and inwards, from a point 875mm forward of the rear face of the cockpit entry template and 240mm from the car centre line, at an angle of 4.5° to the car centre line.

g) A diagonal line from 875mm forward of the rear face of the cockpit entry template and 240mm from the car centre line to 625mm forward of the rear face of the cockpit entry template and 415mm from the car centre line.

h) A transverse line 625mm forward of the rear face of the cockpit entry template.

For reference this area is shown in Drawing 17A in the Appendix to the Technical Regulations.

3.11.2 With the exception of the air ducts described in Article 11.4, in side view, there must be no bodywork in the area formed by two vertical lines, one 325mm behind the front wheel centre line, one 450mm ahead of the front wheel centre line, one diagonal line intersecting the vertical lines at 100mm and 135mm above the reference plane respectively, and one horizontal line on the reference plane.”

The technical regulations for Formula 1 cars are extremely precise and specific. Therefore, only trained race car engineers can really understand them in depth.
1.1.2 History of Race Car Wings

The fundamentals of aerodynamics were developed more than 200 years ago. However, the principle of using wings was not applied until the 1920s. The first cars with attached wings were Opel’s experimental rocked-powered cars, RAK 1 and RAK 2. Opel mounted the wings using the principle of airplanes wings between the two axes with a high negative angle of attack. A negative angle of attack produces negative lift, or downforce [1].

Although, the wings were a major invention, it took another 35 years to be realized as highly potential [1] [3]. The appearance of the GMC-supported 1965 Chaparral 2C with its adjustable pitch rear wing changed the shape of race cars from that day. Within the 1960s, the race car wings made huge progress and appeared all over in racing. The 1966 Chaparral 2E had his wing mounted high over the rear end of the car. In Formula 1, the wings first appeared during the 1968 Belgium grand prix with a fully inverted rear wing [3]. High mounted and adjustable rear wings were prohibited after resulting in several catastrophic failures [1].

The race car wing developed within a short period of time. The first additional feature appeared in 1971 in the form of a so-called gurney-flap. This perpendicular to the chord small flap attached at the trailing edge improved the wing significantly. In 1973, Ferrari started to avoid wing-body interaction by mounting their front wing quite far ahead of the car. The McLaren Formula 1 team was the first in 1984 which applied a multi element wing [3].
The wing design overcame a change in terms of controlling the flow around the car. Tyrell raised the car’s nose in 1990 to improve the flow conditions under the car. The regulations of Formula 1 were adjusted more after each catastrophic accident. For example, the changes were made after the Imola 1994 accident where Ayrton Senna died after a fatal crash. This was the turning point in terms of safety regulations. One new regulation was the implementation of a minimum ground clearance [3].

1.2 Goal of Study

The goal of study is to analyze the behavior of a race car front wing through a parametric study. In the analysis of the race car front wing, there are two major quantities of importance: the amount of downforce created and the amount of drag force. The effective parameters are angle of attack (AOA), ground clearance (H/c), and operating speed. The objective of this study is the understanding of the behavior of a race car wing during operation in disturbed air, which is referred to as the race car is operating in a wake, meaning following another car. This adds another parameter, the distance (D/L) to the leading car. All distances are normalized. The ground clearance is normalized by the chord length (c) of the wing and the distance to the leading car is normalized by a car length (L).

This Computational Fluid Dynamic (CFD) simulation study consists of 4 different phases. The first phase is building the wing model and preparing a benchmark solution for the study. The second phase will be used to build a bluff body and identify appropriate ranges of study. The third phase is analyzing the wing in ground effect
without any bluff body in front of the wing. The fourth phase is analyzing the wing in a wake.

Star CCM+, a commercial CFD code is used to carry out the numerical simulations. All the models are built in SolidWorks. All the simulations are steady state simulations. There would be two different approaches to simulate such a race car wing. First, the car and wing move on the ground and through quiescent air. All possible wind speeds are neglected in this study. However, it is hard to simulate a moving car and steady air in CFD. Therefore, the common approach is modeling the race car front wing and the bluff body as stationary components. Therefore, the surrounding air has a free stream velocity which is the car’s speed. Further, since the car is fixed, the ground has a relative velocity set equal to the speed of the car and wing. Therefore, the ground will be set up with the driving velocity, too. All simulations are with a symmetry plane to save computational resources.

The chosen wing profile is the S1223 designed by Michael Selig [5]. This is a high lift wing profile, which means a single element wing, with no flaps or slats can obtain an extremely high lift coefficient [6].

1.2.1 Phase 1 – A Wing in Freestream

Phase 1 is to build the wing model and compare the simulation results to the results in the literature. Selig’s S1223 airfoil is a common high lift profile. The profile is created with SolidWorks by importing the given data points¹. Figure 1 shows the setup for phase

¹ S1223 data points can be found in the appendix
1. A single wing profile gets analyzed on its behavior by changing the angle of attack. Table 1 shows the parameters for this phase. Reynolds number 250,000 corresponds to an actual velocity of 12.5 m/s with a characteristic length of 300 mm. This Reynolds number is chosen based on the available experimental results provided by Selig et al [5].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of Attack (AOA)</td>
<td>0°</td>
<td>18°</td>
</tr>
<tr>
<td>Chord length (c)</td>
<td>300 mm</td>
<td></td>
</tr>
<tr>
<td>Wing span (s)</td>
<td>1600 mm</td>
<td></td>
</tr>
<tr>
<td>Reynolds number</td>
<td>250,000</td>
<td></td>
</tr>
</tbody>
</table>

Figure 1: Phase 1 model

1.2.2 Phase 2 – Creation of a Bluff Body

Phase 2 is to determine suitable ranges for the actual study in a wake, phase 4. A simple bluff body is created to generate a race car wake. Wilson et. al. [7] showed that a
simple bluff body is enough since the main wake characteristics are produced by a simple body and the rear wing. The bluff body is analyzed at a racing speed of 30 m/s. The bluff body consists of a body, wheels and rear wings as shown in Figure 2. The maximal width of the bluff body including the wheels is set to be 1620 mm. The length between front and rear wing support is 1500 mm. The height is set to be 800 mm. The wheel diameter is 660 mm, which refer to the actual tire diameter of a Formula 1 tire, defined by the FIA rules [2].

Figure 2: Bluff body

1.2.3 Phase 3 – A Wing Operating in Ground Effect

Phase 3 is a study of the S1223 wing profile, which will also be used as a benchmark study. The behavior of the wing is now studied in ground effect with one changing parameter, the ground clearance H/c. The ground clearance is measured between
ground and leading edge point and is normalized by the chord length. The speed, 30 m/s, is chosen corresponding to cornering speeds existing in Formula 1. A 6 \degree angle of attack is chosen. Literature review shows that race car front wings operate with small angles of attack, depending on wing profile. Figure 3 and Table 2 show the actual parameters set in the model accordingly.

Table 2: Phase 3 study parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ground clearance (H/c)</td>
<td>0.15</td>
<td>0.5</td>
</tr>
<tr>
<td>Angle of Attack (AOA)</td>
<td>6\degree</td>
<td></td>
</tr>
<tr>
<td>Chord length (c)</td>
<td>300 mm</td>
<td></td>
</tr>
<tr>
<td>Wing span (s)</td>
<td>1600 mm</td>
<td></td>
</tr>
<tr>
<td>Reynolds number</td>
<td>600,000</td>
<td></td>
</tr>
</tbody>
</table>

Figure 3: Phase 3 model
1.2.4 Phase 4 – A Wing Operating in a Wake

The parameters which will be studied during phase 4 are similar to those in phase 3. However, the study is executed in disturbed flow. The wake will be produced with a bluff body and a rear wing. Wilson et al [7] showed that the wake of a Formula 1 race car can be modeled accurately with a simplified body and wing. The airfoil profile is again Selig’s high lift wing S1223. The ground clearance is measured between the ground and the leading edge point and is normalized by the chord length. The distance between front wing and the bluff body is measured from the rear end of the bluff body and leading edge and normalized by the maximum car length from the Formula 1 race car.
As reference length, the Sauber C34-Ferrari is used with its length of 5,300 mm [8]. The speeds will be chosen similar to the phase 3 study, 30 m/s.

The distance ratio between the bluff body and the leading edge of the front wing is chosen to be D/L = 0.1 to 0.5. Figure 4 shows the wake size of the bluff body and illustrates the points where the wing will be placed for the study. The results of phase two show that the wake from the bluff body weakens significantly after a distance D/L = 0.2. The results in phase two also showed that the main body wake impact weakens quicker than the wing tip vortices produced by the modeled rear wing. The wing tip vortices are still strong after a distance of D/L = 0.5. The biggest change on the wing operating in a wake is expected to happen between the distance D/L = 0.3 to D/L = 0.5.
Further, the limitation on computational resources do not support distances D/L bigger than 0.5. Figure 5 and Table 3 show the setup and parameters used.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ground clearance (H/c)</td>
<td>0.15</td>
<td>0.4</td>
</tr>
<tr>
<td>Distance ratio (D/L)</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>Angle of Attack (A)</td>
<td>6°</td>
<td></td>
</tr>
<tr>
<td>Chord length (c)</td>
<td>300 mm</td>
<td></td>
</tr>
<tr>
<td>Wing span (s)</td>
<td>1600 mm</td>
<td></td>
</tr>
<tr>
<td>Reynolds number</td>
<td>600,000</td>
<td></td>
</tr>
</tbody>
</table>
In addition, a small study is done on a higher speed to see how different speeds affect the operational behavior of a race car wing. Therefore, the speed gets increased to 60 m/s. This corresponds to a Reynolds number of 1,200,000.
2 Literature Review

In the 1990s, studies on actual race car wings were started. As of today, multiple studies of race car wings exist in various parameter configurations such as undisturbed and disturbed flow, with and without ground effect and different wind profiles. An overview of the different studies and their main parameters are given in the appendix.

2.1 Wing Profiles

Hundreds of different possible wing profiles exist. However, this includes wings from airplanes, wind turbines, and racecars. Since not all of them are suitable for race cars, it can be seen that some of the profiles are actually very common on race cars. For example, several studies include the wing profiles Tyrrell026, LS(1)-0417, or S1223. A variety of NACA profiles were found as well. The aerodynamics of race care front wings are especially crucial. Not only because approximately 30 % of the downforce is created by the front wing, the front wing also defines the flow around the rest of the car’s aerodynamic components [9].

The S1223, displayed in Figure 6, is a cambered airfoil designed by Michael S. Selig. It is a high lift low Reynolds number airfoil with maximum thickness of 12.1 % at 19.8 % chord length. The maximum camber is 8.1 % at 49 % chord length [10].

Figure 6: S1223 airfoil geometry
2.2 Single Element Front Wings

Single element wings were studied first. The studies can be divided into two main categories. The first are numerical studies based on different commercial CFD programs, and the second category is based on experimental data. Single element front wings contribute a lot to the understanding of the flow around race car wings. The numerical studies of the front wing itself are limited to undisturbed flow. Few experimental studies have been carried out simulating the wing in disturbed flow. Ranzenbach and Barlow studied the ground influence numerically and experimentally in a series of two dimensional studies. The NACA 0015 [11] and NACA 4412 [12] airfoils are studied as single element wings. Both research categories have their well-known advantages and disadvantages.

2.2.1 Numerical Studies in Undisturbed Flow

Numerical studies of single element race car front wings are one of the foundations in this research area. Most of the numerical studies are performed on the wing itself. However, some studies include having an endplate which has an influence on the wing’s performance. The background of endplates is discussed in Section 2.4. The numerical studies of single element airfoil can be categorized as studies with ground influence and studies without ground influence. Both categories rely on undisturbed flow simulation.
2.2.1.1 Numerical Studies without Ground Influence

Numerical studies without ground influence are not very common for race car front wings since a race car front wing is in rudimentary expression an inverted wing. However, as Katz [1] highlighted early, race car front wings operate in strong ground effect. Gopalarathnam and Selig [13], as well as Pakkam [14] used numerical studies without ground influence for wing design. Mokhtar used XFOIL, a panel method code for a primary study to determine the effective ranges of angle of attack and Reynolds number. His findings include the effective range for angle of attack lies between 6° and 12°. Mokhtar [15] also showed that the Reynolds number has the least effect of all parameters on the wing performance. By investigating the different studies, if a front wing is studied without ground influence, then it is only for its effective range and the overall wing behavior but not for race cars explicitly.

2.2.1.2 Numerical Studies with Ground Influence

Ground influence is highlighted all over as a major parameter for race car front wings. The significant results include the pressure distribution underneath a wing and the effect of downforce and drag. Kiffer et al. [16] studied the influence of angle of attack and ground effect on a Formula Mazda wing. It is reported that the ground clearance has significant influence on the downforce production. In dependency of angle of attack, it is shown that the downforce increases about 20 % from an angle of attack of 0° to 12°. The Mazda race car wing starts with stall conditions at about 12° angle of attack. The drag is increasing by about 50% at 12° compared to 0°. Ranzenbach and Barlow [11] used Reynolds Averaged Navier Stokes to study the NACA0015 profile as a numerical
study in addition to their experimental study. Their findings include that downforce is a function of ground clearance and increases with decreasing ground clearance. Further, the drag increases with decreasing ground clearance. They also found that large separation occurs on the suction surface of the wing at small ground clearance.

Mokhtar [15] studied the influence of ground clearance on four airfoil sections, the S1223, E423, LNV109A, and NACA9315. All of the airfoil sections have a similar behavior for downforce and drag. Large ground clearance does not detect most of the effect on the airfoil. The downforce increases with decreasing ground clearance, and the downforce remains more or less constant for a ground clearance bigger than height to chord ratio $H/c = 0.6$. The drag increases with decreasing ground clearance. However, the drag is way more influenced by the ground clearance than the downforce. The effect weakens with increasing ground clearance; however, it never gets steady like the downforce. The effective range of ground clearance does not get influenced by endplates. Mokhtar [17] analyzed the flow around a wing and showed the changes of pressure and velocity which are the reason for the downforce and drag increase. A study of a symmetric airfoil, the NACA0012, shows that the generated downforce reaches its maximum at a ground clearance of 10% of the wings’ chord length. With decreasing ground clearance under 10% the downforce decreases significantly. The reduction from 0.1 to 0.09 is observed to be 3.8% whereas the decrease between 0.06 and 0.05 is 57%. The drag increases at an almost constant rate as the ground clearance decreases with its peak at 0.08. The lift over drag ratio increases as the $H/c$ increases. Although the lift over drag ratio increases, it is not a ratio race car designers are very interested in; it is more the magnitude of the actual forces. The study showed
the influence in terms of flow characteristics and pressure distribution. For both, the upper surface of the wing is less influenced by the ground clearance than the lower surface [18].

Price [19] simulated a FC 63-137 front wing on a SAE race car. He reported that the suction peak moves backwards in ground effect compared to free stream case. The suction peak at ground clearance $H/c = 0.1$ is for the pressure coefficient 278 % higher than in free stream. Further, Price [19] showed that the vortices on the wing tips have a negative influence. His study did not include endplate. He reported a negative effect of the wake on the wheels and pointed out that endplates are used to redirect the air around the tire.

### 2.2.2 Numerical Studies in Disturbed Flow

Numerical single wing studies in disturbed flow have not been carried out. However, Wilson et al [7] used FloWorks CFD solver to evaluate the fluid mechanics of a bluff body model of a Formula 1 race car. From preceding results, the goal of the bluff body is to generate strong streamwise vortices superimposed onto relatively low velocity and high turbulent wake. The rear wing and rear wheels were kept in the bluff body model, but the chassis was dramatically shortened. The forward part of the chassis was replaced by a semi-circular nose. No computation results of the bluff body are provided. However, a moving ground and rotating wheels were used to simulate it. The simple bluff body showed remarkably similar results in the experimental testing which are explained in more detail in Section 2.2.4.
2.2.3 Experimental Studies in Undisturbed Flow with Ground Influence

Experimental studies are building the other pillar of single wing airfoil studies. Ranzenbach and Barlow started in the 1990’s with experimental studies which were conducted by numerous other researchers. Experimental studies without ground influence are not really associated with race car wings as Katz [1] stated early. Zhang and Zerihan [20] [21] did a lot on research in that field. Experimental studies with ground influence show similar results like the numerical studies.

Experimental studies show that with decreasing ground clearance, the downforce increases. Very small ground clearance has a negative impact on the downforce. Ranzenbach and Barlow [12] measured the critical height for the NACA 0015 at 0.0361 at an angle of attack of zero degrees and a freestream Reynolds number of 1.5 million. The observation was that with too small of a ground clearance, the boundary layer distance between airfoil and ground approaches zero which causes the negative impact. Zerihan and Zhang [20] used in their study a reference incidence of one degree angle of attack on a Thyrell026 wing. The physical effect of the ground is to constrain the airflow over the lower surface of the wing. This causes the flow to accelerate compared to cases without ground clearance and results in a negative pressure, or suction, which results in higher downforce. The maximum downforce was found to be at a ground clearance of $H/c = 0.08$ and results in a lift coefficient of 1.72 at a speed of 30 m/s. Ground clearance smaller than 0.08 reduced the downforce significantly. An investigation of the Reynolds number showed that the lift coefficient versus ground clearance follow the same trend. The main difference is that the slower speed, 20 m/s, results in a higher downforce. The maximum downforce occurs at $H/c = 0.08$, but the lift
coefficient increased to 1.77. The reason for this difference is believed to be due to the larger separation region at lower speed. This separation contributes to the increment in downforce. The drag is reported to increase with decreasing ground clearance. Fixing the ground is decreasing the downforce significantly [20].

Zerihan and Zhang [20] studied the influence of transition free versus transition fixed wings. This study is done on a Thyrell026 wing. Transition fixed refers to a fixed point to trip the boundary layer from laminar to turbulent flow. A marked difference was found between the two cases as illustrated in Figure 7. Transition fixing reduces the lift coefficient $C_{L_{max}}$ from 1.72 to 1.15. The corresponding increase in downforce from freestream to ground influence is 141 % downforce increase for transition free and 64 % increase for transition fixed. The level of ground clearance with the maximum downforce increases from $H/c = 0.08$ for transition free to 0.14 for transition fixed. Transition fixing causes a thicker boundary layer and makes separation more likely to occur even at a higher ride height. This results in a significant loss of downforce.
2.2.4 Experimental Studies in a Wake

An experimental study done by Soso and Wilson [22] shows the behavior of a single wing airfoil in different wake conditions. The conditions are free stream (FC1), wake of a rear wing model (FC2) and wake of a bluff body with a rear wing (FC3). The studied front wing was idealized as a single element LS(1)-0417 wing with a constant angle of attack of 5 degrees. The analysis of the downforce showed that with a smaller ground clearance, less downforce was lost. The downforce loss at a ground clearance was found to be 33% at a ground clearance of $H/c = 0.833$ and 18% at $H/c = 0.204$. The downforce curves follow the normal trend, which was found at undisturbed flow. However, at very small ground clearance, the downforce increases abruptly.
The drag coefficient indicates a drag increase of the wing in disturbed flow. At ground clearance above $H/C = 0.4$, the bluff body with a rear wing generated the most drag whereas at smaller ground clearance, the model with only a wing generated the most drag. It is also shown that the wing in disturbed flow generates less downforce than a wing in undisturbed flow. However, the stall condition of the wing gets altered in disturbed flow. For the tested wing profile, an abrupt stall occurred at 23 degree angle of attack whereas the disturbed flow became more gradual. The conclusion is that this result occurs because the boundary layer characteristics of the wing could be altered in disturbed flow. A lateral movement of the wing was studied, and the result showed the closer the wing gets to the bluff body, the more downforce gets lost. As the bluff body gets moved away, the downforce recovers to the free stream value [22].
Wilson et al [7] showed that with a short bluff body containing the main element of the rear of a race car, the main wake stays similar, quantitative and qualitative. The velocity distribution on a plane behind the bluff body has a kind of a mushroom shape. The same “mushroom” wake can be seen on the simplified bluff body, and the vortex core is identically positioned relative to the projected car. However, the study shows that the wake is not a perfect representation but fairly close. Nevertheless, a simple bluff body is able to represent the main features of the wake.

The study of lift and drag on the wing downstream the wake are based on reference velocity that equals to the equivalent speed of the car ahead. The absolute magnitude of downforce and drag are smaller compared to the free stream case. The stall condition of the wing is delayed by approximately 5 % in the wake at all different ground clearances tested. The increase of stall angle can be either due to the high turbulence of the wake or the change of the true angle of attack, which means that the air already hits the wing at a certain angle. This reduces the angle between flow and wing. Therefore, the true angle of attack is different as it would be in freestream condition [7].

2.3 Multiple Element Front Wings

Few studies have been carried out for multiple element wings. Katz et al [23] studied numerically a generic Indy car with a multi-element front wing. Zhang and Zerihan [24] studied a double-element wing with ground effect and moving ground experimentally. A double-element wing with fixed ground was tested in a wind tunnel by Jasinski and Selig [25].
2.3.1 Numerical Study in Undisturbed Flow

The viscous flow simulation of Katz et al [23] shows the flow around a generic Indy race car. The multi-element front wing shows clearly different application of wing elements. Generally known, the wing produces downforce. However, the race car wing design is not a steady design. The middle section provides the ability to let more air underneath the car to improve the downforce of the body. The flaps on the multiple wings also redirect the air in certain ways. It can be seen that the stream lines clearly designed to hit the cooling duct. The study showed that numerical simulations are great tool to capture the flow structure.

2.3.2 Experimental Studies with moving Ground Influence

Zhang and Zerihan [24] studied a multiple element wing in ground influence with a moving ground. The main element is a modified General Aviation-Whitcomb (GAW) airfoil. The study shows that the main characteristic of a double element wing is similar to the single element wing. A high and a low flap angle are tested for the second element. The maximum downforce occurs at a ground clearance of $H/c=0.066$ for the low flap angle and at $H/c = 0.079$ for the high flap angle. Figure 9 shows the behavior of the downforce for high and low flap angle. Region c shows the region for ground clearance smaller as the maximum occurrence. Both flap angles show a transition from region a to region b. Where in region a, the downforce curve has a high gradient which turns into a small gradient at the beginning of region b. The high flap angle produces significantly greater downforce at larger ground clearance than the low flap angle. With decreasing ground clearance, the difference gets smaller.
The study shows that the main element produces most of the downforce and dominates the turbulent wake development. However, the wake for the high flap angle was found to be bigger than the low flap angle case. The high flap angle case shows a sharp reduction after reaching the maximum downforce because of the boundary layer separation. It can be seen that the maximum downforce point in terms of ground clearance is lower for the low angle flap than at a single element wing. The high angle flap maximum occurs just slightly lower since single element wings reach their maximum at a ground clearance of approximately $H/c = 0.08$ to $0.09$. 

Figure 9: Downforce vs. ground clearance for double-element wing [24]
2.3.3 Experimental Studies with fixed Ground Influence

Jasinski and Selig [25] studied a multiple element half span front wing in a wind tunnel with fixed ground. The represented data were taken at a ground clearance of H/c = 0.3. The model for the performed test is based on the UIUC700 two-element airfoil. The different configurations and test included a variation of endplate design and flap design. The tested parameters are angle of attack, speed, and flap deflection angle. The study of using different Reynolds number showed that with increasing Reynolds number, the lift coefficient increases and the drag decreases. While increasing the Reynolds number from $0.7 \times 10^6$ to $1.1 \times 10^6$ the average lift increase is 2.5 % whereas the drag decreases of 2.3 %. The angle of attack behaves as shown in other studies; the stall condition was reached within the ranges of 15 to 17 degrees depending on the speed. The flap deflection study shows that a change of the flap deflection by 10 degrees at a constant angle of attack leads to an average increase of the lift coefficient by 0.5. It can be seen that there is no appreciable change in overall drag with changing flap deflection (Figure 10), which states that the overall drag characteristics are dominated by induced effects. By increasing the flap deflection, the trailing vortices move closer to the root. This has to do with competing effects. First, the increase of the lift at high flap deflection will cause more drag. However, more flow might be forced through underbody because of the vortex. This will increase the downforce of the body. By introducing endplates, the lift coefficient increases by an average of 0.0958 at constant angle of attack, while drag coefficient at constant lift coefficient decreases by an average of 13.7%.
Influence of Endplates

Endplates on race car front wings affect the flow characteristics of the wing. The overall characteristics are reported to be nearly the same by Mokhtar [17]. Endplates weaken the wing tip vortices, and the wake is less deformed compared to wings without endplates. Downforce and drag follow a similar trend for wings with and without endplates, and their dependency on the ground clearance is similar. The drag has a slightly larger effective range of dependency on the ground clearance on a wing with endplates. Price [19] showed that the endplates are also important to control the direction of the flow. On his SAE Formula race car, he reported a negative influence on the tires without wing tips which he thinks could be avoided by guiding the air around the tire. Katz [1] reported that removing endplates on rear wings causes loss of lift but the no stall characteristics remains.
Gogel and Sakuri [26] studied the effect of end plates on downforce in yaw. A single element wing was used in the Toyota Atlantic series designed by Swift Engineering. Six different designs of endplates were studied numerically. The baseline design, which is basically a flat rectangular plate, is compared to the five other designs at a yaw angle of 20 degrees. The baseline case had a decrease in downforce of 9.63% from 0 degrees to 20 degrees yaw. This loss is explained by the reduction of flow on the windward side of the wing. Each tested design had a positive impact on the downforce loss. The most effective design tested were one-way holes through the end plate on the first half of the wing chord. The downforce decrease from the baseline could be minimized to 6.39% compared to the baseline case. However, the studied optimizations of the endplate design for a standalone rear wing are only general representations.

### 2.5 Gurney Wing Flap

Dan Gurney introduced his Gurney wing flap in the 1970s on race cars. The cars equipped with this flap modification achieved a dramatic improvement through increased cornering speeds. The Gurney flap is a thin, narrow plate positioned at the trailing edge perpendicular to the chord plane of the wing on the pressure side. Nikolic [27] studied the effect of the Gurney flap on the wake. It appears that the flap affects at least the near-field wing vortex wake. Marked differences were found in the wake vortex rollup patterns from a Gurney flap equipped wing versus a clean wing. A wing with a Gurney flap over the full span still has the classical trailing vortex as a clean wing. However, there are unorganized vortex structures between wing tip and centerline. The flap increases the strength of the tip vortices and hinders the usual spanwise flow at the
trailing edge, affecting the vortex roll-up process. Zhang and Zerihan [28] studied the Gurney flap in ground influence. The effect of the flap is similar in near ground as in free stream. A small Gurney flap increases the downforce disproportionately more than a large one. Reducing the flap height for fully attached flow shows that the behavior of the Gurney flap is similar to the angle of attack. With increasing flap, the downforce increases. However, the increase of downforce in ground effect can be twice as much as in free stream because the onset of flow separation causes a sharper stall in ground effect.

### 2.6Studies of Rear Wing

Kieffer et al [16] studied the behavior of the rear wing of a Formula Mazda race car in a computational study. The study shows that the downforce increases with increasing angle of attack. The peak gets reached for an angle of attack of 12 degrees. Meanwhile, the drag coefficient increases with increasing angle of attack. The study shows that the stall condition of the rear wing starts at an approximately 8 degree angle of attack. By increasing the angle of attack, the low pressure area near the trailing edge causes the drag increases. As the angle of attack gets bigger, 12 – 16 degrees, the high pressure area on the upper surface increases and moves towards the leading edge. This movement causes an increase in drag. Further, at a 12 degree angle of attack, the highest lift coefficient, the flow starts to separate from the airfoil which causes a negative impact.

Katz et al [23] showed that the streamlines along a symmetry plane on a generic Indy race car are not leaving the trailing edge of the rear wing parallel to the flap because of
local separation. This separation is common due to high angle of attack to produce more downforce.

3 Methodology

This is a computational fluid dynamic (CFD) study; therefore, the main methodology is the theory of CFD. Further, this section will state the generally used equations.

3.1 Theoretical Equations

In this section, the most important equation used in this study are stated such as Reynolds number, aerodynamic forces, and other used equations.

3.1.1 Reynolds Number

The Reynolds number is a dimensionless quantity expressing the ratio of inertia forces to viscous forces in the flow which is used to compare different flow patterns. It is an important parameter in fluid flows. If a similar Reynolds number is given for geometrical similar bodies in all respects, achieved results can be compared since they might only differ in geometric scale and speed. This is true even for different fluids. The Reynolds number is calculated the following way as described by Houghton [29]:

\[ Re = \frac{\rho U_\infty c}{\mu} \]  

where, \( c \) = chord length, \( U_\infty \) = freestream velocity, \( \rho \) = fluid density, and \( \mu \) = dynamic viscosity of the fluid.
3.1.2 Lift or Downforce Coefficient

The lift coefficient is another dimensionless quantity which represents the lift force. In race car aerodynamics, the lift coefficient is also known as the downforce coefficient. In terms of race car aerodynamics, the term downforce is common. However, depending on the author, either downforce or lift coefficient is used, but both coefficients are defined into the negative Z direction, which means that the downforce has an actual positive number. The lift coefficient is a force normalized by the wings’ area and dynamic pressure \( \frac{1}{2} \rho U_{\infty}^2 \). This allows to compare results independently from size and speed. The lift or downforce coefficient is defined as follows [29]:

\[
C_L = \frac{F_{\text{lift}}}{\frac{1}{2} \rho U_{\infty}^2 A}
\]

where, \( F_{\text{lift}} \) = the aerodynamic force, \( U_{\infty} \) = the freestream velocity, \( \rho \) = the fluid density, and \( A \) = the planform area of the wing.

3.1.3 Drag Coefficient

Similar to the lift coefficient, the drag coefficient is a dimensionless quantity which indicates the drag of the wing. The drag coefficient is a force normalized by the wings’ area and dynamic pressure. This allows to compare results independently from size and speed. The drag coefficient is defined as follows [29]:

\[
C_D = \frac{F_{\text{drag}}}{\frac{1}{2} \rho U_{\infty}^2 A}
\]
where, \( F_{\text{drag}} \) = the aerodynamic force, \( U_\infty \) = the freestream velocity, \( \rho \) = the fluid density, and \( A \) = the planform area of the wing.

### 3.1.4 Percentage Change of a Quantity

Throughout this study, various changes of aerodynamic forces or speeds are stated as a change compared to the baseline case. This way, the changes can be tracked really well.

\[
\Delta X\% = \frac{X - X_{\text{base}}}{X_{\text{base}}} \times 100\%
\]

where, \( X \) = the comparing quantity, \( X_{\text{base}} \) = the baseline value

### 3.2 CFD Modeling

Computational fluid dynamics (CFD) is the mathematical simulation of flow based on the governing equations, turbulence models and different types of solvers. CFD codes are numerical algorithms to solve fluid flow problems. A CFD program is based on three main modules, the pre-processor, the solver, and post processor. The pre-processor contains all the input selections which are used to solve a fluid problem; the solver contains the numerical solver which performs the integration of the governing equations, Navier-Stokes equation and turbulence models. The post processor performs the analysis of the calculated flow problem in terms of visualization, force calculation, particle tracking, and many more.
3.2.1 Pre-Processor

The pre-processor transforms the user input into a mathematically solvable problem. The different stages here include the definition of the calculating domain, grid or mesh generation and selection of physical models. The implementation of the domain, the CAD model can be loaded into the CFD program as an IGES, parasolid, step, or other types of surface files.

3.2.1.1 Mesh Generation

The mesh generation can simply be stated as dividing the domain into non-overlapping smaller sub-domains, also known as cells or control volumes. Meshing is an important part of the CFD simulation. A correct mesh, or the selection of a mesh type can influence the accuracy of the simulation. It is not only a question about the cell type; it is also a question about structured or unstructured mesh. In very basic description, in a structured grid, the grid lines pass through the whole domain whereas the unstructured grid may not have a physical relation between the cells.

Structured Mesh

A structured or Cartesian grid follows the following arrangement:

- Grid points are placed at the intersections of co-ordinates lines
- Interior grid points have a fixed number of neighboring grid points
- Grid points can be mapped into a matrix; their location in the grid structure and in the matrix is given by indices.
Structured curvilinear grids or body-fitted grids are based on mapping of the flow domain onto a computational domain with simple shape. Structured grid deals effectively with simple shapes, or the domain has to be divided into sub-regions. However, for more complex geometries, the block structured grids are considered to be more flexible than Cartesian or body-fitted grids. In a block-structured grid, the one Star CCM+ uses, the domain is sub-divided into regions. This allows to refine the mesh where greater resolution is needed. Each of the individual sub-regions can have its own coordinate system, so that the mesh can be more flexible. The interface of adjacent blocks may have grids on either side that are matching or non-matching, but, either way, they must be properly treated in a fully conservative manner. Block-structured grids with overlapping regions are called composite grids or chimera grids. The resulting grid structure combines the advantage of Cartesian grids – easy to generate, equations simple to discretize and solve with the ability of curvilinear grids to accommodate curved complex boundaries. The block-structured mesh come in three basic varieties: H-grids, O-grids, and C-grids. O-grids wrap around a circle and the last point matches the first one, the outcome will be a grid in form of the letter O. The example of a C-grid, which is used in the simulation of a 2D wing, has a rounded input edge and looks roughly like the letter C. H-grids are basically everything which are not O-, or C-grids.

**Unstructured Mesh**

Unstructured meshes are normally used for very complex geometries. The advantage of an unstructured grid is that no implicit structure of coordinate lines is imposed by the grid. Hence, the grid can be concentrated where necessary without wasting computer storage. Moreover, the control volumes or cells can have any shape. Such a grid is not
limited to one type of cell and would be called a hybrid mesh. The most attractive feature is that an unstructured mesh allows the calculation of flows in or around geometrical features of arbitrary complexity without spending a lot of time in meshing.

3.2.1.2 Boundary Conditions

CFD problems are defined as of initial and boundary conditions. Initial conditions help for faster convergence but should not affect the final solution for steady simulations. However, the boundary conditions are the fixed end on a model. Possible boundary conditions could be walls, inlet, outlet, symmetry, or periodicity to name the most common ones. A boundary condition describes the flow at a certain point and takes off some of the unknowns in the equation.

For example, the wall is the most common boundary condition in fluid problems. To show how the wall is defined as boundary, it is assumed that a solid wall is parallel to the x-axis. The no-slip condition of the wall defines the velocities $u$ (x-direction) and $v$ (y-direction) to be zero. The normal velocity in the first cell of the boundary condition wall can be set to zero and the one after that without any modification. Since the velocity at the wall is known, a pressure correction is not necessary. Pressure corrector method is described in Section 3.2.1.5.2 and referred to as SIMPLE algorithm.

A detailed description of how boundary conditions are implemented into the solver is presented in the book entitled *An Introduction to Computational Fluid Dynamics* by Versteeg and Malalasekera [30].
3.2.1.3 Physics

The physics’ continuum is the definition of the flow which includes the physical property of the fluid and the mathematical models to solve it. An example of fluid properties of air is given here. In the case of simulating a race car front wing, 14 physical models are used in this study. These physical inputs can be divided into two groups, the fluid physics and the overall physics model.

3.2.1.3.1 Overall Physical Models

For the overall physical models, four different options are available. A CFD user can choose between a normal three dimensional model, a shell three dimensional model, a two dimensional model, or axisymmetric model. Most common used is the normal three dimensional model. However, for a rotor or turbine, an axisymmetric model would save a lot of computational resources by having just a small part of the rotor and extend it with the axisymmetric model.

3.2.1.3.2 Time Modeling

CFD codes are able to solve time dependent problems. However, there are four different time models within the Star CCM+ CFD code. First, the most common used one is steady. This means there are no changes with time within the simulation. This works for both coupled and segregated flow models, which are described later on. There are also implicit and explicit unsteady models. Both models allow calculated time steps. Implicit unsteady time model solves the whole domain for each time step. It also
allows the use of coupled or segregated flow model whereas the explicit unsteady model only works for coupled flow. The explicit model marches in time and space at the same time but is only compatible for inviscid or laminar flow. The fourth model is the harmonic balance model which solves periodic flow. The model solves the unsteady flow as a repeated steady case.

### 3.2.1.3.3 Motion Modeling

Motion modeling includes six different models. The most common one is stationary because motion modeling is expensive in terms of computational time. Stationary will have no kind of motion between the parts of the model. However, a rotational axis can still be added to a part or the model. The more advanced settings include moving reference frame model, rigid body motion, morpher, or 6-Degree of Freedom. The moving reference frame model is used for steady state cases which has motion with constant rotation or translation such as fans, turbomachinery, or mixers. The rigid body motion is used for unsteady simulations. However, there will be the need for sliding meshes. The morpher model allows some of the mesh points to move based on the solution. This includes an auto re-meshing tool which calculates the mesh new after the motion. This model is limited to three dimensional unsteady flow only. Last, the 6-Degree of Freedom model is used for rigid body simulations as a response to pressure or shear forces. These forces can be generated by the fluid or predefined by the user.
3.2.1.3.4 Segregated and Coupled Flow Models

There are two flow models available to solve fluids. It can be either chosen to be the segregated flow model or the coupled flow model. Coupled flow model is suitable for all ranges of flow, from incompressible to supersonic. The calculation time is linear for this model with the number of cells in the domain. This model solves the conservation equation for mass and momentum simultaneously. The coupled energy flow model extension includes the energy equation, too. The coupled energy flow model is used where heat transfer is considered and is robust in solving compressible flow. Its conversion rate is independent from the speed setting.

The segregated flow model is suitable for low speeds and incompressible flow. However, it may also work to solve compressible flow. This model solves the flow equations in a segregated manner, which means one equation for each component of velocity and one for the pressure. The momentum and continuity equations get linked through a pressure-corrector approach, also known as SIMPLE method, described in Section 3.2.1.5.2. The model is not suitable for high speeds and cases which includes natural convection. The segregated flow model has three energy extensions. The segregated fluid enthalpy is used for cases which include combustion. Segregated fluid temperature model is normally taken and suitable for all cases which do not include combustion. The third model, segregated fluid isothermal is used in cases with constant temperature. The first two extensions solve the total energy equation in a continuum using the segregated formulation. Segregated fluid isothermal model uses a constant setting for the temperature.
3.2.1.3.5 Fluid Physics

The fluid physics define the fluid. Most commercial CFD codes have the ability to solve solids and multicomponent material. For this study, the fluid will be limited to gas, respectively air, which is a default setting in Star CCM+.

Gas as fluid offers five different models for gas. These five models are Constant Density, Ideal Gas, Polynomial Density, Real Gas, or User Defined Equation of State (EOS). Each of these models add equations or have constant settings. Ideal gas calculations are sufficient for most applications where flow structure is the key analysis.

3.2.1.3.6 Solver

A CFD solver is a complex mathematical solving algorithm. It is based on equation models to describe a flow. Since the flow has for each cell five governing equations with five unknowns, it is impossible to calculate them directly. Therefore, an iterative process is used, mostly the finite volume method.

3.2.1.4 Equations

A flow can be described mathematically through the Navier-Stokes equations. However, there is until today no real way to solve those equations except through an iterative process. The governing equations which build the conservation laws are used and coupled with the turbulence equation models to solve computational fluid dynamics problems.
3.2.1.4.1 Governing Equations

The governing equations build the mathematical representation of fluid flow and state the conservation laws of physics. The following conservation equations state: the mass of fluid is conserved, the rate of momentum change equals the sum of forces on a fluid particle, and the rate of energy change is equal to the addition of heat and work done.

**Conservation of Mass**

The conservation of mass equation states that the rate of increase of mass in a fluid element is equal to the net rate of flow of mass into the fluid element. Flow which is directed into the fluid element increases the mass of the element, and the flow decreasing the mass of the fluid element is directed out of the particle as shown in Figure 11.

![Figure 11: Mass flow in and out of a fluid element [30]](image)

Combining all the terms stated in Figure 11 with the rate of change over time states the conservation of mass equation. This equation for unsteady, three-dimensional mass...
conversation is also known as the continuity equation \([30]\) at a point in a compressible flow.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\]  

(5)

For steady cases, the first term will disappear. If the fluid is defined to be incompressible, the density \(\rho\) will be a constant.

**Conservation of Momentum**

The momentum conservation equation is based on Newton’s second law, which states that the rate of change of momentum of a fluid is equal to the sum of forces acting on it. The conservation of momentum cannot be combined into a single equation. The different forces acting on a particle are surface forces and body forces. It is common practice to include the effect of body forces as a source term. The body forces include centrifugal force, Coriolis force and electromagnetic force. The surface forces include pressure and viscous forces and the body force is the gravity forces. To derive the conservation of momentum equation in each direction, the stress components and pressure are needed as shown in Figure 12.
Figure 12: Stress components in x-direction [30]

Taking the rate of change into account and combining the stress components, the equation of momentum conservation in the x-direction is the following [30]:

$$
\rho \frac{Du}{Dt} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial (\tau_{yx})}{\partial y} + \frac{\partial (\tau_{zx})}{\partial z} + S_{Mx}
$$

(6)

Similar to the x-direction, the conservation of momentum equation in the y-direction is:

$$
\rho \frac{Dv}{Dt} = \frac{\partial (\tau_{xy})}{\partial x} + \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial (\tau_{zy})}{\partial z} + S_{My}
$$

(7)

Third, the conservation of momentum equation in z-direction is:

$$
\rho \frac{Dw}{Dt} = \frac{\partial (\tau_{xz})}{\partial x} + \frac{\partial (\tau_{yz})}{\partial y} + \frac{\partial (-p + \tau_{zz})}{\partial z} + S_{Mz}
$$

(8)
The effect of surface stresses are accounted, the source terms $S_{Mx}$, $S_{My}$, and $S_{Mz}$ include the body forces.

**Conservation of Energy**

The equation for conservation of energy is derived from the first law of thermodynamics, which states that the rate of change of energy is equal to the rate of work done by the fluid and the rate of heat transferred to the fluid due to conduction.

The rate of work done is defined by a surface force which is equal to the product of the components of force and velocity in the direction of the force. All the component of all directions combined states the following equation of total rate of work done on a fluid particle [30]:

\[
[-\text{div}(pu)] + \left[ \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} \\
+ \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} \right]
\]

(9)

The heat flux vector can be divided into its three components. The rate of heat transferred to the fluid particle can be calculated in all three directions with its component shown in Figure 13.
Combining the components yields the final form of the rate of heat addition to the fluid particle due to heat conduction across the element boundaries.

$$-\text{div } q = \text{div}(k \text{ grad } T)$$  \hfill (10)

The final conservation of energy equation includes the potential energy changes as source term. This will lead to the following energy equation [30]:

$$\rho \frac{DE}{Dt} = [-\text{div}(pu)] + \left[ \frac{\partial(u_{tx})}{\partial x} + \frac{\partial(u_{ty})}{\partial y} + \frac{\partial(u_{tz})}{\partial z} + \frac{\partial(v_{tx})}{\partial x} + \frac{\partial(v_{ty})}{\partial y} + \frac{\partial(v_{tz})}{\partial z} + \frac{\partial(w_{tx})}{\partial x} + \frac{\partial(w_{ty})}{\partial y} + \frac{\partial(w_{tz})}{\partial z} \right] + \text{div}(k \text{ grad } T) + S_E$$  \hfill (11)
3.2.1.4.2 Turbulence Model – k-ω SST

Within the Reynolds-averaged Navier-Stokes method, two equation models have been used most frequently for various applications. The turbulence model k-ω SST Menter is a well-known robust turbulence model with separation regions. The k-ω SST Menter model is an extension to the original k-ω turbulence model. It has the k-ε turbulence model embedded for near wall treatment. The Menter Shear Stress model is a two layer model which employs the k-ω model near walls and k-ε model in the outer region. It has been found that the SST model provides good results of wall bounded flow with highly separated regions. The following equations define the model [31]. The specific dissipation rate, $\omega$ is defined as:

$$\omega = \frac{\varepsilon}{\beta^* k}$$  \hspace{1cm} (12)

The k-ω equations:

$$\frac{D(\rho k)}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{k2} \mu_t) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (13)

$$\frac{D(\rho \omega)}{Dt} = \gamma \frac{\omega}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{\omega2} \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2 \rho \sigma_{\omega2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + 2 \rho \sigma_{\omega2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$  \hspace{1cm} (14)

The viscosity is then calculated as:

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, \Omega)}$$  \hspace{1cm} (15)

The used constants are: $\beta^* = C_\mu = 0.09, \beta_2 = 0.0828, \gamma = 0.44, \sigma_{k2} = 1.0, \text{ and } \sigma_{\omega2} = 0.857, a_1 = 0.31$ and $\Omega$ is the absolute value of the vorticity. The k-equation does
transform from that of the baseline k-ε model, Equation 12, but the standard ε-equation for ω results in the following equation:

\[
\frac{D(\rho \omega)}{D\tau} = \gamma \frac{\omega}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta_2 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_{\omega^2} \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \rho \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \\
+ \frac{\omega}{k} \frac{\partial}{\partial x_j} \left[ (\sigma_{\omega^2} + \sigma_{k^2}) \mu_t \frac{\partial k}{\partial x_j} \right]
\] (16)

The last term is an exact transformation from Equation 13 which is not included in Equation 6. In addition, the ω-equation diffusion coefficient transforms from the ε equation as \( \sigma_{\omega^2} = 1 \), \( \sigma_{\epsilon} = 1/1.3 = 0.769 \) and \( \sigma_{\omega^2} = 0.857 \) which corresponds to value of \( \sigma_{\epsilon} = 1.17 \). The value of the production of dissipation \( \gamma = 0.44 \) comes from the following equation with Karma constant \( \kappa = 0.41 \).

\[
\gamma = \frac{\beta_2}{\beta^*} - \frac{\sigma_{\omega^2} \kappa^2}{\sqrt{\beta^*}}
\] (17)

A more detailed explanation and possible extensions to the k-ω turbulence model are very well described by Georgiadis et al [31].

3.2.1.4.3 Navier-Stokes Equations

The Navier-Stokes equations describe the motion of a viscous fluid. These equations arise form Newton’s second law to fluid motion. The Navier-Stokes equations are not a conservation equation and are built on the assumption of Newtonian and isotropic fluid,
which means that viscous stresses are not a function of direction (isotropic) and proportional to the strain rate (Newtonian). The three Navier-Stokes equation looks like the following in the general form for incompressible flow [29]:

\[
\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho g_x - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \tag{18}
\]

\[
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho g_y - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \tag{19}
\]

\[
\rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho g_z - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \tag{20}
\]

In this study, the Reynolds-Averaged Navier-Stokes (RANS) equation is used. The RANS equation considers the turbulent flow. Therefore, the velocities and pressure get separated into the time-averaged and fluctuation values. For example, the \( u \) velocity is built the following [32]:

\[
u = \bar{u} + u' \tag{21}
\]

For the final RANS equations, the velocities and pressure get replaced by the time-averaged and fluctuation values. This leads to the following RANS equations [32]:

\[
\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x} + \frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} + \mu \left( \frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \tag{22}
\]

\[
\frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial y} + \frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} + \mu \left( \frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} \right) \tag{23}
\]

\[
\frac{\partial \bar{w}}{\partial t} + \bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial z} + \frac{\partial^2 \bar{w}}{\partial x^2} + \frac{\partial^2 \bar{w}}{\partial y^2} + \frac{\partial^2 \bar{w}}{\partial z^2} + \mu \left( \frac{\partial^2 \bar{w}}{\partial x^2} + \frac{\partial^2 \bar{w}}{\partial y^2} + \frac{\partial^2 \bar{w}}{\partial z^2} \right) \tag{24}
\]
3.2.1.5 Solver Models

There are two main actual solver models to solve the governing equations. On the one hand, there is the finite element method and on the other hand the finite volume method. The finite element method is commonly used in material analysis, FEA, whereas finite volume is the common one for CFD codes. Therefore, only the finite volume method is explained here. Further, the semi-implicit method for pressure linked equations is an additional algorithm which is used in segregated flow models.

3.2.1.5.1 Finite Volume Method

The finite volume method is described very well in Versteeg and Malalasekra’s [30] book using a simple pure diffusion in the steady state case. The finite volume method uses the governing equations to solve the problem and form the control volume. Unlike the finite element method, the finite volume refers to a small control volume which is surrounding each calculation node in a mesh and not the point itself. The differential form of the governing equation gets integrated over each control volume. Within the finite volume method, having the mesh, or grid of the CFD model, each cell serves as a control volume. To describe the variation of the concerned variables between cell centroids, interpolation profiles are assumed. The resulting discretization equation expresses the conservation principles of the conservation principles described in section 3.2.1.4. The resulting solution satisfy the conservation quantities for each control volume and therefore for each model independent of any number of control volumes.
The discretization equation in general form is the same for one-, two-, or three-dimensional as stated by Versteeg and Malalasekra [30].

\[ \alpha_p \phi_p = \sum \alpha_{nb} \phi_{nb} + S_u \]  

(25)

where \( \sum \) indicates summation over all neighbouring nodes (nb).

3.2.1.5.2 SIMPLE Algorithm

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) method was originally developed by Patankar and Spalding in 1972 [33]. The SIMPLE algorithm, or pressure-corrector procedure is very useful for incompressible flow. The solution procedure is simple and proceeds by a cyclic series of guess and correct operations. The important operations are described by Van Doormaal and Raithby [34] in the following steps below.

i. Guess the pressure field, \( p^* \).

ii. Solve the momentum equation to obtain \( u^* \) and \( v^* \).

iii. Solve the pressure correction equation to obtain \( p' \).

iv. Calculate \( p \) from equation by adding \( p' \) and \( p^* \).

v. Calculate \( u \) and \( v \) from their starred values using velocity correction equation.

vi. Solve all other discretized transport equations.

vii. Treat the corrected pressure \( p \) as new guessed \( p^* \), and return to step ii until convergence is obtained.
To initiate the SIMPLE calculation process, the pressure field $p^*$ needs to be guessed. After that, the discretized momentum equations need to be solved. In order to numerically solve the velocity and pressure fields that obey the discretized momentum and continuity equation, the finite difference method is applied. This method involves integrating the continuity and momentum equations over a two-dimensional flow field. The derivation of the SIMPLE algorithm is broken down by Van Doormaal and Raithby [34] or Versteeg and Malalasekera [30].

After the process of discretization, the discretized of $u$-momentum equation becomes:

$$a_e u_e^* = \sum a_{nb} u_{nb}^* + b_e + A_e (p_p^* - p_E^*)$$

where, $p$ is pressure, $A_e$ is the area of the face of the P control volume.

Similar to the $u$-momentum equation, the $v$-momentum equation:

$$a_e v_e^* = \sum a_{nb} v_{nb}^* + b_e + A_e (p_p^* - p_E^*)$$

Now, the correction of the guessed pressure is defined as $p'$. The same is done for the velocity components $v$ and $u$. Therefore, to satisfy both the mass and momentum constraints, we get the following equations:

$$p = p^* + p'$$

$$u = u^* + u'$$

$$v = v^* + v'$$

The relation between $p'$ and $u'$ is obtained by the following equation:
Next, \( p' \) needs to be found. The exact equation for \( p' \) is derived from Equation (28) and (29) and the continuity constraint. The SIMPLE procedure derives a more suitable equation by neglecting the first term on the right hand side of Equation (29). Combining the simplified Equation (29) and Equation (28 a) gets:

\[
\begin{align*}
\dot{u}_e &= \dot{u}_e^* + d_e(p_p' - p_E') \\
\end{align*}
\]

where,

\[
\begin{align*}
d_e &= \frac{A_e}{a_e} \\
\end{align*}
\]

The continuity equation of the control volume is:

\[
\begin{align*}
(\rho u A)_w - (\rho u A)_e + (\rho v A)_s - (\rho v A)_n &= 0 \\
\end{align*}
\]

Introducing now equations for \( u \) and \( v \) into the continuity equation leads to:

\[
\begin{align*}
ap' &= \dot{a}_e p_E' + \dot{a}_w p_W' + \dot{a}_N p_N' + \dot{a}_S p_S' + b \\
\end{align*}
\]

where,

\[
\begin{align*}
ap &= a_E + a_W + a_N + a_S \\
ap_w &= (\rho A d)_w \\
ap_e &= (\rho A d)_e \\
ap_n &= (\rho A d)_n \\
\end{align*}
\]
\[ a_S = (\rho A)S \]  \hspace{1cm} (34 e)

\[ b = (\rho u^* A)_w - (\rho u^* A)_e + (\rho v^* A)_s - (\rho v^* A)_n \]  \hspace{1cm} (34 f)

Since the first term on the right hand side of Equation (29) is neglected, this approximation results in \( p' \) values that are too large. To remedy this, Patankar recommends under-relaxation in the momentum equation by employing \( \alpha \approx 0.5 \) (\( E \approx 1 \)), and under-relaxation of the pressure correction by replacing Equation (28 a) by

\[ p = p^* + \alpha_p p' \]  \hspace{1cm} (35)

where, \( \alpha_p \approx 0.8 \).

This completes the SIMPLE method.

A few recommendations can be followed to improve the SIMPLE method. First, the SIMPLEC approximation can improve the economy of the method. The changes to implement SIMPLEC into the SIMPLE code are minor. This method removes the need of \( \alpha_p \) under relaxation. Another recommendation, which is made in the paper entitled Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows [34], is the treatment of \( p' \) where velocity boundary conditions are prescribed. Last, the pressure \( p' \) can be modified at points where the pressure is specified. All of these recommendations are described in the paper [34].

### 3.2.2 Errors and Uncertainties in CFD

The use of CFD can only be justified by the level of accuracy and confidence in results. To address the issue of accuracy and confidence, different guidelines were formulated
by the American Institute of Aeronautics and Astronautics (AIAA) and the European Research Community On Flow, Turbulence And Combustion (ERCOFTAC). In the context of accuracy and confidence in CFD modeling, the definition of the error and uncertainty is widely accepted and state the following:

- **Errors:**
  - Numerical errors – which include roundoff-, iterative convergence-, and discretization errors
  - Coding errors – which include mistakes in the software
  - User errors – state the incorrect use of the software

- **Uncertainty:**
  - Input uncertainty – which include boundary condition, material and model
  - Physical model uncertainty – this is the discrepancy between real flow and CFD due to inadequate physical models.

### 3.2.2.1 Error Analysis

Coding errors are software errors, and user errors are human errors due to incorrect use of the software. CFD solves systems of non-linear partial differential equations in discretized form on mesh which covers the region of interest and boundaries. This rises the three sources of numerical errors: roundoff error, iterative convergence errors and discretization errors.

Roundoff errors are the result of a finite number of digits in the representation of real flow. These errors are generally controlled by carefully arranged floating point arithmetic operations to avoid subtracting equal size numbers or adding numbers with a large
difference in magnitude. A common practice in CFD is using gauge pressure relative to specified base pressure. This is a simple example of controlling the error by good code design.

The numerical solution of a flow problem requires an iterative process until the solution satisfy exactly the discretized flow equation in the interior and the boundary conditions. There are several different ways to construct useful truncation criteria in CFD. The most common one is the residual. The final solution after a number of iterations will have a difference between the left and the right hand side of each cell, which is the local residual.

The absolute residual value in the definition of local residuals prevents cancelation of positive and negative values which would result in a global residual of zero. There are three different ways of normalizing the global residual, and all of them have advantages and disadvantages in specific cases. Whichever is used, the normalized global residual always equals zero when the final solution is reached. CFD codes involve default specification of tolerances for the global residuals in mass, momentum, and energy, which are determined by systematical trials by the code supplier.

Discretization error can be made arbitrarily small by progressive reduction of time step and space mesh size. However, this requires more computational time. The discretization error comes from the truncation of the higher order terms of the Tylor series. The ingenuity of the CFD user and the resource constraint dictate the lowest level of numerical errors which is dependent on the simplification of the profile assumptions.
3.2.2.2 Uncertainty Study

The uncertainties are divided into the categories of input and physical model uncertainties. The input uncertainties include the three following headings: domain geometry, boundary conditions, and fluid properties. The domain geometry is the specification of the shape and size of the region of interest. The uncertainty lies between the design specification and the actual manufacturing. Since manufacturing tolerances exist, there is a discrepancy between model and product. In summary, the macroscopic and microscopic geometry is somewhat different between CFD model and actual model. The boundary conditions deal with the specific condition on the model surface. Simple assumptions are always made such as temperature, heat flux or adiabatic walls. Some cases have only partial information on boundary conditions and, therefore, the rest has to be assumed and generated through calculation. The contribution to uncertainty is the inaccuracy of the assumptions which are involved during the process of boundary inputs. The third input uncertainty are the fluid properties. Often they are assumed to be constant due to minimal variation of their properties on a certain simulation. This benefits the solution economy; however, if the assumption is inaccurate, an uncertainty of the solution exists. If the parameters of the fluid vary, errors due to experimental uncertainty and calculation show up.

The physical model uncertainty or limited accuracy is divided into lack of validity of sub models and lack of validity of simplifying assumptions. Modeling complex phenomena such as turbulence, combustion, heat and mass transfer involves so-called semi-empirical sub models. The following listed reasons bring uncertainty due to sub-models in CFD results:
- A complex flow may involve new or unexpected physical or chemical processes which are not accounted in the original sub model.
- In spite of availability of a more comprehensive sub model, a simpler model with less accuracy is selected to save computational resources.
- A complex flow may include the same mixture of physics/chemistry as the original simple flow but requires adjustment of the sub-model constants.
- The empirical constant may represent experimental data which have an uncertainty themselves.

Lack of validity of simplifying assumptions deal with the simplifications made in the setup. In many cases, the simplification is justifiable to good accuracy. However, the following simplifications are given an uncertainty of the solution:

- Steady vs. transient
- Two-dimensional, axisymmetric, symmetrical across one or more planes vs fully three dimensional
- Incompressible vs. compressible
- Adiabatic vs. heat transfer across the boundaries
- Single species/phase vs. multi-component/phase

The accuracy of simplifying assumptions contributes to the uncertainty of the physical model.
### 3.2.2.3 Verification and Validation

The verification process quantifies the error. Roach [35] coined the phrase ‘solving the equation right’ for the verification process. The validation process is to quantify the uncertainty, which Roach [35] stated as ‘solving the right equation’.

To verify the solution, the following assumptions for the numerical solution are made:

- The flow field is sufficiently smooth to justify the use of Taylor series expansion
- The convergence is monotonic
- The numerical method is in its asymptotic range

For two meshes with a refinement and solutions, the discretization error can be written in terms of the difference of the two flow solution. This leads to the following error calculation [30]:

$$E_{U,1} = \frac{U_2 - U_1}{1 - r^p}$$  \hspace{1cm} (36)

$$E_{U,2} = r^p \frac{U_2 - U_1}{1 - r^p}$$  \hspace{1cm} (37)

where, $E_{U,1}$ is the error in the coarse solution and $E_{U,2}$ is the error in the refined mesh solution, $r$ is the refinement ratio, and $p$ is the order of the numerical scheme.

Further, the grid convergence index, which is proposed by Roache [35], can be introduced by the following:
Through the grid convergence index (GCI), the error can be quantified. For constant refinement, the observed order \( \bar{p} \) of the truncation rate decay can be found as following [30]:

\[
\bar{p} = \frac{\ln \left( \frac{U_3 - U_2}{U_2 - U_1} \right)}{\ln(r)}
\]  

Finally, it can be said that the above noted method merely estimates the error of the code as is and does not test whether the code itself is accurate or not.

The validation of the input uncertainties can only be done with multiple CFD simulations. The observed results can be used for upper and lower bounds for their expected range. A quantitative assessment of the physical modeling uncertainties requires the comparison of CFD results and high quality experimental results.

For verification and validation, AIAA guide [36] and ERCOFTAC guideline [37] provide comprehensive strategies to conduct CFD modeling studies. Further, several public-access databases are listed to provide support for CFD validation work. Finally, a guideline for CFD simulation documentation is given which deals with the input documentation and result interpretation and reporting. For reporting, note that high-quality presentation is not necessarily the same as high-quality results. It is always essential to verify and validate the results carefully.
4 Phase 1 – Study of a Wing in Freestream

The goal of Phase 1 includes the building of a benchmark solution for the S1223 wing profile. The wing is analyzed at a Reynolds number of 250,000 which corresponds to an airspeed of 12.5 m/s. To have the whole effective range of the wing profile, the angle of attack gets varied between 0 and 18 degrees. This study shows that the lift, or downforce, increases with increasing angle of attack up to a transition point where the angle of attack gets too big and large separation occurs. A separation that is too large leads to stall condition in which the downforce decreases significantly.

4.1 Model

The model is built in SolidWorks by importing the available set of data points of the wing profile which can be found in the appendix. The model is built by cutting the actual wing out of the far field. The wing has a chord length c=300 mm and a wing span s=1500 mm. The dimensions of the far field has dimensions of 2000 mm in length, 2400 mm in width and 1000 mm in height. The symmetry function of Star CCM+ is used to save computational resources. Therefore, the model can be cut in half at the center plane. The wing is placed 600 mm downstream at a height of 500 mm. Figure 14 shows the Star CCM + model at an angle of attack of 6 degrees. The wing surface is defined as wall boundary condition whereas the boundaries of the far field are defined as freestream boundary condition.
4.1.1 Meshing

Several mesh iterations were done to reach an acceptable good mesh which catches everything. Within the meshing, two mesh refinements got placed around the wing as shown below in Figure 15.

Figure 15: Mesh refinement blocks (a) wing refinement (b) trailing edge refinement
The base size is chosen to be the cord length of the wing, 0.3 m. The wing refinement is set to be 1.5 % of the base size, and the trailing edge refinement is 0.8 %. One percent is also the size of the mesh of the wing profile. Further, an additional surface size refinement on the wing itself is set. The target size is set to be 0.9 % of the chord length with a minimum size of 0.5 %. The thickness of the prism layer is set to be 1.5 % with a number of 15 prism layers. Depending on the angle of attack the number of cells are 4 to 5 million. Figure 16 and Figure 17 show the mesh on the center plane.

Figure 16: Mesh view of center plane, freestream at AOA=6°

Figure 17: Mesh around the wing, freestream at AOA=6°
4.1.2 Physics

The physical flow model is chosen to be the segregated flow model. Segregated flow is chosen because the study is done within low speeds. As the turbulence model, K-Omega Turbulence model is chosen. K-Omega is a two equation model designed for models where flow separation is expected. Star CCM+ uses the actual STT turbulence model, which is a combination of the K-Epsilon and K-Omega. In Table 4, all physical models used are shown. These models are as they exist in Star CCM+. These physical settings are used throughout all the different phases.
Table 4: Physical models for freestream analysis

<table>
<thead>
<tr>
<th>Physical Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Three Dimensional</td>
</tr>
<tr>
<td>Steady</td>
</tr>
<tr>
<td>Gradients</td>
</tr>
<tr>
<td>Gas</td>
</tr>
<tr>
<td>Ideal Gas</td>
</tr>
<tr>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>All y + Wall Treatments</td>
</tr>
<tr>
<td>K-Omega Turbulence</td>
</tr>
<tr>
<td>SST (Menter) K-Omega</td>
</tr>
<tr>
<td>Turbulent Suppression</td>
</tr>
<tr>
<td>Turbulent</td>
</tr>
<tr>
<td>Transition Boundary Distance</td>
</tr>
<tr>
<td>Segregated Flow</td>
</tr>
</tbody>
</table>

### 4.1.3 Error Analysis

Analyzing the accuracy of the simulations is done by tracking the residuals of the governing equations. The residuals of the governing equations, Figure 18, show that all values of the governing equations have leveled out below 1%. However, the residuals are only a first indication of the accuracy of the simulation.
The second analysis for accuracy was done through the different iterations of meshes. Multiple meshes are done within this process. The accuracy of the mesh was tracked by the result. Once the results did not change between the different meshes, the solution can be seen as mesh independent. This was achieved with the previously mentioned settings for the mesh. Further, all simulations were run until there was no more fluctuations in drag or lift coefficient as shown in Figure 19 and Figure 20.
A third indication of the error analysis is the actual result. Therefore, the boundary layer Figure 21 can be analyzed and identified as fully developed within the prism layer mesh. A second indication if a simulation is fully converged is if the far field, the air far away from the wing, is a steady state. Therefore, a view on the velocity distribution (Figure 22) shows that the far field is fully converged by not having any unexpected changes in velocities.

Figure 20: Lift coefficient tracker freestream case with AOA=6°

Figure 21: Boundary layer at 2/3 of chord length on top surface for freestream with AOA=6°
4.2 Results

As a result of phase 1, the high lift wing profile S1223 is analyzed by the lift or downforce coefficient, the drag coefficient and the main flow structure results. These flow structure results include pressure distribution, velocity distribution, velocity vector analysis and the wake analysis.

4.2.1 Lift Coefficient Study in Freestream

For each angle of attack, the lift coefficient got calculated. The lift coefficient reveals the effective range of the wing profile. The effective range is normally from 0 degrees up to the point where stall condition occurs. A decrease in lift coefficient is known as stall condition. The results of the lift coefficient are compared to the experimental results of the S1223 profile.
The lift coefficient of the wing shows the normal behavior of a wing. By increasing the angle of attack, the lift coefficient increases. The wing reaches its maximum lift coefficient at an angle of attack of 14 degrees with a lift coefficient of 1.73. Comparing it to the experimental data, the S1223 reaches its maximum lift coefficient at an angle of attack of 16 degrees with a magnitude of 2.13. In Figure 23, the experimental results are compared to the CFD results done in this phase, it can be seen that the curves are similar up to 10 degrees angle of attack. Between 10 and 14 degrees, the simulation still shows an increase in in lift coefficient but not as strong as in the experimental results [5].

![Lift coefficient vs. Angle of Attack](image)

**Figure 23: Lift coefficient vs. angle of attack in free stream**

The difference between the two curves can be explained by the difference of experimental vs. computational results. However, during the process of building the
model, simulation showed that small changes on the profile can have a large impact on the lift coefficient. Since the model exists by only 80 location points in an X and Y table, the curve between the points gets built automatically by SolidWorks. By redrawing the profile with the spline function of SolidWorks, the lift coefficient got changed significantly. Redrawing the trailing edge, approximately the last 10% of the chord length, the lift coefficient decreased from 1.31 to approximately 1.1. Overall, the qualitative behavior of the lift coefficient is similar to the literature review, but offset.

4.2.2 Drag Coefficient Study in Freestream

The drag coefficient shows the normal behavior of a wing. By increasing the angle of attack, the frontal area gets automatically larger. This leads to an increase of the drag coefficient. Another impact on the drag is the size of the separation region around the trailing edge. As the flow structure analysis will show, the separation region grows by increasing the angle of attack. The separation of the boundary layer is also the main factor for stall condition. Figure 24 clearly shows the increase of drag while increasing the angle of attack. While the lift coefficient decreases once stall occurs, the drag coefficient keeps increasing as illustrated.
4.2.3 Flow Structure Analysis of a Wing in Freestream

The flow structure analysis reveals the behavior of the lift and drag coefficient discussed before. The freestream cases got analyzed for different angles of attack, namely AOA = 0°, 6°, 8°, and 14°. The shown analysis is conducted at the center plane for the velocity distribution, pressure distribution, and velocity vectors. The wake produced by the wing is conducted at a plane with a distance of D/c = 0.66, distance normalized by chord length, behind the trailing edge.

4.2.3.1 Velocity Distribution

The center plane velocity distribution around the wing shows the typical behavior around a wing. A zero velocity point at the leading edge can be seen. This is the stagnation point of the air hitting the wing. Further, a high velocity region underneath the wing is clearly visible. Since race car wings are fundamentally inverted airplane wings,
the high velocity region is underneath the wing and not above the wing as on an airplane wing. The flow accelerates while traveling along the wing profile underneath and creates a low pressure region, which can be seen clearly at the pressure distribution analysis. A third characteristic which can be observed is a low velocity region underneath the trailing edge. This region is the separation region of the flow from the profile. A better overview of this can be seen from the velocity vector analysis.

Comparing the four chosen cases, it can be seen that the high velocity region has a significant difference in actual velocity. Figure 25 to Figure 28 visualize the velocity distribution on the center plane for the chosen cases. At zero degrees angle of attack, the velocity increases by 52 % compared to the free stream velocity. By increasing the angle of attack, the maximum velocity underneath the wing increases. Six degrees angle of attack results in a velocity increase of 70.4 %, for AOA = 8° it is 76.8 % and for AOA = 14° it results in an increase by 88.8 %. By increasing the angle of attack, the stagnation point moves towards the top surface. Comparing the low velocity region near the trailing edge, it can be seen that it is growing significantly by increasing the angle of attack. This separation region growth from almost nothing at AOA = 0° to approximately 2/3 of the chord length at AOA = 14°.
Figure 25: Velocity distribution on center plane, freestream with AOA=0°

Figure 26: Velocity distribution on center plane, freestream with AOA=6°

Figure 27: Velocity distribution on center plane, freestream with AOA=8°
4.2.3.2 Pressure Distribution

The pressure distribution shows three major areas: the stagnation pressure region, low pressure region and the pressure region above the wing. The stagnation pressure does not really change, only the location. By increasing the angle of attack, the stagnation point moves towards the upper surface as already seen in the velocity distribution. The maximum pressure decreases slightly between AOA = 0° to AOA = 6° and AOA = 8°, which can be seen in Figure 29 to Figure 31. This small decrease can be explained by moving the stagnation point closer to the leading edge of the wing profile. However, there is an increase of the stagnation pressure for AOA = 14° visual in Figure 32. In this case, it can be seen that the stagnation region is getting bigger and affects the whole pressure side on top of the wing. It can be seen that by increasing the angle of attack, the pressure region on the top surface increases and higher pressure occurs. This pressure leads automatically to an increase of the downforce. The counter part of the
high pressure region above the wing is the low pressure region underneath the wing. The low pressure region is an actual suction region. At AOA = 0°, there is an actual negative pressure of negative 133 Pa. By increasing the angle of attack, this negative pressure gets even larger in magnitude. This is the effect of the accelerated air which could be seen before. At an angle of attack of AOA = 6° the negative pressure increases by 33.95 % compared to zero degrees, for AOA = 14° the negative pressure increases by 80.45%. This behavior is the same as seen from the velocity distribution. Higher velocity gets reflected in lower pressure.

![Figure 29: Pressure distribution on center plane, freestream with AOA=0°](image1)

![Figure 30: Pressure distribution on center plane, freestream with AOA=6°](image2)
4.2.3.3 Velocity Vectors

The velocity vectors capturing the separation region show in all of the cases a clear separation. A small separation region with a small vortex can be seen at AOA = 0° in Figure 33. Changes in size of the separation region could already been seen during the velocity distribution analysis. Now it can be seen clearly that the vortex grow in size by increasing the angle of attack. At an angle of attack of 14 degrees, a huge separation
region can be identified with multiple vortices. Figure 33 to Figure 36 show the velocity vectors catching the separation region for the different cases.

Figure 33: Velocity vectors at separation region on center plane, freestream with $\text{AOA}=0^\circ$

Figure 34: Velocity vectors at separation region on center plane, freestream with $\text{AOA}=6^\circ$

Figure 35: Velocity vectors at separation region on center plane, freestream with $\text{AOA}=8^\circ$
4.2.3.4 Wake Analysis

The wake characteristic is as expected—the wing creates a low velocity region along the span of the wing. Further, as it is very common for wing, the wing tip vortices can be seen clearly.

The velocity distribution of the wake, Figure 37 to Figure 40, shows that by increasing the angle of attack, the wake growth, as it could be seen already at the velocity distribution on the center plane. The low velocity along the wing span is significantly decreasing. Further, the wing tip vortex is getting stronger by increasing the angle of attack. For AOA = 6° and 8°, the wing tip vortex velocity is not changing since the wake is more or less the same size. At AOA = 14°, there is a significant higher vortex velocity and a very low velocity behind the wing.
Figure 38: Velocity distribution at a distance $D/c=0.66$ for freestream at AOA=6°

Figure 39: Velocity distribution at a distance $D/c=0.66$ for freestream at AOA=8°

Figure 40: Velocity distribution at a distance $D/c=0.66$ for freestream at AOA=14°

A closer look on the wing tip vortices show that there is not a big difference in the actual flow characteristics as may be seen in Figure 41 to Figure 44. All the vortices rotate clockwise. The only difference is the already mentioned velocity difference, also known as the strength of the vortex.
Figure 41: Wing tip vortex for freestream at AOA = 0°

Figure 42: Wing tip vortex for freestream at AOA = 6°
Figure 43: Wing tip vortex for freestream at AOA = 8°

Figure 44: Wing tip vortex for freestream at AOA = 14°
4.3 Conclusion of Phase 1 – Study of a Wing in Freestream

Phase 1 showed the normal behavior of a wing in freestream. The lift coefficient analysis shows that this model of the wing is slightly off compared to the published experimental results. This difference can be explained by various factors. One of these reasons would be computational vs. experimental results. Further, it could be seen that minor changes on the profile have significant impact on the lift coefficient results. The used set of points are the theoretical wing profile, an actual difference to the manufactured wing for the experimental data could be a factor too. Overall, this study is the first benchmark solution for phases 3 and 4.
5 Phase 2 – Creation of a Bluff Body

The goal of this phase is to create a bluff body to generate a wake in which the wing will operate in phase 4. Further, the wake analysis of this bluff body is used to determine suitable distance ranges and meshing parameters for the study in phase 4. The bluff body is analyzed at a racing speed of 30 m/s.

5.1 Model

Wilson et al [7] showed that a simple bluff body is sufficient to capture the significant parts of a race car wake. Therefore, a model is created which looks similar to the one Wilson et al [7] presented. This particular model has some simplification on the rear end of the car since the main effect of the wake will be modeled by the appropriate size of the body and the rear wing. The drawing in Figure 45 shows the dimension of the created bluff body.

![Bluff body dimensions](image)

Figure 45: Bluff body dimensions
The designed bluff body as shown in Figure 46 (b) is cut out of a far field and cut in half to use the symmetry option. The dimension of the far field is 7500 mm in length and 4500 mm in width with a height of 2000 mm. The bluff body is placed 1250 mm downstream. The physical settings are the same as those used in phase 1. Figure 46 (a) shows the complete simulation model used in Star CCM+.

![Figure 46: (a) CFD model of bluff body simulation (b) 3D bluff body model](image)

5.1.1 Meshing

Several mesh iterations were done to reach an acceptable good mesh which captures the complete flow with its disturbance. Within the meshing, two mesh refinements blocks got placed around the bluff body and far enough downstream to capture the wake as may be seen in Figure 47. Both of the refinement blocks are set to the same refinement. Block 1 does not reach the full height of the car; therefore, block 2 got implemented to have a mesh refinement for the wake coming from the rear wing. Having the cutout of the refinement area saves a significant amount of computing resources. The number of cells saved reaches a few million.
The base size is chosen to be the cord length of the wing, 0.3 m, which is used in all other simulations. The modeled rear wing of the bluff body got a local refinement to make sure the flow around that wing gets modeled accurately. The target size is set to be 1.5 % of the chord length with a minimum size of 1.0 %. These refinement parameters are bigger than the one for the simulated wing in the other phases since there is no force calculation needed in this case. The block refinement is set to be 2.8 % of the chord length. The final number of cells is 19.8 million. The described refinements are achieved through different mesh iterations. Figure 48 and Figure 49 show the mesh on the center plane.
5.1.2 Error Analysis

Since there are no aerodynamic forces of importance, there are only the residuals of the governing equations to track and the actual flow structure solution. The residuals shown in Figure 50 show a converged solution. The three momentum equations and the energy equation are in the range of approximately 0.5% error. The continuity equation fluctuates at about 5% error. The fluctuation is common since the wake is not a steady solution. The wake changes because there are many vortices existing, which may be seen in the results in section 5.2. Therefore, the unsteady solution within a steady state simulation starts to repeat the solution which results in the fluctuation of the residuals.
The velocity distribution is tracked during the simulation after 600, 900, 1000, and 1225 iterations shown in Figure 51 to Figure 55. It can be seen that the wake size does not change between 900 and 1225 iterations. However, there is a continuous change of the rear bottom end of the wake where the wake hits the ground. This is because all the vortices within the wake have an influence on the flow and make it change continuously. The actual velocity around the bluff body itself does not change. This is an indication that the flow structure has converged.

Figure 51: Velocity distribution of bluff body wake after 600 iterations

Figure 52: Velocity distribution of bluff body wake after 900 iterations
5.2 Results

The bluff body is only analyzed through flow structure. Any aerodynamic forces acting on the body are not of interest in this study. The objective of this phase is to see how the wake looks for the designed bluff body. Therefore, the velocity distribution has been analyzed on the center plane, Figure 56, and three cross-section planes in the wake.
which can be seen in Figure 59 to Figure 64. The cross-section planes are located at distances normalized by a car length of $D/L = 0.1$, $0.4$, and $0.5$. Further, the velocity vectors are used to determine vortices within the wake, and the streamlines show the general flow with its disturbance around the car.

The velocity on the center plane shows a stagnation point at the leading edge of the car, accelerated flow underneath and above the car as well as a high velocity region underneath the modeled rear wing. The rear wing shows a large separation region which starts approximately after $50\%$ of the chord length. This is because the angle of attack of the rear wing was chosen to be $20$ degrees. The body wake region behind the bluff body is approximately the length of the bluff body itself. Figure 56 shows a wake leaving the rear wing and a wake leaving the body. Since this is a highly disturbed flow, the error analysis before showed that the wake is never constant. However, the size stays approximately the same. It can be seen that the wake is longer near the ground than in the middle of the body.

![Figure 56: Velocity distribution on center plane of the bluff body](image)

The view on velocities above set freestream in Figure 57 shows that high velocity near the ground exists near the bluff body. This high velocity is produced by the underbody. It can also be seen that the velocity is at or above the set freestream velocity towards the
end of the wake. Everything which is not included in the defined velocity scale are white areas.

![Velocity distribution for velocities higher than 30 m/s](image1)

**Figure 57:** Velocity distribution for velocities higher than 30 m/s

Having a look on the actual wake length, the three cross section planes can be seen in Figure 58. It can be seen that at a distance of D/L = 0.1, the plane is located within the main wake of the body where very low velocities exist. D/L = 0.3 and 0.5 are at the end of the wake. At D/L = 0.3, has still more wake effect from the bluff body than 0.5.

![Cross-section plane placing in bluff body wake](image2)

**Figure 58:** Cross-section plane placing in bluff body wake

The cross-section view of the bluff body wake at D/L = 0.1 shown in Figure 59 shows three major areas. There is a high velocity region centered which comes from the accelerated flow from the underbody of the bluff body. Further, there is the “mushroom” shaped wake centered from the rear wing which was already found by Wilson et al [7]. The third region are two outside vortices which are initiated by the wheels. Outside the low velocity wake region, the local velocities are higher compared to the set freestream.
velocities. This is initiated by the tangential velocity of the vortices, which increases the magnitude of the represented velocity.

Looking at the projected velocity vectors on the cross-section plane at D/L = 0.1 in Figure 60, it can be seen that there are multiple vortices existing in the wake. The two big ones in the center are the wing tip trailing vortices of the rear wing. Further, there are two vortices right above the high velocity section, which are initiated by the main body. The ones near the ground on the outside are produced by the wheel. It can be seen that at a short distance behind the bluff body, the flow is extremely disturbed, which will have a big impact on any object placed in there.
The further away from the bluff body, the smaller the impact becomes. A significant decrease in velocity extremes can be observed. At a distance of $D/L = 0.3$, the mushroom wake is still clearly visible, Figure 61, whereas at a distance of $D/L = 0.5$ this “mushroom” shape has almost disappeared, which may be seen in Figure 62. Recall from the velocity distribution on the center plane, at a distance $D/L = 0.5$, the main wake effect from the body is almost gone. However, the wing tip vortices from the rear wing are clearly visible. These wing tip vortices are visible a long way down stream through the complete model of the CFD simulation. Since the low velocity region of the wake gets smaller in height downstream, the disturbance and low velocity region is still visible near the ground at a distance of $D/L = 0.5$ downstream. Although these low velocity regions are still visible, the velocity recovers more towards its original speed. Another observation is that the main velocity on the cross-section planes is higher than the set freestream velocity. Through the vortices, the air achieves locally higher velocity magnitudes.
The projected velocity vectors in Figure 63 show that some of the vortices have disappeared at a distance of D/L = 0.3, and even more have disappeared and obviously weakened at D/L = 0.5 shown in Figure 64. However, the two main trailing vortices from the modeled rear wing are still clearly visible.
The pressure distribution on the bluff body shows multiple high pressure regions. These high pressure regions are located on the leading edge of the body, the front of the wheels, the leading edge of the rear wing and the leading face of the rear wings end plates that can be seen in Figure 65. The highest pressure is not the stagnation
pressure on the leading edge of the body; it is higher on the leading edge of the rear wing, the wheels, and endplate since there is accelerated air hitting the surface. The low pressure regions are the underbody and the rear end.

The visualized streamlines around the bluff body in Figure 66 show that the air is accelerated off the edges and the streamlines visualize an attached flow until they hit the rear wing or the wheels. The flow around the rear wing pushes the streamlines down and force them to go through the opening between the body and rear wing.
Placing multiple streamline tubes around the car show how highly disturbed the flow behind the car is. In Figure 67, it is clearly visible that the main body wake loses quickly on size whereas the wing tip vortices of the rear wing go on until the end of model.

![Streamline tubes around the bluff body](image)

Figure 67: Streamline tubes around the bluff body

### 5.3 Phase 2 Conclusion – Creation of a Bluff Body

The creation of a simple bluff body showed that all the main wake characteristics could be modeled. The “mushroom” shaped wake described by Wilson et al [7] could be identified clearly. The flow structure analysis showed that the main wake of the body lost its strength continuously and is almost gone at a distance of D/L = 0.5. The velocity distribution showed that within the rear end of the body wake, higher velocities exist than freestream velocity. The wake analysis leads to an appropriate range of D/L = 0.1 to 0.5 for the distance between bluff body and wing in phase 4. This range is chosen to have the strong effect of the body wake at D/L = 0.1 and at the end and behind the body wake at D/L = 0.5. The interesting range of change is assumed to be towards the end of the chosen range.
6 Phase 3 - A Wing Operating in Ground Effect

Within phase 3, the effect of the ground clearance is studied. The ground clearance, \( H/c \), which is normalized by the chord length is measured between ground and leading edge point and is normalized by the chord length. Since the wing is modeled as a stationary object, the air around is modeled with a relative velocity to the wing. A race car wing also has a relative velocity to the ground. Therefore, the ground gets defined as a so-called moving ground and has a relative velocity compared to the wing.

6.1 Model

The wing dimensions are the same as in phase 1. The chord length is 300 mm and the span is 1500 mm. The wing is placed in a far field with a length of 2000 mm, a width of 4000 mm, and a height of 1000 mm. Since the symmetry option is used, the whole domain got cut into half and the center plane defined as a symmetry plane. Because the ground has a relative velocity to the wing, a velocity vector is defined with the defined racing speed. The Star CCM+ model can be seen in Figure 68 with the above mentioned dimensions.
6.1.1 Meshing

The common different mesh iterations were performed to find a mesh setting where the results do not change any more to get a mesh independent result. Two refinement blocks are placed within each other to model the flow accurately around the wing as shown in Figure 69. Block 1, the larger block, models the flow of the wake, and Block 2 models the immediate flow around the wing. Block 1 has a refinement to a cell size of 2 % of the chord length, and Block 2 is set to be 0.8 % of the chord length. The small refinement around the wing is needed to model the flow accurately between the wing and ground.
Figure 69: Refinement blocks phase 3 model at H/c=0.3

The base size is chosen to be the cord length of the wing, 0.3 m. A surface size refinement on the wing itself is set. The target size is set to be 0.9 % of the chord length with a minimum size of 0.5 %. The thickness of the prism layer is set to be 1.5 % with a number of 15 prism layers. Depending on the ground clearance, the number of cells is 6 to 10 million. Figure 70 and Figure 71 show the created mesh on the center plane for the case with a ground clearance of H/c = 0.3.
6.1.2 Physics

The physical flow model is chosen to be the segregated flow model, which is selected because the study is done within low speeds. During the process of finding the ideal physics settings, coupled flow was sampled. The final result was equal to the segregated flow model. However, the number of iterations doubled until a conversion of the aerodynamic forces was achieved. Therefore, it was decided to use the segregated flow model. The physical settings are, therefore, the same as in phase 1 and 2. All the chosen model can be found in Section 4.1.2 in Table 4.
6.1.3 Error Analysis

The three steps of error analysis have been performed for phase 3. These three steps are the same as in phase 1; analyzing the residuals of the governing equation, observe the aerodynamic forces until they converge to a result, and through post processing.

Because of the near ground, the flow is not steady anymore which leads to fluctuation in the residuals that can be seen in Figure 72. The momentum and energy equation residuals fluctuate with an error rate of below 1 %. The continuity equation, the scalar dissipation rate (sdr), and turbulent kinetic energy (Tke) have fluctuating residuals at approximately 10 % error. However, high residuals in the turbulence model residuals are not necessarily an indicator for a bad solution.

![Residuals of governing equation, phase 3 at H/c=0.3](image)

Figure 72: Residuals of governing equation, phase 3 at H/c=0.3

Since the wing operates near ground, the solution starts to get unsteady. Therefore, it can be seen in Figure 73 and Figure 74 that lift and drag coefficient do not converge to a steady number. However, it is illustrated that both of the aerodynamic force coefficient fluctuate after a while at the same level. Therefore, the solution has fully converged and is accurate. Since different mesh refinement iterations were performed in advance, it is
also known that the solution is mesh independent. The aerodynamic force coefficients, the lift, Figure 73, and drag coefficient, Figure 74, are taken as an average of the oscillating solution propagation.

![Lift Coefficient Tracker](image1)

**Figure 73:** Lift coefficient tracker, phase 3 at H/c=0.3

![Drag Coefficient Tracker](image2)

**Figure 74:** Drag Coefficient Tracker, phase 3 at H/c=0.3

Third, the solution gets inspected to see if it fully converged to a solution. This can be seen if there are no unexpected changes in velocity in the far field. Figure 75 shows that the area in front of the wing and far above the wing have a constant velocity. Further, the wake recovers fully to the free stream velocity which exists again at the end of the domain. Therefore, the solutions are accurate.
6.2 Results

The aerodynamic force coefficients are analyzed in terms of their values and compared to the free stream model. Further, the flow structure analysis is used to support and explain the observed behavior of lift coefficient and drag coefficient.

6.2.1 Lift Coefficient Study of a Wing Operating in Ground Effect

The lift or downforce coefficient is calculated for all the different ground clearance values. The overall expected behavior is to have an increase in downforce while decreasing the ground clearance H/c up to a certain point. If the wing gets too close to the ground, the boundary layers start to merge, which has a negative impact since the boundary layer have lower velocities. Therefore, if boundary layers merge, the velocity underneath the wing gets slowed down.
The lift coefficient change shown in Figure 76 of the Figure 73 phase 3 study, the wing operating near ground shows that there is continuous increase of the lift coefficient up to a ground clearance $H/c = 0.22$. The effect of the ground, which is moving with the relative velocity to the wing, can be seen by a maximum increase of 46.4% in lift coefficient. As soon as the lift coefficient reaches its maximum, there is a significant decrease in downforce which was also stated by various other studies [9] [18] [20]. Analyzing the lift coefficient at a very low ground clearance, $H/c = 0.15$, it actually has a negative impact. The lift coefficient decreased by 16.7% compared to the free stream case in phase 1 whereas all the other tested ground clearances increase the downforce compared to the phase 1 result.

![Change in Lift Coefficient vs. Freestream](image)

**Figure 76: Change in lift coefficient vs. freestream in ground effect**

The seen phenomena of increasing downforce while decreasing the ground clearance can be explained by Bernoulli’s principle which states that if an increase in the velocity
of a fluid occurs simultaneously with a decrease in pressure or the potential energy of the fluid and vice versa. The wing and the ground build a duct together. By decreasing the ground clearance, the air gets accelerated underneath the wing. The closer to the ground it gets, the more the air gets increased which causes negative pressure. This negative pressure acts as a suction region which pulls the wing towards the ground. The velocity distribution around the wing can be found in section 6.2.3.1.

### 6.2.2 Drag Coefficient Study of a Wing Operating in Ground Effect

The overall drag can be divided into pressure and skin-friction drag. The pressure drag consists of form drag and induced drag, also known as the vortex drag. The induced drag depends mainly on the wake behind the wing. Skin-friction or surface-friction drag is depending on shear stresses acting on the wings surface [29].

The drag coefficient in Figure 77 shows an increase while decreasing the ground clearance \( H/c \). The drag coefficient does not reach its maximum at a ground clearance of \( H/c = 0.22 \) such as the lift coefficient. The drag coefficient increases by 63.2 % at the maximum lift coefficient ground clearance. Since the drag keeps increasing at a ground clearance of \( H/c = 0.15 \), the drag has increased by 95.6 % compared to the freestream case in phase 1.
By decreasing the ground clearance, the wing and ground build a duct. This leads to an acceleration of the air. Higher velocity corresponds to higher skin-friction drag. The friction drag is not the only component which affects the overall drag. The induced drag starts to play a bigger impact by decreasing the ground clearance. The induced drag is produced by the wake. The wake gets more disturbed, and multiple vortices are added to it at lower ground clearance. At freestream, the only two vortices which could be seen were the wing tip vortices. The closer the wing gets to the ground, more vortices become visible and have a negative impact, which means increased drag.

6.2.3 Flow Structure Analysis of a Wing Operating in Ground Effect

The flow structure analysis is performed on multiple cases to show the impact on lift and drag coefficient. The velocity and pressure distribution is analyzed at the center plane as well as the stream lines to show the flow around the wing. Further, the wake is analyzed. The velocity distribution on the plane at $D/c = 0.66$ shows the different vortices and changes in vortices between different ground clearances.
### 6.2.3.1 Velocity Distribution

An increase of the maximum velocity underneath the wing can be seen while analyzing the velocity distribution in Figure 78 to Figure 83. It can also be seen that by decreasing the ground clearance, the wake is not steady anymore. The ground has a huge impact on the wake and its vortices. As stated before, the maximum lift coefficient is reached at \( H/c = 0.22 \), which also corresponds to the ground clearance with the maximum velocity. The velocity increases by decreasing the ground clearance up to \( H/c = 0.22 \). The velocity increases between a ground clearance of \( H/c = 0.3 \) and 0.22 by 6.8%. As illustrated in the lift coefficient analysis, the lift coefficient drops significantly after reaching the maximum. The velocity distribution shows that the smaller the ground clearance, the more disturbance occurs within the wake. At a larger ground clearance of \( H/c = 0.3 \), the wake has significantly less vertical disturbance, whereas at \( H/c = 0.17 \) the wake covers a significant larger area and grows in vertical size. This growth can be seen clearly at the wake analysis.

![Velocity distribution on center plane at H/c=0.3 in ground effect](image)

**Figure 78**: Velocity distribution on center plane at \( H/c=0.3 \) in ground effect
Figure 79: Velocity distribution on center plane at H/c=0.24 in ground effect

Figure 80: Velocity distribution on center plane at H/c=0.2333 in ground effect

Figure 81: Velocity distribution on center plane at H/c=0.2266 in ground effect
6.2.3.2 Pressure Distribution

Similar to the freestream analysis, the pressure distribution shows three main areas: the stagnation pressure at the leading edge, a high pressure region above the wing, and a low pressure region underneath the wing. The presented different ground clearance cases in Figure 84 to Figure 87 show only a small fluctuation on the stagnation pressure, which is also the highest acting pressure on the wing, producing a fair amount of the total drag. The low pressure region underneath the wing, which is a negative gauge pressure, behaves correspondent to the velocity. The velocity distribution showed that the flow is accelerating underneath the wing which causes a decrease of
pressure. Where the velocity increases by 6.8% between a ground clearance of H/c = 0.3 and 0.22, the pressure decreases by 18.6%. Further, it can be seen that the low pressure region extends beyond the trailing edge of the wing. This shows that low or even negative pressure exists in the wake region.

Figure 84: Pressure distribution on center plane at H/c=0.3 in ground effect

Figure 85: Pressure distribution on center plane at H/c=0.24 in ground effect
6.2.3.3 Center Plane Streamlines

The streamlines located on the center plane show the separation region of the flow clearly. It can be seen that the flow gets force underneath the wing. Further, the point of separation occurs approximately 2/3 of the chord length downstream. The streamlines show that the fluctuation in the vertical axis changes and the separation region grows vertical and horizontal. The streamline Figure 88 to Figure 93 give a good representation how the wake is not steady. For example, at H/c = 0.22, the wake gets pulled right to the ground which can also be seen at H/c = 0.17.
Figure 88: Streamlines on center plane at H/c=0.3 in ground effect

Figure 89: Streamlines on center plane at H/c=0.24 in ground effect

Figure 90: Streamlines on center plane at H/c=0.2266 in ground effect

Figure 91: Streamlines on center plane at H/c=0.22 in ground effect
6.2.3.4 Wake Velocity Distribution

Viewing the velocity distribution within the wake on a plane with a distance D/c = 0.66 shows how the wake does change by changing the ground clearance. Recall from phase 1, the freestream case in Figure 94, which shows a continuous low velocity region across the wing span and two wing tip vortices. In Figure 95 to Figure 98, it can be seen that by decreasing the ground clearance, vortices start to develop across the wing span. The more the ground clearance gets reduced, the more and stronger these mid-span vortices get. These vortices have direct influence on the induced drag and, therefore, have a negative influence on the aerodynamic forces. The low velocity area, which started as a constant wake in freestream, gets more and more shattered the smaller the ground clearance gets. At a ground clearance H/c = 0.22, the wake which looked like a beam became a kind of wave through all the existing span-wise vortices.
While more vortices start to occur, the wing tip vortices get slowed down at smaller ground clearance.

Figure 94: Wake velocity distribution at a distance D/c=0.66 for freestream

Figure 95: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.3

Figure 96: Wake velocity distribution at a distance D/c=0.66 for ground clearance H/c=0.24
6.2.3.5 Wake Vortices

Placing a constant vector field over the velocity distribution done in Figure 99 to Figure 102 show the behavior of all the vortices. At a ground clearance of $H/c = 0.3$, the mid-span vortices start to build. The closer to the ground the wing gets moved, the stronger these vortices get. In addition, it can be seen that with decreasing ground clearance, the vortices do not only get stronger, they also increase in their number.
Figure 99: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.3

Figure 100: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.24

Figure 101: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.22

Figure 102: Wake vortices at a distance D/c=0.66 for ground clearance H/c=0.17
6.3 Conclusion of Phase 3 - A Wing Operating in Ground Effect

In conclusion of phase 3, the wing behaved as expected. The increase in lift coefficient could be seen up to a maximum point. The continuous drag increase showed the effect of increasing skin-friction drag and induced drag through the wake. The analysis of the wake showed that in ground effect, multiple span-wise vortices start to build and get stronger with decreasing ground clearance. This second benchmark solution of the S1223 wing profile is used in phase 4 to see the difference between operating in ground effect only vs. operating in ground effect in a wake. This case can be used as a solution if a race car operates on an open track with no other cars around.

7 Phase 4 – A Wing Operating in a Wake

To analyze the wing operating in a wake, the wing is placed into the bluff body’s wake. The created bluff body from phase 2 is taken and placed in front of the wing. The wing is similar to phase 3 placed near the ground, and wing and bluff body are placed as stationary objects and the surrounding air and ground is moved with its relative speed. The chosen distances between the wing and the bluff body are D/L = 0.1, 0.3, and 0.5. These distances are chosen as a result of phase 2. At a distance of D/L = 0.1, the wing is placed in the body wake which will have a strong influence. D/L = 0.3 features the transition region between body wake and the rear wing vortices, and at D/L = 0.5, the main influence are the wing tip vortices from the rear wing of the bluff body. For each of the chosen distances, a flow structure analysis is performed to show the different influences of the bluff body on the wing.
Last, the effect of the velocity on a wing operating in a wake is studied. Therefore, the most effective ground clearance is chosen for each of the distances. The lift and drag coefficient are analyzed on compared to the values of a wing operating in ground effect only.

7.1 Model

The bluff body as developed in phase 2 is used to create the wake. The wing dimensions are the same as in phases 1 and 3. The chord length is 300 mm, and the span is 1500 mm. Again, the ground clearance is normalized by the chord length of the wing. Further, the distance between the bluff body and the wing is normalized by a Formula 1 car length, which is 5300 mm. The overall setup can be seen in Figure 103.

![Figure 103: Model setup phase 4](image)

The bluff body is placed 1250 mm downstream in the far field. The far field is 4500 mm wide and 2000 mm in height. The length varies on the distance D/L to save computational resources for the cases with a smaller distance. The wheel of the bluff body is placed within the ground. A fillet of 20 mm is used around the intersection of wheel and ground to improve the meshing quality. As in the three phases before, the symmetry option is used. The ground is equipped with a moving component, and a vector defines the relative speed of the ground compared to the wing with 30 m/s in x-
direction. The rest of the far field outside walls are defined as free stream boundaries with a mach number $= 0.08816$, which is 30 m/s. The wing and the bluff body are defined as normal wall boundaries. The rear wing of the bluff body is split up from the rest of the bluff body. The reason is to have the possibility for local refinement. The Star CCM+ model is with the symmetry plane is shown in Figure 104.

![Figure 104: Star CCM+ symmetric model at H/c=0.3 and D/L=0.3](image)

### 7.1.1 Meshing

Several different mesh iterations were performed to find a mesh setting where the results do not change anymore to get a mesh independent result. This step was especially crucial because it was also an attempt to find a mesh with not too many cells since the computational resources reached their limit. The proven mesh settings of the wing were not changed in the process. Recall, the base size is chosen to be the cord
length of the wing, 0.3 m. A surface size refinement on the wing itself is set. The target size is set to be 0.9 % of the chord length with a minimum size of 0.5 %. The thickness of the prism layer is set to be 1.5 % with a number of 15 prism layers.

Four different mesh refinement blocks are implemented into the mesh setup. Three of them are building the refinement to carry the wake from the bluff body all the way to the wing and past the wing as shown in Figure 105 (a). Therefore, the refinement areas use similar cutouts which were already introduced in phase 2 to save a significant number of cells. Block 1 and 4 are carrying the wake and reach from the front of the bluff body all the way to mid wing. Block 2 is defined to catch the wake of the wing. This time, it is defined a little bigger than in phase 3 to catch wake influences from the bluff body if necessary. Block 3, shown in Figure 105 (b) is needed to catch the flow between the wing and the ground, similar to phase 3. Block 1, 2, and 3 have a refinement to a cell size of 2.8 % of the chord length to catch the wake. The wing refinement, Block 2, is set similar to phase 3 to be 0.8 % of the chord length. The far field cell size setting has a target of 35 % of the chord length.
These chosen refinements lead to a mesh which has a greater cell size in the far field, a small enough size around the bluff body and the wake of the bluff body to catch the flow features and a really small refinement around the wing to model the flow as accurate as possible to get the aerodynamic forces. These different stages of refinement are shown in Figure 106 to Figure 108. The number of cells varies between 18 and 30 million cells, depending on ground clearance H/c and the distance D/L.
7.1.2 Physics

The physical models are not changed compared to the previous phases. The segregated model worked satisfactory. A change in physical models is not recommended since the benchmark results, phase one to three, are done with the same settings. Changing the physical models would add another unknown factor to the results. All the used models can be found in Section 4.1.2 in Table 4.

7.1.3 Error Analysis

The error analysis is performed as usual for every single case to verify accuracy. As usual, the common residual and aerodynamic force trackers are analyzed and the solution checked to ensure it makes sense. If the solution was how it is expected, the
visual analysis is performed to check that the solution fully converged. Since the wake is an unsteady feature of these simulations, the aerodynamic forces and residuals show fluctuations.

The residuals, shown in Figure 109, shows that all the values of the governing equation and turbulence model equation fluctuate. The continuity, $Z$-, and $Y$-momentum equation residuals fluctuate at approximately 10% residuals. The $X$-momentum levels are between 2 and 3 percent whereas the energy equation fluctuates at approximately 1.5% residuals. The scalar dissipation rate ($sdr$) fluctuates around a value of 0.1% residuals and the turbulent kinetic energy ($Tke$) stays around 100%. Since all the residuals fluctuate around a certain level, it can be said that the solution converged.

![Figure 109: Residuals of governing equation, phase 4 at H/c=0.21 and D/L=0.5](image)

The aerodynamic forces are tracked during the simulation to analyze at which point the solution can be considered as converged. Since the wake is an unsteady solution within a steady state simulation, the lift and drag coefficient fluctuate throughout the whole simulation. However, as seen already in section 6.1.3, after approximately 700 to 800 iterations, the aerodynamic forces fluctuate at an almost steady level because the solution repeats itself. It can be seen that the drag coefficient, Figure 111, reaches an
acceptable level earlier than the lift coefficient. In the shown case in Figure 110, the lift coefficient fluctuates at approximately the same level for the last 200 iterations. Therefore, the solution is converged. The noted drag and lift coefficients in the study are the average value of the steady fluctuation part.

Figure 110: Lift coefficient tracker, phase 4 at H/c=0.21 and D/L=0.5

Figure 111: Drag coefficient tracker, phase 4 at H/c=0.21 and D/L=0.5

Last, the visual solution is analyzed. Therefore, the velocity distribution on the center plane is analyzed for no sudden and unexpected changes. The velocity analysis in Figure 112 shows that the flow is entering the model on a steady velocity which starts to get influenced by the bluff body which is an expected phenomenon. Further, at the top end of the far field, no unexpected changes are happening, and the wake of the bluff body and wing are carried out. In this case, the wakes do not completely recover
towards the end of the far field. The computational resources limited the size of the model. However, the wake is carried out way beyond the wing where it has no more impact on the flow around the bodies. Therefore, the simulations are accurate.

![Velocity distribution including far field, phase 4 at H/c=0.21 and D/L=0.5](image)

**Figure 112: Velocity distribution including far field, phase 4 at H/c=0.21 and D/L=0.5**

### 7.2 Results of a Wing Operating in a Wake

Lift and drag coefficient are analyzed throughout all three studied distances and compared to the free stream case in phase 1 as well as to the results in ground effect of phase 3. All results of the aerodynamic forces are stated as percentage changes compared to the free stream case. Individual flow structure analysis is performed for each of the different distance cases to support the findings of the force analysis.

#### 7.2.1 Lift Coefficient Study of a Wing Operating in a Wake

The calculated lift coefficient for all the different cases ran are compared to the freestream value of the S1223 wing profile observed in phase 1. All the values are normalized by the original lift coefficient and presented as percentage change.

Section 6.2.1 already showed that the lift coefficient is increasing by decreasing the ground clearance. The maximum lift coefficient was reached at a ground clearance level
of H/c = 0.22 with an increase of 46.4 %. Figure 113 shows that the comparison of all three distance level behind the bluff body and the undisturbed results of section 6.2.1. It can be seen that there is a significant loss of downforce at D/L = 0.1. The wing operating 10 % of a car length behind the bluff body reaches its maximum lift coefficient at a ground clearance of H/c = 0.22, which is equal to the result in undisturbed flow. While decreasing the ground clearance, the wing gains on downforce until it reaches its maximum of negative 54.6 % compared to the free stream. At very small ground clearance, H/c = 0.15, the downforce loss is 74.5 % and at larger ground clearance, H/c = 0.3, the loss can be identified at 77.3 %. This massive loss of downforce can be explained by the body wake. Within the wake of the bluff body, multiple vortices and low velocities exists. Therefore, the suction pressure and the pressure on top of the wing are reduced significantly, which can be seen in section 7.2.3. Moving the wing further away from the bluff body has a positive effect on the lift coefficient of the wing compared to a distance D/L = 0.1. The main difference between a wing operating in ground effect only and operating in a wake at distances of D/L = 0.3 and 0.5 is that the maximum is reached at a ground clearance of H/c = 0.25 vs. 0.22 in undisturbed flow. Further, it can be seen that the maximum increase is larger at D/L = 0.5 than 0.3. This shows that the wings are now operating outside of the main body wake but are still influenced by the disturbances coming from the bluff body.
Comparing the undisturbed results of section 6.2.1 with the results of a wing operating in a wake at a distance of $D/L = 0.5$ clearly shows in Figure 114 that the point of maximum lift coefficient is shifted from $H/c = 0.22$ to $H/c = 0.25$. The maximum increase on lift coefficient at a distance of $D/L = 0.5$ with a ground clearance of $H/c = 0.25$ is 47.1 % compared to the freestream results. This shows that the wake has a positive influence on the lift coefficient since the maximum at undisturbed flow is 46.4 %.

Further, this positive influence can be seen at larger ground clearances. However, at small ground clearances, the lift coefficient is smaller for a wing operating in a wake at a distance of $D/L = 0.5$ compared to undisturbed flow. At larger ground clearances, the increase of the lift coefficient is approximately 3% more when the wing is operating in a wake at a distance $D/L = 0.5$ compared to undisturbed operation. The fact that at large ground clearances a positive effect exists shows that the wing tip vortices have a positive influence. However, the negative effect at small ground clearances indicates
that the vortices produced near the ground have a negative influence on the wings’ lift coefficient.

Figure 114: Comparison of change of lift coefficient between D/L=0.5 and undisturbed flow

A wing operating in a wake at a distance of D/L = 0.3 and D/L = 0.5 shows for both cases a similar behavior. In both cases, the maximum lift coefficient is reached at a ground clearance of H/c = 0.25. It can be seen that the wing tip vortices of the bluff body have a larger impact on the wing placed at D/L = 0.5 than at 0.3, especially for large ground clearances. Figure 115 shows that the difference in lift coefficient is 13.7 % for a ground clearance of H/c = 0.4 but only 6.1 % at H/c = 0.27. The maximum increase of downforce vs the freestream case is 47.1 % at a distance of D/L = 0.5 and 41.8 % at D/L = 0.3.
A detailed flow structure analysis is given in sections 7.2.3 to 7.2.5 to support the findings of the above presented lift coefficients.

### 7.2.2 Drag Coefficient Study of a Wing Operating in a Wake

Decreasing ground clearance leads to an increase in the overall drag which was already found in section 6.2.2. Operating a wing in ground clearance shows that the drag coefficient increases by 63.2 % at the maximum lift coefficient ground clearance.

Comparing the three studies of a wing operating in a wake to a wing operating in ground effect only shows that the overall behavior is for all the cases the same as it may be seen in Figure 116. By decreasing the ground clearance, the drag increases. However, there is a notable difference between the cases. Placing a wing at a distance of D/L = 0.1, there is a significant decrease of drag, similar to the decrease of the lift coefficient. This decrease is due to the fact that lower velocities to very small velocities exists in the body wake of the bluff body. At the maximum downforce point, which is at a ground clearance of H/c = 0.22, the drag decreases by 80.3 % compared to the wing in
freestream. The reason for this significant decrease is the decrease of the skin-friction drag. Lower velocities around the wing results automatically in smaller skin-friction drag. The drag coefficient behaves similar for a wing placed at distances D/L = 0.3 and D/L = 0.5.

Figure 116: Change in drag coefficient vs. freestream for a wing operating in a wake

Comparing the change of drag coefficient vs. freestream for the wing placed at distances D/L = 0.3, D/L = 0.5 and undisturbed flow shows in Figure 117 that the decrease of the ground clearance has a greater impact on the freestream case than when the wing is operating in a wake. This is due to the fact that lower velocities are existing and the velocity increase in the wake is not as large as it is in undisturbed flow. Further, it can be seen that at large ground clearance, H/c = 0.4, the drag is higher at D/L = 0.5 than in undisturbed flow. At the point of maximum ground clearance, the drag increases by 21.9% at D/L = 0.3 and 35.1 % at D/L = 0.5. As seen in section 7.2.1, the maximum lift coefficient occurs for both of these cases at a ground clearance of H/c = 0.25. At the maximum lift coefficient point of the undisturbed case, H/c = 0.22, the
drag increases by 28.9 % for a wing placed at a distance of D/L = 0.3 and 38.6 % at D/L = 0.5.

Figure 117: Comparison of change of drag coefficient for D/L=0.3, 0.5 and undisturbed

### 7.2.3 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.1

Flow structure analysis is performed to understand why the aerodynamic forces behave the way described in Section 7.2.1 and 7.2.2. At a distance of D/L = 0.1, which is 10 % of a car length, the wing performs in the body wake of the bluff body. Velocity and pressure distribution show the surrounding flow on the center plane. The center plane streamlines give a good overview how the air flows around the wing and the bluff body. The combined wake of the wing and bluff body is analyzed at D/c = 0.66 behind the wing. The overall streamlines show the flow around the complete bluff body and wing.
7.2.3.1 Velocity Distribution

The overall velocity distribution shows the common high and low velocity regions which were found already in phases 1 to 3. Figure 118 shows the velocity distribution for the case where a wing is placed at a distance D/L = 0.1 behind the bluff body with a ground clearance of H/c = 0.25. The high velocity regions are underneath the bluff body, at the top edge of the bluff body, underneath the rear wing of the bluff body, and underneath the analyzed wing behind the bluff body. The low velocity regions include the stagnation region of bluff body, rear wing and analyzed wing as well as the wakes produced of these bodies.

![Velocity distribution on center plane at H/c=0.25 and D/L=0.1](image)

Figure 118: Velocity distribution on center plane at H/c=0.25 and D/L=0.1

Since the area of interest is not the flow around the bluff body, the velocity distribution is analyzed starting at the end of the bluff body to see the wing operating in a wake. Figure 119 compares the wake of the bluff body without a wing, Figure 119 (a), and a wing placed at D/L = 0.1 behind the body at a ground clearance of H/C = 0.25, Figure 119 (b), shows the lower part of the wake is pushed together and stops before the wing. Further, it can be seen that the body wake has a huge influence on the wing.
The velocity distribution around the wing in Figure 120 to Figure 124 shows somewhat similar behavior than in phase 3, a wing operating in ground effect. The high velocity region underneath the wing is still existing. The velocity scale is adjusted to the actual velocities acting around the wing. However, the velocity is significantly lower compared to the wing in undisturbed air, which decreases the suction underneath the wing and therefore results in a loss of downforce. Further, the overall surrounding velocity is lower than in undisturbed air. This also leads to the upstream in front of the wing. There are low velocity regions above the wing, too. This shows that separation occurs immediately after the leading edge, and the velocity distribution captures also show clearly that the wake is not steady. There is no real pattern of the wake produced of the wing. At ground clearance $H/c = 0.25$, 0.22, and 0.2 a significantly larger separation region underneath the wing can be seen compared to the ground clearance levels $H/c = 0.23$ and 0.21. A better overview of all the vortices acting can be seen in the velocity vector analysis. Due to the lower velocities compared to the undisturbed case in phase 3, the friction drag and stagnation region is smaller, which also results in lower overall drag.
Figure 120: Velocity distribution on center plane at H/c=0.25 and D/L=0.1

Figure 121: Velocity distribution on center plane at H/c=0.23 and D/L=0.1

Figure 122: Velocity distribution on center plane at H/c=0.22 and D/L=0.1
Figure 123: Velocity distribution on center plane at H/c=0.21 and D/L=0.1

Figure 124: Velocity distribution on center plane at H/c=0.2 and D/L=0.1

7.2.3.2 Velocity Vectors

The velocity vectors show strong vertices in the wake of the bluff body. These vortices reach to the leading edge of the wing where the upstream occurs from the high velocity coming from underneath the bluff body and get redirected upwards by the wing which can be seen in Figure 125 to Figure 127. Further, it can be seen that in all of the cases, vortices above the wing exist, which also have a strong upward component. This
upward flow has a negative impact on the downforce. All those vortices have direct impact on lift and drag coefficient.

Figure 125: Velocity vectors on center plane at H/c=0.25 and D/L=0.1

Figure 126: Velocity vectors on center plane at H/c=0.22 and D/L=0.1
7.2.3.3 Pressure Distribution

The pressure distribution of the model shows the expected high and low pressure regions: low pressure regions underneath the bodies and high pressure regions at the three stagnation points. There is also low pressure existing in the bluff body wake region which may be seen in Figure 128. It also can be seen that the pressure around the operating wing is significantly lower than around the bluff body.

Figure 127: Velocity vectors on center plane at H/c=0.21 and D/L=0.1

Figure 128: Pressure distribution on center plane of complete model at H/c=0.25 and D/L=0.1
However, the pressure around the bluff body is not the region of interest in this study. Therefore, the pressure scale is adjusted to the existing pressures around the operating wing in the wake.

The low pressure underneath the wing exists the same way as in undisturbed flow as it can be seen in Figure 129 to Figure 132. Nevertheless, due to lower speeds surrounding the wing, the magnitude of the negative pressure is not as high anymore which leads to weaker suction and therefore less downforce. Further, it can be seen that the pressure region above the wing is not as clear as it was in undisturbed flow. Because of the immediate separation after the leading edge, a low pressure area exists above the wing, too. The wake coming from the bluff body supports the low pressure and destroys the actual high pressure region above the wing.

Figure 129: Pressure distribution on center plane at H/c=0.25 and D/L=0.1

Figure 130: Pressure distribution on center plane at H/c=0.23 and D/L=0.1
Center Plane Streamlines

The streamlines plotted on the center plane clearly show the before mentioned upstream of the air coming from underneath the bluff body and gets redirected by the wing. Shown by the velocity vectors, the vortices between the bluff body and the wing can be seen as rotating air. Since the wake is not a steady component, the streamlines of the different cases show that the wake is change from the bluff body even in the regions where the wing does not have an impact. All the cases, Figure 133 to Figure 137, show the mentioned upstream and vortices, and the upstream meets with the flow coming from the top of the car and redirects the air backwards. During the velocity distribution analysis, it could have been seen that not all the cases have a big
separation region underneath the wing. Viewing the streamlines in cases of separation, the streamlines are following the ground and with new separation, there is an upwash visible.

Figure 133: Center plane streamlines on velocity distribution at H/c=0.25 and D/L=0.1

Figure 134: Center plane streamlines on velocity distribution at H/c=0.23 and D/L=0.1

Figure 135: Center plane streamlines on velocity distribution at H/c=0.22 and D/L=0.1
Comparing the wake of the bluff body only and the wake of the wing placed at $D/L = 0.1$, the main “mushroom” shape is still intact. Since the wake is an unsteady element of the solution, the shape is slightly changing whenever the simulation is stopped. The area of change is the lower part of the wake, where the wake of the bluff body and the wing match. The main difference which can be seen in Figure 138 is that the high velocity region centered has vanished. There is still a higher velocity existing centered but compared to the case without a wing, the area and velocity are significantly smaller. Further, it can be seen that the vortices generated originally from the wheels are not as strong anymore. Additionally to the weakened vortices of the wheels, the wing tip
vortices of the rear wing of the bluff body are weakened, too. These vortices are influenced by the generated upwash seen before. The low velocity region in the mushroom wake produced by the wing tip vortices are significantly weaker behind the wing. However, these wake upper wake influences are a result of the wing but do not directly impact the aerodynamic forces acting on the wing.

Figure 138: Wake velocity distribution (a) Bluff body only at D/L=0.1 (b) Behind a wing, wing placed at H/c=0.25 and D/L=0.1

Since the upper wake region is only influenced by the wings’ redirected air and does not have a primary impact on the aerodynamic forces, the focus is held to the lower wake region in Figure 139 to Figure 143. As mentioned before, the wing the area of the wing tip vortices is weakened. Whereas in undisturbed flow, the wing tip vortices represent themselves with significant higher velocities, in the wake those velocities are smaller when the wing is operating in a wake. However, at the same time the velocities increased compared to the wake of the bluff body itself. The two vortices, the wing tip vortex and the vortex coming from the wheel interact with each other, and the wing tip vortex accelerates the air. The mid span vortices are developing the same way as in undisturbed flow. At larger ground clearance, the low velocity span is more constant than at smaller ground clearances. This is where the induced drag increases.
Figure 139: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.1

Figure 140: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.23 and D/L=0.1

Figure 141: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.1

Figure 142: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.21 and D/L=0.1

Figure 143: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.2 and D/L=0.1
The mentioned change in vortices around the wing tip can be seen clearly in Figure 144. Whereas multiple vortices existing from the bluff body, the wing tip vortex is the dominant one when the wing is placed in the wake. The biggest change is that the far side vortex of the bluff body growth larger and pushes the other two to a minimum size near the ground. The wake of the wing causes the vortices to be more structured and to be close to the ground.

Figure 144: Wing tip vortices (a) bluff body only (b) wing Behind a wing, wing placed at H/c=0.25 and D/L=0.1

7.2.3.6 Streamlines

The streamlines around the bluff body with visible pressure distribution and analyzed wing shows that almost all disturbance is produced by the bluff body in front of the wing. The pressure distribution shows high pressure at the frontal areas, higher pressure on the top side of the rear wing, and also pressure above the wing, which could have been seen on the pressure distribution analysis. Figure 145 to Figure 148 show that there is not a huge difference between the different cases in terms of the streamline propagation. The bluff body wheels and also the body itself push the flow to the outside which mainly travels around the wing and starts to normalize itself far behind the body. Further, it can be seen that some of the streamlines traveling underneath the bluff body gets redirected upwards. This upwash flow, which could have been seen before in
several different analyses, weakens the trailing vortices from the rear wing of the bluff body compare to the results in phase 2.

Figure 145: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$ and $D/L=0.1$ (a) top view (b) 3D view

Figure 146: Streamlines around the model with shown pressure distribution with wing at $H/c=0.23$ and $D/L=0.1$ (a) top view (b) 3D view
7.2.4 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.3

After concluding the flow structure analysis of a wing operating in a wake at D/L = 0.1, the same analysis is performed for the cases at D/L = 0.3 which corresponds to 30 % of
a race car length. Center plane analyses for velocity, velocity vectors, pressure, and streamlines are performed, as well as an analysis of the wake and overall streamlines around the model.

7.2.4.1 Velocity Distribution

The velocity distribution analyzed at the center plane is similar to the cases with a wing placed 10% of a car length behind the bluff body. However, it can be seen in Figure 149 that the wing is now outside the body wake. The wake shows a lot of unsteady areas and vortices already at the center plane velocity analysis. Comparing different cases show that the wake is unsteady. The overall model shows the similar velocity regions such as low velocity within the wake of the bluff body, rear wing, and analyzed wing. Further, the high velocities underneath the bodies exist, too, and the common flow around the bluff body.

Figure 149: Velocity distribution on center plane of complete model at H/c=0.27 and D/L=0.3

Since the area of interest is already defined in Section 7.2.3 as the flow around the wing, a detailed analysis of the velocity distribution of the wing is carried out in Figure 150 to Figure 154. Therefore, the surrounding velocity scales are adjusted to the acting
velocities in this area. The wing shows the normal high velocity area between the wing and the ground. The closer the wing gets moved to the ground, the higher the velocity gets. The nature of the wake created by the bluff body forces the stagnation point moving from the leading edge at a ground clearance of H/c = 0.27 towards the upper surface. The highest stagnation point occurs at H/c = 0.24 and is located clearly on the top surface. The more the stagnation point is towards the upper surface, the higher the downforce gets since the stagnation point directs the force towards the ground. However, at H/c = 0.24 where the stagnation point is clearly on the top surface, the flow underneath the wing slows down. Since the lift coefficient is nearly the same at a ground clearance of H/c = 0.24 and 0.23, it can be seen that the positive effect from the stagnation point on the top surface gets offset by the lower velocities acting underneath the wing. Similar to the wing operating in ground effect only, section 6.2.3, it can be seen that the wake and its separation region of the analyzed wing grow the closure it gets to the ground.

![Velocity distribution on center plane at H/c=0.27 and D/L=0.3](image)

Figure 150: Velocity distribution on center plane at H/c=0.27 and D/L=0.3
Figure 151: Velocity distribution on center plane at H/c=0.25 and D/L=0.3

Figure 152: Velocity distribution on center plane at H/c=0.24 and D/L=0.3

Figure 153: Velocity distribution on center plane at H/c=0.23 and D/L=0.3

Figure 154: Velocity distribution on center plane at H/c=0.22 and D/L=0.3
7.2.4.2 Velocity Vectors

The velocity vectors located on the center plane visualize the different vortices existing in the wake of the bluff body. Section 7.2.3.2 showed already the vortices existing in the wake of the bluff body. However, the wing is now placed further back which does not contain the wake as much as it did at D/L = 0.1. High velocity is entering the wake region from underneath of the bluff body and the rear wing as shown in Figure 155. Multiple vortices can be seen between bluff body and analyzed wing. Further, the flow direction is changing once it gets towards the wing. One part of the air gets redirected upwards and another part pushed underneath the wing.

Figure 155: Velocity vectors of bluff body wake on center plane at H/c=0.27 and D/L=0.3

The wake region of the bluff body has a consistent visual appearance throughout all the cases run at D/L = 0.3. However, some changes could be identified around the wing itself. Therefore, the region of emphasis of the velocity vector analysis is the wing itself.

The velocity vectors visualize the exact direction of the flow. It can be seen that at a ground clearance of H/c = 0.27, the air has an upstream component coming into the area of the wing shown in Figure 156. By decreasing the ground clearance, the influenced air in front of the wing starts getting a downward component which leads to
the movement of the stagnation point. At a ground clearance of H/c = 0.26, it can be seen in Figure 157 that the air is coming in a downward angle. This downward component increases and leads to vortex in front of the wing at H/c = 0.25, which forces the flow hitting the wing on an angle which may be seen in Figure 158. This leads to the higher stagnation point and also to an attached flow on the top surface of the wing as well as underneath the wing. Moving the wing further down, Figure 159 shows that at a ground clearance H/c = 0.24 the flow comes even on a larger downward angle. However, at an even smaller ground clearance, H/c = 0.23, the flow normalized itself and the local angle of attack decreases again as seen in Figure 160.

Figure 156: Velocity vectors around the wing on center plane at H/c=0.27 and D/L=0.3

Figure 157: Velocity vectors around the wing on center plane at H/c=0.26 and D/L=0.3
Figure 158: Velocity vectors around the wing on center plane at H/c=0.25 and D/L=0.3

Figure 159: Velocity vectors around the wing on center plane at H/c=0.24 and D/L=0.3

Figure 160: Velocity vectors around the wing on center plane at H/c=0.23 and D/L=0.3
7.2.4.3 Pressure Distribution

The pressure distribution is an illustration of the velocity distribution. It is known that velocity increases result in low or even negative pressure and velocity decreases in high pressure. The pressure distribution of the model in Figure 161 shows nothing unexpected. The pressure around the bluff body is similar to the cases before. The stagnation pressure on the bluff body and the rear wing of the bluff body can be seen as well as the low pressure regions underneath the bodies.

Figure 161: Pressure distribution on center plane of complete model at H/c=0.27 and D/L=0.3

Seen in the different analyses before, the region of interest and change is limited to the wing itself. Therefore, the pressure scale has been adjusted to the surrounding pressures of the wing.

The pressure distribution around the wing shown in Figure 162 to Figure 166 clearly shows the movement of the stagnation point. All cases show the expected high and low pressure regions. While at a ground clearance of H/c = 0.27, the pressure distribution is similar to the free stream cases, where a slightly higher pressure is acting on the top surface and suction is acting on the lower surface. Further, at this height, the stagnation
point is located at the leading edge of the wing profile. As seen before, the stagnation point is moving upward, which causes higher pressure on the top surface. The stagnation point reaches its highest point at a ground clearance of \( H/c = 0.24 \), which results in a high pressure distribution along the complete wing span. However, as seen before, the suction underneath decreases since lower velocities are existing due to the fact that the flow is entering at a high angle. Normalization of the pressure distribution occurs at \( H/C = 0.23 \), where the stagnation point moves back towards the leading edge, which results in higher suction underneath the wing but also lower pressure on the top surface. The high pressure on the top surface and the strength of the suction underneath the wing have a positive impact on the downforce. It can be seen that the suction has a great impact since high pressure exists all over the top surface at a ground clearance of \( H/c = 0.24 \) but does not result in the highest amount of downforce as seen in section 7.2.1.

![Figure 162: Pressure distribution on center plane at H/c=0.27 and D/L=0.3](image)

Figure 162: Pressure distribution on center plane at H/c=0.27 and D/L=0.3
Figure 163: Pressure distribution on center plane at H/c=0.26 and D/L=0.3

Figure 164: Pressure distribution on center plane at H/c=0.25 and D/L=0.3

Figure 165: Pressure distribution on center plane at H/c=0.24 and D/L=0.3
7.2.4.4 Center Plane Streamlines

The streamlines located on the center plane give a good overview how the air flows around the model. The different features discussed in the velocity distribution, velocity vector and pressure distribution analysis are visualized again by the streamlines. Since the bluff body does not change between the different cases, the flow around the bluff body and its wake is similar. The mentioned vortices of the wake can be seen clearly in Figure 167.

As before, the area of interest is the wing placed at D/L = 0.3. Therefore, the streamlines around the wing are shown below. All streamlines are plotted on top of the velocity distribution.
The constraint streamlines on the center plane in Figure 168 show that at $H/c = 0.27$, the flow is coming from the ground and has an upwash component. However, the upwash is not as strong as at distance $D/L = 0.1$. The flow is still attached on the wing. While decreasing the ground clearance, it can be seen that the angle of the incoming flow is changing which moves the stagnation point towards the top surface which may be seen in Figure 168 to Figure 173. However, it can be seen that a separation of the flow occurs on the top surface. At a ground clearance of $H/c = 0.24$ and $0.23$, the flow is attached along the top surface again, and the upwash component of the flow is reduced significantly. At low ground clearance, $H/c = 0.22$, separation occurs again on the top surface.

Figure 168: Center plane streamlines around the wing at $H/c=0.27$ and $D/L=0.3$

Figure 169: Center plane streamlines around the wing at $H/c=0.26$ and $D/L=0.3$
Figure 170: Center plane streamlines around the wing at $H/c=0.25$ and $D/L=0.3$

Figure 171: Center plane streamlines around the wing at $H/c=0.24$ and $D/L=0.3$

Figure 172: Center plane streamlines around the wing at $H/c=0.23$ and $D/L=0.3$

Figure 173: Center plane streamlines around the wing at $H/c=0.22$ and $D/L=0.3$
7.2.4.5 Wake analysis

Analyzing the wake velocity distribution of bluff body and compare it to the case where the wing is placed with a ground clearance of $H/c = 0.27$ at a distance of $D/L = 0.3$, it can be seen in Figure 174 that the main changes are located near the ground where the wake of the bluff body and the wake of the wing are combined. The found “mushroom” head still exists after placing a wing into the wake. However, the boundaries of the mushroom head are not as clear anymore. Nevertheless, a significant change can be seen near the ground. The wake of the wing is indicated by wing tip vortices of the analyzed wing. Further, past the wing, the disturbed air coming from the wheels of the bluff body can be seen.

![Figure 174: Wake velocity distribution (a) Bluff body only at D/L=0.3 (b) Behind a wing, wing placed at H/c=0.27 and D/L=0.3](image)

Comparing the vortices of the two cases shows that the strength of the wing tip vortices decreased as it may be seen in Figure 175. This is due to the interaction of the wing which redirects the air upwards. However, it can be seen that the most change is happening near the ground in the wake of the wing. The vortices near the ground indicate the merging of the wing wake and the bluff body wake.
Figure 175: Wake vortices (a) Bluff body only at D/L=0.3 (b) Behind a wing, wing placed at H/c=0.27 and D/L=0.3

As shown above, the main changes of the wake are near the ground. Therefore, the area of interest is wake and its vortices near the ground. At close inspection, the wake of the wing and bluff body near the bottom are very similar to the results in phase 3, a wing operating in ground effect. Nevertheless, a few differences can be seen. First, the wing is placed in the wake of the bluff body. As mentioned before, the bluff body wake did not change that much between phase 2, creation of a bluff body, and a wing placed at a distance ratio of D/L = 0.3 behind the car. Unless to the section 7.2.3, where the wing was placed close to the bluff body, this time the actual main elements of the wing wake are clearly visible. The wing tip vortices can be seen as two low velocity points, similar to the results in phase 3. Further, another similarity is that more disturbance occurs while the ground clearances decreases. Moreover, the high velocity regions are developing similarly. However, the overall velocities obtain a decrease, which is due to lower overall velocities acting around the wing due to the wake of the bluff body. The wing tip vortices delimit themselves clearly from the disturbance coming from the wheels of the bluff body. Figure 176 to Figure 182 show the velocity distribution of the wake near the ground where to wake of the wing merges with the wake of the bluff body.
Figure 176: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.27$ and $D/L=0.3$

Figure 177: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.26$ and $D/L=0.3$

Figure 178: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.25$ and $D/L=0.3$

Figure 179: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.24$ and $D/L=0.3$

Figure 180: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.23$ and $D/L=0.3$
Figure 181: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.3

Figure 182: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.2 and D/L=0.3

The wake vortices near the ground in Figure 183 to Figure 186 clearly show the wing tip vortices. Further, the additional vortices coming from the bluff body, which is a wider than the wing, can be seen, too. These additional vortices have their origin from the wheels of the bluff body. It also can be seen that more span wise vortices are growing while the ground clearance is reduced. At a ground clearance of H/c = 0.24, the outside vortices from the bluff body wheels are starting to get weaker. This is due to the changed flow. Recall from the velocity distribution and the center plane streamlines, a significant vortex builds up in front of the wing which changes the flow direction and redirects the air with a strong downward component on the wing. The downward flow leads to the weakening of the outside vortices with decreasing ground clearance.

Figure 183: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.3
Figure 184: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.3

Figure 185: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.3

Figure 186: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.3

7.2.4.6 Streamlines

The streamlines visualize the flow around the complete model. Recall the findings from previous analyses, the rear wing of the bluff body creates two wing tip vortices circling against each other since the air flows on both sides from high pressure, upper surface, to low pressure. These wing tip vortices are carried through the whole domain as shown in Figure 187 (a). The findings of a wing operating in a wake at D/L = 0.1 include the model shown in Figure 187 (b), the streamlines around the bluff body and a wing placed at a ground clearance of H/c = 0.22 at a distance D/L = 0.1. It was found that the wing tip vortices are strongly influenced by the upwash created through the wing placed close
to the bluff body. Therefore, the main wing tip vortices got destroyed or are not as clear anymore as they used to be with no further disturbance downstream.

![Streamlines](image)

Figure 187: Streamlines around the model (a) bluff body only (b) a wing placed at \(H/c=0.22\) and \(D/L=0.1\)

Analyzing the different ground clearance levels at a distance \(D/L = 0.3\) shows that the overall flow around the model is not changing much between the cases which can be seen in Figure 188 to Figure 191. The pressure distribution shows no unexpected features. All the leading edges or faces receive high pressure due to the stagnation of the air. Further, the top surfaces of the wing have a positive pressure whereas underneath the wings negative pressure or suction exists. The flow around the edges of the bluff body forces low pressure on the edges of the body. The streamlines are extending around the body and recover to its normal width downstream after the wing. Unlike the cases at a distance \(D/L = 0.1\), the wing tip vortices of the rear wing stay intact and have no changes compare to the case without a wing.
Figure 188: Streamlines around the model with shown pressure distribution with wing at $H/c=0.27$ and $D/L=0.3$ (a) top view (b) 3D view

Figure 189: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$ and $D/L=0.3$ (a) top view (b) 3D view
Figure 190: Streamlines around the model with shown pressure distribution with wing at $H/c=0.24$ and $D/L=0.3$ (a) top view (b) 3D view

Figure 191: Streamlines around the model with shown pressure distribution with wing at $H/c=0.22$ and $D/L=0.3$ (a) top view (b) 3D view
7.2.5 Flow Structure Analysis of a Wing Operating in a Wake at D/L = 0.5

A half of a car length distance, D/L = 0.5, the flow structure is analyzed similar to the two distances before. Analog to the sections before, section 7.2.3 and 7.2.4, the velocity distribution, pressure distribution, velocity vectors, and constraint streamlines are analyzed on the center plane. Further, the wake and its vortices are analyzed on a plane at a distance D/c = 0.66 behind the wing. Last, streamlines around the whole model show the overall flow around the bluff body and the wing.

7.2.5.1 Velocity Distribution

The velocity distribution on the center plane shows the same main features as the previous studies. The flow around the bluff body has not changed. The stagnation point on the leading edge of the bluff body can be seen clearly in Figure 192. Further, around the edges and underneath the bluff body and the rear wing, high velocity regions are present. The main wake velocity distribution of the wake of the bluff body looks the same as in the cases before. However, it can be seen that the wing is now clearly behind the body wake. Nevertheless, the disturbance of the bluff body has an effect on the wing placed at a distance of D/L = 0.5. Multiple low velocity regions can still be observed between the body wake and the end of the model. Further, in phase two, the creation of the bluff body, it could be seen that there are still vortices affecting the flow at this distance behind the bluff body.
Since the analyzed part is the wing placed at a distance of $D/L = 0.5$, the area of interest is the flow around the wing itself. Further, no changes between the different ground clearances of the wing could be seen in the wake of the body. Therefore, the velocity distribution is analyzed closer around the wing itself, and the velocity scales are adjusted to that specific region.

As seen in other velocity distribution analyses before, the wing builds a duct with the ground which leads to an increase of the velocity underneath the wing. Figure 193 to Figure 198 shows an increase of the velocity while decreasing the ground clearance. Since the wing is placed outside the main body wake, the wing shows the normal high velocity region underneath the wing, the low velocity of the wake and a low velocity region above the wing. At a wing placed at $D/L = 0.5$, the low velocity above the wing looks similar to the undisturbed case in section 6.2.3.1 where at $D/L = 0.3$, extremely low to zero velocity was seen above the wing due to the wake. The velocity increase is approximately 1.5 % per 1 % the ground clearance gets reduced whereas in undisturbed flow the velocity increase per 1 % ground clearance decrease is approximately 1.3 %.
Figure 193: Velocity distribution on center plane at $H/c=0.26$ and $D/L=0.5$

Figure 194: Velocity distribution on center plane at $H/c=0.25$ and $D/L=0.5$

Figure 195: Velocity distribution on center plane at $H/c=0.24$ and $D/L=0.5$

Figure 196: Velocity distribution on center plane at $H/c=0.23$ and $D/L=0.5$
7.2.5.2 Velocity Vectors

As seen in previous flow structure analyses, the velocity vectors give a good overview of the vortices existing in the wake of the bluff body. The main difference to the cases before, where the wing was placed behind the bluff body at a distance of D/L = 0.1 and 0.3, is that the vortices of the main body wake do not reach all the way to the wing. All the different ground clearance level showed the same type of vortices which all are recovered by the time the flow hits the wing as shown in Figure 199.
Since the aerodynamic forces are analyzed on the wing operating in a wake at a distance of \( D/L = 0.5 \), the area of interest is once more the flow around the wing itself. Therefore, a few cases are analyzed at the wing itself. Recall from the previous distances analyzed, a wing placed at a distance of \( D/L = 0.1 \) forces the air into a strong upwash, which was weakened when the wing was placed at \( D/L = 0.3 \) but still existed. Further, at the distance of \( D/L = 0.3 \), a vortex was build up in front of the wing which changed the angle of the flow and led to a downward component in the flow direction.

Comparing the velocity vectors, the visual direction of the flow on the center plane in Figure 200 to Figure 204, it can be seen that the flow direction is not really changing by changing the level of ground clearance. The velocity vectors indicate the flow is hitting the wing almost horizontally, which is similar to the undisturbed case. However, there is still a small upwash component due to the nature of the wake. However, the stronger upwash is located upstream (from the wing) in the wake, which mean the influence of it is not as big as it was at smaller distances. The constraint streamlines, section 7.2.5.4, give a clear view where the upwash is located. Therefore, the velocity vector analysis clearly shows that the wing starts to recover towards the undisturbed case.
Figure 200: Velocity vectors around the wing on center plane at H/c=0.26 and D/L=0.5

Figure 201: Velocity vectors around the wing on center plane at H/c=0.25 and D/L=0.5

Figure 202: Velocity vectors around the wing on center plane at H/c=0.24 and D/L=0.5

Figure 203: Velocity vectors around the wing on center plane at H/c=0.23 and D/L=0.5

Figure 204: Velocity vectors around the wing on center plane at H/c=0.21 and D/L=0.5
7.2.5.3 Pressure Distribution

The pressure distribution shows the common features of low and high pressure areas around the complete model in Figure 205. The bluff body has the common effect of stagnation pressure on the leading edge of the body and rear wing as well as a low pressure region behind. The low pressure region is an illustration of the wake produced by the bluff body. Further, it can be seen that the low pressure region of the wake propagates towards the end of the model with an upwash. This is due to the nature of the wake. As already seen in previous analyses, the bluff body creates an upwash wake, which is clearly visible on the constraint streamlines in the next section.

![Pressure distribution on center plane of complete model at H/c=0.26 and D/L=0.5](image)

Figure 205: Pressure distribution on center plane of complete model at H/c=0.26 and D/L=0.5

Like in previous analysis, the area of interest is the wing behind the bluff body placed at a distance D/L = 0.5 at various ground clearance levels. The pressure distribution on the center plane is very similar to the cases in undisturbed flow. The main elements of high and low pressure are the same which is shown in Figure 206 to Figure 210. Unless the cases at a distance of D/L = 0.3, where the stagnation point was clearly moving up and down on the leading edge, the stagnation point is now constant at the same position. This is due to the smaller influence of the wake since the wing is not placed in the
immediate body wake anymore. Similar to the undisturbed flow cases, the suction or negative pressure increase the closer the wing gets to the ground. This is due to the higher velocity forced by the so called duct the wing and ground build together. The main difference is the stagnation pressure which does fluctuate. This fluctuation can be explained by the fact that the wake is not a steady phenomenon. Therefore, if the air hits the wing more disturbed, the stagnation pressure decreases. If the wing is hit almost undisturbed, the stagnation pressure increases to the same pressure as acting on the bluff body itself, which is approximately 697 Pa.

Figure 206: Pressure distribution on center plane at H/c=0.26 and D/L=0.5

Figure 207: Pressure distribution on center plane at H/c=0.25 and D/L=0.5
Figure 208: Pressure distribution on center plane at H/c=0.24 and D/L=0.5

Figure 209: Pressure distribution on center plane at H/c=0.23 and D/L=0.5

Figure 210: Pressure distribution on center plane at H/c=0.22 and D/L=0.5
7.2.5.4 Center Plane Streamlines

The center plane streamlines plotted on the velocity distribution give a good overview about the nature of the flow around the complete model. Figure 211 indicates all the different stages of the wake and the main character of the streamlines does not change between the different ground clearance cases. The vortices of the body wake can be seen clearly behind the bluff body. Moreover, the upwash which was described earlier can be seen starting behind the vortices of the body wake. However, the further downstream it goes, the weaker that upwash gets. At a distance $D/L = 0.5$, the leading edge of the wing, the upwash near ground is weaker than further away from the ground.

Figure 211: Center plane streamlines on velocity distribution at $H/c=0.26$ and $D/L=0.5$

Comparing the two cases of center plane streamlines around the wing at a ground clearance of $H/c = 0.24$ for the case where the wing is placed in undisturbed flow and at a distance of $D/L = 0.5$ shows a lot of similarities. Figure 212 shows that the flow is in both cases attached to the wing on the top surface and has a separation region on the suction surface. The two visible differences between the two cases are the streamlines around the wake and the direction of the incoming streamlines. The streamline propagation in the wake is a momentary screen shot since the wake is an unsteady element of this situation. Therefore, it can be considered as a similar case. However,
the incoming streamlines at undisturbed flow are horizontally whereas the streamlines of the wake have a small upwards or downwards component.

Figure 212: Center plane streamlines around wing (a) at H/c=0.24 and D/L=0.5 (b) at H/c=0.24 in undisturbed flow

Now comparing a few cases of different ground clearance levels operating in a wake at a distance of D/L = 0.5 behind the bluff body. Figure 213 to Figure 215 show similar behavior of the streamlines. The only visual difference are the incoming streamlines. The upward component changes a little, which can lead to a small downward component. Again, the wake is an unsteady component and therefore this difference does not really come into play in the overall behavior.

Figure 213: Center plane streamlines around the wing at H/c=0.25 and D/L=0.5
7.2.5.5 Wake analysis

Comparing the wake velocity distribution of the bluff body only of section 5.2 at a distance of $D/L = 0.5$ to the wake velocity distribution of a wing placed behind the bluff body at a ground clearance of $H/c = 0.25$ and a distance of $D/L = 0.5$ shows a lot of similarities but also some differences. The wake velocity distribution is taken on a plane with a distance of $D/c = 0.66$ behind the wing. The overall velocities are higher in the case without a wing. Within the flow structure, the main characters look the same. The wing tip trailing vortices from the bluff body exist in both cases. The main difference in the flow structure lies on the bottom where the wing wake matches with the wake of the bluff body. In the case of the bluff body without wing, some disturbance on the ground can be seen clearly. Figure 216 (a) shows the wake of the wing clearly placed into the wake from the bluff body. The wake of the wing has its normal occurrence of wing tip
vortices and mid-span vortices. On each side of the wing, the disturbance from the bluff body can still be seen.

Figure 216: Wake velocity distribution (a) Behind a wing, wing placed at H/c=0.25 and D/L=0.5 (b) Bluff body only at D/L=0.5

Having a look on the vortices in Figure 217, it can be seen that the wing tip vortices are clearly intact. However, near the ground, the vortices have changed and due to the wake of the wing in Figure 217 (a), multiple more vortices occurred.

Figure 217: Wake vortices (a) Behind a wing, wing placed at H/c=0.25 and D/L=0.5 (b) Bluff body only at D/L=0.5

As seen above, the main difference in velocity distribution and vortices is located near the ground. Since the upper part of the wake velocity is not affected by the ground clearance of the wing, the area of interest is where the wake of the wing and the wake of the bluff body joins. The velocity distribution in Figure 218 to Figure 221 clearly shows the wake of the wing surrounded by the wake of the bluff body. The wing tip
vortices of the wing are clearly visible. A difference is the additional vortices originating from the bluff body outside of the wing’s vortices. At a ground clearance of $H/c = 0.26$, next to the wing tip vortices, an additional vortex can be seen. While decreasing the ground clearance, this additional vortex gets deformed. This is due to the fact that the wing tip vortices are stronger influenced the closer they get to the ground. This leads to the observation that this additional vortex gets pushed towards the outside. However, it is believed that it is not strong enough to have a major impact on the aerodynamic forces acting on the wing. A larger role are the mid-span vortices which could already be seen in section 6.2.3. The low velocity region in span wise direction is growing in width and size while decreasing the ground clearance. This low velocity region is the indication of a growth in the wake which increases the induced drag and has a negative effect on the downforce as well.

Figure 218: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.26$ and $D/L=0.5$

Figure 219: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.25$ and $D/L=0.5$
Figure 220: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.5

Figure 221: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.5

Having a look on the actual vortices in the analyzed ground region, it can be seen in Figure 222 to Figure 225 that the wing tip vortex is rotating from the middle to the outside and from top to bottom, which means the air is going from the high pressure region above the wing to the low pressure or suction region underneath the wing. However, the outside vortices next to the wingtip vortices rotating the other side around. This leads to an increase of the strength of the wing tip vortices which has a negative effect on the lift coefficient. Further, the closure the wing gets to the ground the more vortices start to appear in the mid-span region, which increases the wake strength and has therefore a negative effect on the aerodynamic forces acting on the wing. It can be observed that the center of the wing tip vortices is moving towards the ground while decreasing the ground clearance. However, the additional outside vortex keeps its center position more or less, indicating that these outside vortices have only a secondary influence, the strength of the wing tip vortices
Figure 222: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.26 and D/L=0.5

Figure 223: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25 and D/L=0.5

Figure 224: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.24 and D/L=0.5

Figure 225: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.22 and D/L=0.5

7.2.5.6 Streamlines

The streamlines visualize the complete flow around the model. The streamline analysis in section 7.2.3.6, the wing placed at a distance D/L = 0.1, showed that the wing tip
trailing vortices are getting destroyed by the upwash the wing produces. Section 7.2.4.6, the streamline analysis of a wing placed at a distance of D/L = 0.3 behind the bluff body, that the wing tip trailing edge vortices are mostly intact but are still interfered by the flow around the wing. Figure 226 shows the streamlines for (a) the bluff body only and (b) for the model where the wing is placed at a ground clearance of H/c = 0.22 and at a distance of D/L = 0.5 behind the bluff body. Comparing these two cases shows that the wing tip vortices are intact and go through the whole model. However, it can be seen that the wing tip trailing vortices still receive influence from the wing placed behind the bluff body.

![Figure 226: Streamlines around the model (a) bluff body only (b) a wing placed at H/c=0.22 and D/L=0.5](image)

Comparing the streamlines of different cases in Figure 227 to Figure 230 shows that the overall flow does not change. The ground clearance has no real effect on the wake coming from the bluff body. All the cases show the expansion of the flow around the bluff body and the normalization downstream. As mentioned before, the wing itself has a minimal influence on the wing tip vortices originated from the modeled rear wing of the bluff body. The pressure distribution on the wing and bluff body shows its normal behavior. The leading edges and faces of the bodies are marked as high pressure
regions whereas the edges of the bluff body and underneath the body low pressure exists. Further, the top surfaces of the wing receive pressure whereas underneath suction occurs.

Figure 227: Streamlines around the model with shown pressure distribution with wing at H/c=0.26 and D/L=0.5 (a) top view (b) 3D view
Figure 228: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$ and $D/L=0.5$ (a) top view (b) 3D view

Figure 229: Streamlines around the model with shown pressure distribution with wing at $H/c=0.24$ and $D/L=0.5$ (a) top view (b) 3D view
Figure 230: Streamlines around the model with shown pressure distribution with wing at H/c=0.22 and D/L=0.5 (a) top view (b) 3D view

7.3 The Effect of Velocity on a Wing Operating in a Wake

The effect of velocity on a wing operating in a wake is studied for the S1223 wing profile at its most effective ground clearance at various speeds. As it was seen in section 7.2.1, the most effective ground clearance changes between a wing operating in undisturbed air and at the different distances. The most effective ground clearance for this particular wing studied is for undisturbed air at a ground clearance of H/c = 0.22. While the wing is operating in a wake, the most effective ground clearance stays at H/c = 0.22 when the wing is placed at a distance of D/L = 0.1. Moving the wing further away from the bluff body increases the most effective ground clearance to H/c = 0.25. To investigate the influence of the velocity on a wing operating in a wake, the wing is studied at three different speed settings for each distance behind the bluff body. The three different
speeds are chosen to be 30 m/s, 60 m/s, and 90 m/s which are common race car speeds around a track. The respective Reynolds number can be found in Table 5.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Distance D/L=0.1</th>
<th>Distance D/L=0.3</th>
<th>Distance D/L=0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ground Clearance [H/c]</td>
<td>0.22</td>
<td>0.25</td>
<td>0.25</td>
</tr>
<tr>
<td>Velocity 1 [Re]</td>
<td>600,000</td>
<td>600,000</td>
<td>600,000</td>
</tr>
<tr>
<td>Velocity 2 [Re]</td>
<td>1,200,000</td>
<td>1,200,000</td>
<td>1,200,000</td>
</tr>
<tr>
<td>Velocity 3 [Re]</td>
<td>1,800,000</td>
<td>1,800,000</td>
<td>1,800,000</td>
</tr>
</tbody>
</table>

The Star CCM+ model is not changing compared to the rest of the cases in phase 4 – a wing operating in a wake. However, the freestream velocity of the far field is adjusted as well as the velocity of the moving ground.

7.3.1 Lift Coefficient Study on Velocity Effect

The study of different velocities shows that the velocity has an impact on the aerodynamic forces acting on the wing. Since the size of the wake is change while increasing the velocity, the influence of the wake on the wing place downstream is subjected to change.

Looking at Figure 231, it can be seen that depending on the distance between the bluff body and the wing, the effect of the velocity change is different. Since the wing is operating under strong wake influence at a distance of D/L = 0.1, the changes are not as significant at larger distances. While at a velocity of Re = 600,000, the loss of
downforce at the most effective ground clearance is 69.01 %. By increasing the velocity, the loss of downforce increases to 74.35 % for a Reynolds number of Re = 1,200,000 and lastly to 72.51 % for a velocity equal to Re = 1,800,000. The change of the loss of downforce can be explained by minor changes to flow structure of the wake due to the different velocities. However, the flow structure analysis in section 7.3.3.1 shows that all three cases look very similar. A significant change can be seen where the wing is placed at a distance of D/L = 0.3 downstream. At low speed, the lift coefficient has a minimal increase of 0.11 % compared to undisturbed flow because the wing is not operating in the main body wake at Re = 600,000. By increasing the velocity and its corresponding Reynolds number, the wake region growth may be seen in section 7.3.3.2. This growth affects the flow structure around the wing and leads to a decrease of 12.03 % in downforce at Re = 1,200,000 and 14.43 % at Re = 1,800,000 compared to the wing operating in ground effect only. The flow structure analysis shows that at high speed the body wake reaches all the way back to the leading edge of the wing and therefore has a significant influence on the flow around the wing. A half of a car length, D/L = 0.5, downstream the wake has a positive impact on the lift coefficient as already seen in section 7.2.5 where the wing tip trailing vortices were identified as a positive influence. At a velocity of 30 m/s, corresponding to Re = 600,000, the lift coefficient is increasing by 3.88 % compared to the case where the wing is operating in ground effect only. By increasing the velocity, the body wake does not only grow in length, the wing tip vortices also gain in strength which can be seen in the flow structure analysis in section 7.3.3.3.1 to 7.3.3.3.4. Since the wing is now placed far enough downstream, the velocity increase from 30 m/s to 60 m/s has an actual positive influence on the lift
coefficient. The lift coefficient increases by 4.79 % compared to undisturbed operation in ground effect only. For high velocities, 90 m/s and its corresponding Reynolds number of Re = 1,800,000, the positive effect of the weak gets weakened, and the lift increase decreases to 0.51 %.

Figure 231: Change in lift coefficient vs freestream case at various velocities

Changing the velocity arises a change in the wake. Mainly, the wake is growing in size and therefore the behavior of the wing. It can be seen that changing the velocity from 30 m/s to 90 m/s when the wing is placed at a distance of D/L = 0.5 downstream, the lift coefficient acts similar to the case where the velocity is set to be 30 m/s and the wing placed at D/L = 0.3. This is due to the fact that the relationship between wake length and lift coefficient behavior is similar in both cases.

7.3.2 Drag Coefficient Study on Velocity Effect

The drag coefficient in Figure 232 shows that by increasing the velocity the drag coefficient is decreasing for a wing placed at the same distance downstream and a
constant ground clearance. Since the overall drag is an addition of stagnation drag, skin-friction drag, and induced drag, the behavior of the drag coefficient can be analyzed on these three components.

Placing the wing at $D/L = 0.1$ downstream, the drag reduction is significant as already seen in section 7.2.2. However, by increasing the velocity, the drag decreases even more in percentage compared to a wing operating in ground effect only. At a velocity of 30 m/s, the drag reduction is 87.85 %, at 60 m/s its 90.67% and at 90 m/s, the reduction comes to be 91.09%. The flow structure analysis in section 7.3.3.1 shows that the velocity increase underneath the wing is less the higher the velocity gets, which results in a decrease of skin-friction drag. Since the relative velocity within the wake is slower at high speed compared to low speed, the stagnation drag is reduced as well. The same principle can be seen when the wing is placed at a distance of $D/L = 0.3$. Since the overall velocities around the wing are significantly higher compared to the cases at a distance of $D/L = 0.1$, the loss of drag is not as large anymore. At a velocity of 30 m/s, the drag reduction is 16.77 %, at 60 m/s its 33.19% and at 90 m/s, the reduction comes to be 39.42%. Placing a wing at a distance of $D/L = 0.5$ downstream results in a drag reduction as well when the velocity is increased. A Reynolds number of $Re = 600,000$ which corresponds to 30 m/s results in 7.78 % drag reduction. Further, by doubling the velocity, the drag reduction is 8.32 % and at a velocity of 90 m/s, the drag coefficient decreases by 10.48 % compared to a wing operating in ground effect only.
Changing the velocity results in growth of the wake in length downstream. This wake increase reflects itself with lower relative velocities around the wing which decreases the stagnation and skin-friction drag.

### 7.3.3 Flow Structure Analysis

Similar to section 7.2.3 to 7.2.5, the flow structure is analyzed for the cases at different speeds. The overall behavior of the flow around the bluff body and the wing is not changing. However, the main body wake is growing while the speed is increasing. The flow structure analysis is split up again for the different distances of a wing placed behind the bluff body.

#### 7.3.3.1 Flow Structure Analysis at Various Speeds at a Distance D/L = 0.1

The flow structure analysis is performed on the center plane for the velocity distribution and the constraint streamlines. The wake is analyzed on a cross-section plane at a
distance $D/c = 0.66$ behind the wing. Learned from the previous analysis, the pressure analysis is an illustration of the velocity. Therefore, the main difference can be seen in the velocity analysis on the center plane and the wake as well as the various different streamlines to capture the overall flow.

### 7.3.3.1.1 Velocity Distribution on Center Plane

The velocity distribution on center plane in Figure 233 captures the wake region of the different speed cases where the wing is placed at a ground clearance of $H/c = 0.22$ and a distance of $D/L = 0.1$ behind the bluff body. The main characteristics look the same for all three cases. The wing is placed in the body wake of the bluff body and is therefore strongly influenced by the wake. However, the variation of velocity does not affect the overall velocity distribution except the growth in length of the wake. The upwash in front of the wing is still visible in all of the three cases.

![Figure 233: Velocity distribution on center plane in wake region at $D/L=0.1$, $H/c=0.22$, and (a) $Re=600,000$ (b) $Re=1,200,000$ (c) $Re=1,800,000$](image)

Having a look at the immediate area around the wing in Figure 234, the flow structure shows a lot of similarities. The velocity scales are adjusted to the existing velocities around the wing. All three cases show an immediate separation on the top surface after the leading edge. The only visible difference in flow structure is in Figure 234 (a) where
the velocity is \( \text{Re} = 600,000 \), the flow shows a separation region underneath the wing whereas in the other two cases, the flow stays attached almost to the end of the wing. It shows that by increasing the velocity, the flow stays longer attached on the suction surface of the wing. At a velocity of \( \text{Re} = 600,000 \) the maximum velocity increases by 93.66 \%. While increasing the velocity of the simulation, the maximum velocity underneath the wing increases at \( \text{Re} = 1,200,000 \) by 85.16 \% and for \( \text{Re} = 1,800,000 \) by 34.22 \% compared to the set freestream velocity. The effect of not having the same amount of velocity increase has a negative influence on the lift coefficient.

Figure 234: Velocity distribution on center plane around the wing at \( H/c=0.22, D/L=0.1 \), and (a) \( \text{Re}=600,000 \) (b) \( \text{Re}=1,200,000 \) (c) \( \text{Re}=1,800,000 \)

### 7.3.3.1.2 Center Plane Streamlines

Center plane streamlines visualize the flow around the model. Figure 235 to Figure 237 show the streamlines for the three velocity cases where the wing is placed at a distance of \( D/L = 0.1 \) behind the bluff body. It can be clearly seen that the wake of the bluff body is growing in length which could already been seen in section 7.3.3.1.1. The immediate separation of the flow on the top surface of the wing resulting in an upwash can be seen in all of the three cases. The wing redirects the flow upwards and keeps the wake disturbance similar between the bluff body and the wing. For a velocity of 60 m/s or a Reynolds number of \( \text{Re} = 1,200,000 \), the streamlines show a small upwash behind the
wing. This upwash is similar to the ones seen in section 7.2.3.4 and can be explained by the nature of the wake which is an unsteady component within the simulation results.

Figure 235: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=600,000

Figure 236: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=1,200,000

Figure 237: Center plane streamlines at H/c=0.22, D/L=0.1 at Re=1,200,000
### 7.3.3.1.3 Wake Analysis

The velocity distribution on a cross-section plane at a distance $D/c = 0.66$ behind the wing shows some differences between the three cases which may be seen in Figure 238. All three cases show the mushroom shape of the wake. However, the head has some differences in velocity distribution. At a speed of $Re = 1,200,000$ the two outer vortices are merged with the two inner whereas they separate again for a speed of $Re = 1,800,000$. Another difference which can be seen are the different disturbances near the ground. Whereas at a velocity of $30$ m/s, $Re = 600,000$, the wake gets smaller in height towards the outside. An increase in height is visible at the far end width where the trailing vortices of the wheels merge with the wake of the wing. These ground disturbances grow at a velocity of $60$ m/s, $Re = 1,200,000$ compared to the slower speed. However, by increasing the velocity to $90$ m/s, $Re = 1,800,000$, the wake flattens out near the ground.

![Figure 238: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.22$, $D/L=0.1$ and (a) $Re=600,000$ (b) $Re=1,200,000$ (c) $Re=1,800,000$](image)

A better view on the behavior of the wake near the ground give the vortices plotted on top of the velocity distribution for the three cases in Figure 239 to Figure 241. The additional vortices near the ground at a velocity of $Re = 1,200,000$ can be seen clearly. These additional vortices have a negative impact on the induced drag. The larger the
wake vortices of the wing get, the more the negative influence on the lift coefficient.

Further, it can be clearly seen that higher velocity, \( Re = 1,800,000 \), has an impact on the mid-span vortices. Figure 241 shows an almost constant velocity distribution along the wing span and therefore a clear reduction of the mid-span vortices which can be seen in the lift coefficient since the downforce increase towards to high velocity case.

Figure 239: Wake vortices near ground at a distance \( D/c=0.66 \) behind the wing for ground clearance \( H/c=0.22, D/L=0.1, \) and \( Re=600,000 \)

Figure 240: Wake vortices near ground at a distance \( D/c=0.66 \) behind the wing for ground clearance \( H/c=0.22, D/L=0.1, \) and \( Re=1,200,000 \)

Figure 241: Wake vortices near ground at a distance \( D/c=0.66 \) behind the wing for ground clearance \( H/c=0.22, D/L=0.1, \) and \( Re=1,800,000 \)
7.3.3.1.4 Streamlines

The three dimensional streamlines visualize the flow around the complete model. The wing tip trailing vortices can be seen in all three cases in Figure 242 (b) to Figure 244 (b). However, the velocity has a major influence on these trailing vortices. Section 7.2.3 pointed out that the upwash caused by the wing slows down the trailing vortices. However, it can be seen that the trailing vortices are still intact at a velocity of 60 m/s or Re = 1,200,000. Further at Re = 1,800,000, the trailing vortices are barely influenced anymore from the upwash. The velocity also has an influence on the flow which gets redirected around the model. By increasing the velocity, the air pulls towards the center faster at higher velocity. This is the reason why the mid-span vortices got weaker at a velocity of Re = 1,800,000.

Figure 242: Streamlines around the model with shown pressure distribution with wing at H/c=0.22, D/L=0.1, and Re=600,000 (a) top view (b) 3D view
Figure 243: Streamlines around the model with shown pressure distribution with wing at 
H/c=0.22, D/L=0.1, and Re=1,200,000 (a) top view (b) 3D view

Figure 244: Streamlines around the model with shown pressure distribution with wing at 
H/c=0.22, D/L=0.1, and Re=1,800,000 (a) top view (b) 3D view

7.3.3.2 Flow Structure Analysis at Various Speeds at a Distance D/L = 0.3

Similar to the various speed cases in section 7.3.3.1, the velocity distribution and 
constraint streamlines are analyzed on the center plane. Further, a cross-section plane
66 % of the chord length downstream behind the wing is used to analyze the velocity
distribution of the wake and its vortices. Last, the streamlines give an overview of the
flow around the complete model.

7.3.3.2.1 Velocity Distribution

The velocity distribution for the cases where the wing is placed at a distance \( D/L = 0.3 \)
downstream can be seen in Figure 245 for all three velocity cases. Having a look on the
wake region propagating from the bluff body, it can be seen that the body wake is
growing while the velocity is increasing. At the highest velocity, 90 m/s or its
corresponding Reynolds number \( Re = 1,800,000 \), the wake reaches all the way towards
the leading edge of the wing whereas the other two cases show a clear split between
the wake and the wing. The fact that the wake is growing larger downstream is not an
unexpected phenomena. However, it changes the velocities and flow around the wing
which has an influence on the lift and drag coefficient.
Figure 245: Velocity distribution on center plane in wake region at D/L=0.3, H/c=0.25, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

Taking a closer look on the flow around the wing itself, some differences can be seen. The velocity scales around the wing are adjusted to the actual velocities acting on the wing. The stagnation point on the leading edge is moving between the three cases. At low speed, the stagnation point is located towards the upper surface whereas it moves down for middle speed range and up towards upper surface for high speeds. At the two cases where the stagnation point lies on the upper surface, a larger separation region is visible on the suction surface. Having a look on the acting velocities, at Re = 0.6 \times 10^6 the maximum velocity increases by 73.66 %. While increasing the velocity, the maximum velocity underneath the wing increases at Re = 1.2 \times 10^6 by 68.83 % and for a velocity of Re = 1.8 \times 10^6 by 70.22 % compared to the set freestream velocity. Having a lower velocity increase has a negative impact on lift coefficient but also decreases the skin-friction drag which results in a lower drag coefficient.
Figure 246: Velocity distribution on center plane around the wing at H/c=0.25, D/L=0.3, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

7.3.3.2.2 Center Plane Streamlines

For all three cases, it can be seen that the main flow characteristics behind the bluff body is similar. The wake region between the wing and the bluff body shows vortices and an upwash can be seen where the wake from the main body and the rear wing of the bluff body merge. The center plane streamlines show also that the flow is attached at low and high speed cases as it may be seen in Figure 247 and Figure 249. In Figure 248, similar to the cases where the wing is placed at a distance D/L = 0.1 downstream, the wing forces a stronger upwash which leads to the greater decrease in downforce between 30 m/s and 60 m/s compare to the difference of 60 m/s to 90 m/s.
Comparing the three different speeds where the wing is placed at a ground clearance of \( H/c = 0.25 \) and a distance of \( D/L = 0.3 \) downstream in Figure 250, it can be seen that the wake looks similar in all cases. Recall from section 7.3.3.1.3, the velocity increase caused the disturbance near the ground to flatten out and the vortices got less. However, a wing placed 30 % of a car length downstream, shows that the wake near
ground area shows the wake of the wing placed into the wake of the bluff body. That means the normal wake of a wing in ground effect is clearly visible. Outside of the wing tip vortices of the analyzed wing, the disturbance from the wheels of the bluff body can be seen clearly. The velocity change between the different cases affect the transition region between wing wake and bluff body wake. Further, around the mushroom head caused by the wing tip vortices of the rear wing on the bluff body, an even higher velocity region is visible at the case for Re = 1,200,000. This can be explained by the additional velocity component caused by the upstream. However, it clearly shows that this upstream is not strong enough to effect the wing tip vortices significant.

Figure 250: Wake velocity distribution at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.3 and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

Since the region of interest is near the ground, the wake of the actual wing, Figure 251 to Figure 253 show the tangential velocity vectors displayed on the cross-section plane. As already seen on the velocity distribution on the cross-section plane, all three cases show a lot of similarities. First, the wing tip trailing vortices of the analyzed wing are clearly visible and are the main element in all three cases. Second, mid-span vortices are present throughout the three cases. However, these mid-span vortices gain in strength by increasing the velocity which lets the wake grow and has a negative influence on the lift coefficient. Last, the vortices originated from the bluff body outside of the wing area are changing. These outside vortices are influenced by the velocity.
Higher velocity results in stronger wing tip vortices which generates a separation between the vortices from the bluff body and the vortices from the wing. Since the outside vortices are growing between 60 m/s and 90 m/s, the velocity has a direct impact on the bluff body vortices.

Figure 251: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.3, and Re=600,000

Figure 252: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.3, and Re=1,200,000

Figure 253: Wake vortices near ground at a distance D/c=0.66 behind the wing for ground clearance H/c=0.25, D/L=0.3, and Re=1,800,000
7.3.3.2.4 Streamlines

The flow around the complete model visualized by streamlines shows that the increase of velocity results in stretching the wing tip vortices of the bluff body which may be seen in Figure 254 to Figure 256. It can be clearly seen that at low velocity, the trailing vortices have fully developed. Increasing the velocity and stretching the wing tip vortices results in getting more disturbance towards the wing. At high speed, 90 m/s or its corresponding Reynolds number $Re = 1,800,000$, it can be seen that the trailing vortices are not fully developed until they hit the wing, decreasing the positive effect of the trailing vortices on the wing which leads to the decrease in lift coefficient. Further, since the vortices are still developing to its shape, the velocity is lower which has a positive impact on the drag coefficient.

Figure 254: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$, $D/L=0.3$, and $Re=600,000$ (a) top view (b) 3D view
Figure 255: Streamlines around the model with shown pressure distribution with wing at H/c=0.25, D/L=0.3, and Re=1,200,000 (a) top view (b) 3D view

Figure 256: Streamlines around the model with shown pressure distribution with wing at H/c=0.25, D/L=0.3, and Re=1,800,000 (a) top view (b) 3D view
7.3.3.3 Flow Structure Analysis at Various Speeds at a Distance D/L = 0.5

To analyze a wing placed a half of a car length downstream behind the bluff body at its most effective ground clearance, H/c = 0.25, is done similar to section 7.3.3.1 and 7.3.3.2. The velocity distribution and the constraint streamlines are analyzed on the center plane for the velocities of 30 m/s, 60 m/s, and 90 m/s. The wake is analyzed on a cross-section plane 66 % of the chord length downstream from the wings trailing edge. Last, streamlines are used to visualize the flow around the bluff body and wing.

7.3.3.3.1 Velocity Distribution

A wing placed at a distance of D/L = 0.5 behind the bluff body shows in Figure 257 that the body wake of the bluff body does not reach all the way to the wing for any studied velocity. The only difference which can be seen between the three different velocities is the size of the wake. At a velocity of 30 m/s, the body wake reached approximately a third of the way to the wing. By increasing the velocity to 60 m/s, the wake reaches approximately 50 % of the way to the wing. Last, at 90 m/s, the body wake reaches a little bit further than at 60 m/s, nonetheless, the influence of the wake carries on which can be seen as lower velocity reaching back to the leading edge of the wing.
Figure 257: Velocity distribution on center plane in wake region at D/L=0.5, H/c=0.25, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

In Figure 258, a closer look is provided on the flow around the analyzed wing. The main flow characteristics look the same in all three cases. One of the differences is that at high speed, the surrounding velocity is lower compared to the other cases, which is the influence of the low velocity which is reaching further back of the wake. At a set velocity of 30 m/s, the maximum velocity increase underneath the wing increases by 102.16 %. Increasing the simulation velocity to 60 m/s the maximum velocity increase is 133.25 %, whereas at 90 m/s, the maximum velocity increases to 206.5 m/s which corresponds to an increase of 129.44 %. The significant increase of velocity between 30 m/s and 60 m/s has a positive impact on the lift coefficient. However, the lift coefficient increase is the lowest at 90 m/s even though the maximum velocity increases more compared to the case at 30 m/s. This only explains a decrease in lift coefficient compared to 60 m/s. Another decrease comes from the trailing vortices which are not as effective as at lower speeds, which can be seen in section 7.3.3.3.3.
Figure 258: Velocity distribution on center plane around the wing at H/c=0.25, D/L=0.5, and (a) Re=600,000 (b) Re=1,200,000 (c) Re=1,800,000

7.3.3.3.2 Center Plane Streamlines

The center plane streamlines in Figure 259 to Figure 261 show the flow on the center plane around the bluff body and the wing. The velocity increase results in a stronger upwash at the end of the body wake. The vortices of the body wake do not really change between the three different velocity cases, which is not surprising since the wake is a separation region of the body. Figure 259 (b) to Figure 261 (b) show the flow around the wing itself. In all three cases an upwash component is present. However, the flow stays attached along the top surface from leading to trailing edge and underneath the common separation region occurs.

Figure 259: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=600,000 (a) Complete model (b) Around the wing
Figure 260: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=1,200,000
(a) Complete model (b) Around the wing

Figure 261: Center plane streamlines at H/c=0.25, D/L=0.5, and Re=1,800,000
(a) Complete model (b) Around the wing

7.3.3.3.3 Wake Analysis

The velocity distribution on a cross-section plane 66 % of the chord length downstream behind the wing placed at a distance of D/L = 0.5 shows some differences between the three different velocity cases. The upper part of the wake, which is referred to as the mushroom head in this study is growing in height between the low and high velocity case. At a speed of 60 m/s, or its corresponding Reynolds number Re = 1,200,000, the mushroom head is pushed upwards due to the upwash produced in the wake. At high speed, 90 m/s, the mushroom head is deformed in height, which is a phenomena of the velocity. The velocity distribution is near the ground looks similar in all three cases as shown in Figure 262.
Figure 262: Wake velocity distribution at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.25$, $D/L=0.5$ and (a) $Re=600,000$ (b) $Re=1,200,000$ (c) $Re=1,800,000$

Focusing on the wake vortices and velocity distribution near the ground, it can be seen that the wing wake stays more or less the same between the three different velocities. A small change can be seen in the low velocity region behind the trailing edge of the wing. This small region stays the same for the velocity equal 30 m/s and 60 m/s as it may be seen in Figure 263 and Figure 264. However, at high speed, 90 m/s, this low velocity region growth in width compared to the first two cases which can be seen in Figure 265. This growth of the low velocity region indicates a growth in the wake which has a negative influence on the lift coefficient. Further, through that growth of the low velocity region, the vortices get weakened which has a positive influence on the drag. The main wing tip vortices stay throughout the three cases the same and are the main part of the wake near the ground.

Figure 263: Wake vortices near ground at a distance $D/c=0.66$ behind the wing for ground clearance $H/c=0.25$, $D/L=0.5$, and $Re=600,000$
Figure 264: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25, \ D/L = 0.5, \) and \( \text{Re} = 1,200,000 \)

Figure 265: Wake vortices near ground at a distance \( D/c = 0.66 \) behind the wing for ground clearance \( H/c = 0.25, \ D/L = 0.5, \) and \( \text{Re} = 1,800,000 \)

7.3.3.3.4 Streamlines

The streamlines in Figure 266 to Figure 268 show a lot of similarities. However, the known trailing vortices originated from the rear wing of the bluff body changes with the increase of the velocity. As already seen in section 7.2.5.6, the trailing vortices progress throughout the domain at a velocity of 30 m/s or its corresponding Reynolds number of \( \text{Re} = 600,000 \) in a steady occurrence. However, by increasing the velocity the rotation of these trailing vortices stretch out and tend to rise upwards. At a velocity of 90 m/s or \( \text{Re} = 1,800,000 \), the wing tip vortices of the rear wing rise upwards and are really stretched out. This results in the loss of the positive effect on the lift coefficient which explains the decrease in downforce. Further, the high velocity results in additional disturbance.
originated by the bluff body. However, the wing is placed far enough downstream that these additional disturbances have no effect on the lift or drag coefficient.

Figure 266: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$, $D/L=0.5$, and $Re=600,000$ (a) top view (b) 3D view

Figure 267: Streamlines around the model with shown pressure distribution with wing at $H/c=0.25$, $D/L=0.5$, and $Re=1,200,000$ (a) top view (b) 3D view
Figure 268: Streamlines around the model with shown pressure distribution with wing at 
H/c=0.25, D/L=0.5, and Re=1,800,000 (a) top view (b) 3D view

7.4 Conclusion of Phase 4 - A Wing operating in a Wake

A wing operating in a wake at various distances behind the bluff body demonstrate the 
behavior of a race car front wing. The three chosen distances, which were identified in 
Phase 2 – Creation of a Bluff Body, gave a good overview of different stages of 
influence.

Placing a wing close behind the bluff body, D/L = 0.1, showed a massive loss in 
downforce. Nevertheless, the upside of it is a massive decrease in drag. The nature of 
the wing’s behavior does not change at this distance. The most effective ground 
clearance in terms of lift coefficient is still at a ground clearance of H/c = 0.22. This level 
of ground clearance was already observed as the most effective for a wing operating in
ground effect only. The flow structure analysis showed that the wing is operating in lower air velocities due to the bluff body in front. Further, it could be seen that the wing creates an upwash within the wake which destroys the trailing vortices originated from the wing tips of the rear wing of the bluff body. The flow around the analyzed wing is mostly separated. Due to the mentioned upwash, the flow separates at the leading edge on the top surface but stays mostly attached underneath the wing.

Moving the wing farther away, to a distance $D/L = 0.3$ changed the flow and aerodynamic forces significantly. The lift and drag coefficient started to recover towards their initial values found in section 4.2. However, the wing still obtains a loss in downforce for the most effective ground clearance though the most effective ground clearance moved to $H/c = 0.25$. The flow structure analysis showed that a lot of disturbance of the wake occurs near the ground. At a ground clearance of $H/c = 0.25$, the wing is more effective in the wake than in undisturbed flow. This downforce increase is produced by the trailing vortices originated from the rear wing of the bluff body. The circulation of the flow produces an additional downforce on the wing. Further, the level of drag is lower since still lower velocities exists around the wing compared to undisturbed flow. The flow structure analysis shows that the wing is operating outside of the main body wake when the wing is placed at a distance of $D/L = 0.3$ behind the bluff body. An upwash which affects the wing tip vortices from the bluff body still exists, but it is significantly weaker than in the case where the wing is closer to the body.

When the wing is placed a half a car length behind the bluff body, $D/L = 0.5$, it is now operating clearly outside of the body wake. However, the creation of the bluff body already showed that the wing tip vortices from the rear wing of the bluff body will be
effective at this distance. Further, the disturbances near the ground are still present. Therefore, the most effective ground clearance occurred at $H/c = 0.25$, the same as at $D/L = 0.3$. The lift coefficient does exceed the level of undisturbed flow. This led to the finding that the wing tip vortices originated from the rear wing of the bluff body has a positive effect on the wing. Moreover, since the wing is still operating in the wake, the velocities have not recovered to freestream by the time the flow hits the wing. Therefore, the wing has a smaller drag coefficient than in undisturbed flow. The flow structure analysis showed that the main character of the flow is similar to the undisturbed case.

The effect of velocity on a wing operating in a wake shows that by increasing the velocity, the lift coefficient decreases in most of the cases as well as the drag in all of them. A positive impact on the lift coefficient can be seen at medium velocity, set to be 30 m/s. The flow structure analysis showed that by increasing the velocity, the size of the wake increases too. This size increase leads to the observation that the relation of the wake length to the distance between bluff body and wing is similar for the cases where the wing is placed at a distance of $D/L = 0.3$ downstream at a velocity of 30 m/s and at a distance of $D/L = 0.5$ with a velocity of 90 m/s. The lower relative velocities acting around the wing result not only in a decrease of drag coefficient but also in lift coefficient.
8 Discussion

The goal of this thesis was to analyze a race car front wing operating in a wake. The very specific rules and regulations in racing sport makes the design of a race car a complicated matter. An extended literature review showed the known influence of angle of attack, ground clearance on single and multi-element wings. Further, the effect of endplates on front and rear wings as well as gurney wing flaps are covered.

The work in this thesis was divided into multiple phases. Phase 1 analyzed the high-lift wing profile S1223 in freestream condition to verify to geometry of the model. Small deviation compared to the published literature data by Selig et al. [5] on the chosen wing profile were found during the study on the effect of angle of attack on a wing in freestream condition. The curve of lift coefficient vs. angle of attack is shifted down and the effective range of the wing is up to 14 degree angle of attack, whereas the literature results showed an effective range up to 16 degrees. However, in the range from 0 to 15 degrees angle of attack, the lift coefficient is only shifted down. This difference is a combination of different explanations such as computation vs experimental, measuring uncertainties in experimental results, influence of the wind tunnel (even though it should be accounted for), and theoretical profile vs actual manufactured profile. Since the behavior of the wing is similar for the range of 0 to 14 degrees, the geometry of the wing profile was used from that point for all future studies in this thesis.

The final goal of this study was to analyze the wing operating in a wake, a bluff body had to be created in phase 2 to simulate an ahead driving car. Wilson et al [7] showed that a simple bluff body can create the main elements of a wing. Therefore, a bluff body
containing a body, wheels, and rear wing was created and its wake analyzed for its size and main characteristics. The analysis showed that an appropriate range of distance would be placing the wing between $D/L = 0.1$ and $D/L = 0.5$ downstream. The distance $D/L = 0.5$ was also a computational limitation. Any larger distance could not be simulated because of the mesh sizes exceeded the computational resources.

Phase 3, a wing operating in ground effect served as a real benchmark solution for the wing operating in a wake in phase 4. By decreasing the ground clearance, the increase in lift coefficient could be seen up to a maximum point, which has been shown by various different authors [9] [17] [20] before. The continuous drag increase showed the effect of increasing skin-friction drag and induced drag through the wake. The analysis of the wake showed that in ground effect, multiple span-wise vortices start to build and get stronger with decreasing ground clearance. The most effective ground clearance was identified at 22 % of the chord length, measured from the ground to the leading edge. The downforce increased by 46.4 % compared to freestream at a ground clearance of $H/c = 0.22$, whereas the drag coefficient increased at the same ground clearance by 63.2 %. Unlike the lift coefficient, the drag coefficient remains increasing even after the most efficient ground clearance.

Phase 4, a wing operating in a wake showed some major differences compared to the case where it was just operating in ground effect. The analysis showed that depending on where the wing is placed in the disturbed air, it has a major influence on the aerodynamic forces. For example, a massive loss of downforce and drag could be identified at a short distance, $D/L = 0.1$, behind the bluff body. Compared to phase 3, where the wing is operating in undisturbed air and ground clearance, the lift coefficient
decreases by 69.01%. However, since the wing is operating in lower relative velocities, the drag coefficient decreases by 87.85 % at the most effective ground clearance of $H/c = 0.22$ compared to the result in phase 3 – A wing operating in ground effect only. At mid distance, the most efficient ground clearance moves from 22 % of the chord length to 25 %. For a velocity of 30 m/s, the lift coefficient increases slightly in magnitude by 0.11 % at a ground clearance of $H/c = 0.25$, when the wing is operating in a wake instead of only in ground effect. However, the maximum lift coefficient is still higher in undisturbed flow. At this distance downstream, the drag coefficient decreases by 16.77 % compared to undisturbed flow due to the lower relative velocities existing around the wing. By moving the wing further downstream to a distance equal to half of a car length, $D/L = 0.5$, the most effective ground clearance stays at $H/c = 0.25$. In this case, the lift coefficient increases even more compared to undisturbed flow at the most efficient ground clearance. The increase in downforce compared to undisturbed flow is 3.88 % at a ground clearance of $H/c = 0.25$. Further, the maximum downforce at a distance of $D/L = 0.5$ at a ground clearance of $H/c = 0.25$ exceeds the most effective ground clearance of undisturbed flow, $H/c = 0.22$ by 0.52 %. The reduction of the drag coefficient in the wake can be quantified by 7.78 % compared to undisturbed flow at the most effective ground clearance within the wake. The study of the effect of velocity on the aerodynamic forces in phase 4 showed that the velocity has a significant impact. The characteristics of the flow stay similar. However, by increasing the velocity, the wake of the bluff body grows and puts therefore the wing back into the wake for some cases which has a negative influence on the lift coefficient.
9 Conclusion

As a result of the conducted CFD studies, several conclusions can be made. This thesis analyzes a single element wing in different states of operation and provides an understanding of the aerodynamic forces. The different parameter studied within the four phases, show that the velocity, ground clearance, angle of attack, and distance between a car and the wing influence the aerodynamic forces. Shown within the literature, the velocity is identified as the least effective parameter for a wing operating in freestream or ground effect only, more compliance comes to the velocity when a race car front wing is operating in a wake. This is because the size of the wake is not only controlled by the size of the bluff body, it is also controlled by the velocity. A change in the wake, results in change of the relative velocities surrounding the wing. The studies also showed that the effect of ground clearance is similar within a wake and in undisturbed flow, but the downforce reaches its maximum at a larger ground clearance due to the additional disturbance near the ground for larger distances between the bluff body and the wing. The analysis of a wing operating in a wake showed that a simple bluff body is sufficient to investigate the behavior of the aerodynamic forces. However, it is believed that at small distances, 10 percent of a car length, between the bluff body and the wing, the geometry of the bluff body becomes a more significant factor since the body wake characteristics is likely to be changed by modifying the body geometry. Nevertheless, for larger distances, a half of a car length, the main element of the wake are the trailing vortices of the rear wing of the bluff body.

Further, the study in phase 4 – *A Wing Operating in a Wake* provides explanation for racing behavior too. A significant decrease in drag is observed for close distances, 10 %
of a car length, behind the bluff body. This supports the fact that racing down a straight track, the driver following another car should reduce this distance as much as possible to benefit from the drag reduction. However, since there is also a significant reduction in downforce, a larger distance between the two cars should be chosen since the downforce increases the ability of a faster cornering speed. At the optimal distance, which is changing depending on the racing speed, an advantage can be taken by creating more downforce in a wake compared to undisturbed flow. This leads to the fact that at the end of a turn, higher car speed in the chasing car can be achieved. Any effect on the leading cars’ performance has not been analyzed in this work. However, by changing the nature of the wake of a car, its performance is influenced by changing the momentum within the flow.
10 Future Work

There are multiple different possibilities to continue this research. As shown in the literature review, only few studies of a race car wing operating in a wake have been carried out. Therefore, the wing could now be analyzed with different add-ons such as endplates or gurney wing flaps to determine their influence in disturbed air. Moreover, the study carried out in this thesis could also been carried out for different airfoils, multi-element wings and additional flaps. Further, it would also be interesting to see how the aerodynamic forces behave when the car is driving through a turn; respectively, what is the effect if the wake hits the wing on a yaw angle. Since it is also believed that the geometry of the bluff body has a greater impact on a wing when it is placed a short distance downstream, the wing could be analyzed behind an actual race car model.
## 11 Appendix

### S1223 Profile [10]

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0.08545</td>
<td>0.08879</td>
</tr>
<tr>
<td>0.99838</td>
<td>0.00126</td>
<td>0.06789</td>
<td>0.0794</td>
</tr>
<tr>
<td>0.99417</td>
<td>0.00494</td>
<td>0.05223</td>
<td>0.06965</td>
</tr>
<tr>
<td>0.98825</td>
<td>0.01037</td>
<td>0.03855</td>
<td>0.05968</td>
</tr>
<tr>
<td>0.98075</td>
<td>0.01646</td>
<td>0.02694</td>
<td>0.04966</td>
</tr>
<tr>
<td>0.97111</td>
<td>0.0225</td>
<td>0.01755</td>
<td>0.03961</td>
</tr>
<tr>
<td>0.95884</td>
<td>0.02853</td>
<td>0.01028</td>
<td>0.02954</td>
</tr>
<tr>
<td>0.94389</td>
<td>0.03476</td>
<td>0.00495</td>
<td>0.01969</td>
</tr>
<tr>
<td>0.92639</td>
<td>0.04116</td>
<td>0.00155</td>
<td>0.01033</td>
</tr>
<tr>
<td>0.90641</td>
<td>0.04768</td>
<td>0.00005</td>
<td>0.00178</td>
</tr>
<tr>
<td>0.88406</td>
<td>0.05427</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.85947</td>
<td>0.06089</td>
<td>0.17006</td>
<td>-0.00075</td>
</tr>
<tr>
<td>0.83277</td>
<td>0.06749</td>
<td>0.20278</td>
<td>0.00535</td>
</tr>
<tr>
<td>0.80412</td>
<td>0.07402</td>
<td>0.2384</td>
<td>0.01213</td>
</tr>
<tr>
<td>0.77369</td>
<td>0.08044</td>
<td>0.27673</td>
<td>0.01928</td>
</tr>
<tr>
<td>0.74166</td>
<td>0.08671</td>
<td>0.3175</td>
<td>0.02652</td>
</tr>
<tr>
<td>0.70823</td>
<td>0.09277</td>
<td>0.36044</td>
<td>0.03358</td>
</tr>
<tr>
<td>0.6736</td>
<td>0.09859</td>
<td>0.40519</td>
<td>0.04021</td>
</tr>
<tr>
<td>0.63798</td>
<td>0.10412</td>
<td>0.45139</td>
<td>0.04618</td>
</tr>
<tr>
<td>0.60158</td>
<td>0.10935</td>
<td>0.4986</td>
<td>0.05129</td>
</tr>
<tr>
<td>0.56465</td>
<td>0.11425</td>
<td>0.54639</td>
<td>0.05534</td>
</tr>
<tr>
<td>0.52744</td>
<td>0.11881</td>
<td>0.59428</td>
<td>0.0582</td>
</tr>
<tr>
<td>0.49025</td>
<td>0.12303</td>
<td>0.64176</td>
<td>0.05976</td>
</tr>
<tr>
<td>0.4534</td>
<td>0.12683</td>
<td>0.68832</td>
<td>0.05994</td>
</tr>
<tr>
<td>0.41721</td>
<td>0.13011</td>
<td>0.73344</td>
<td>0.05872</td>
</tr>
<tr>
<td>0.38193</td>
<td>0.13271</td>
<td>0.7766</td>
<td>0.05612</td>
</tr>
<tr>
<td>0.34777</td>
<td>0.13447</td>
<td>0.81729</td>
<td>0.05219</td>
</tr>
<tr>
<td>0.31488</td>
<td>0.13526</td>
<td>0.855</td>
<td>0.04706</td>
</tr>
<tr>
<td>0.28347</td>
<td>0.13505</td>
<td>0.88928</td>
<td>0.04088</td>
</tr>
<tr>
<td>0.2537</td>
<td>0.13346</td>
<td>0.91966</td>
<td>0.03387</td>
</tr>
<tr>
<td>0.22541</td>
<td>0.13037</td>
<td>0.94573</td>
<td>0.02624</td>
</tr>
<tr>
<td>0.19846</td>
<td>0.12594</td>
<td>0.96693</td>
<td>0.01822</td>
</tr>
<tr>
<td>0.17286</td>
<td>0.12026</td>
<td>0.98255</td>
<td>0.0106</td>
</tr>
<tr>
<td>0.14863</td>
<td>0.11355</td>
<td>0.99268</td>
<td>0.00468</td>
</tr>
<tr>
<td>0.12591</td>
<td>0.10598</td>
<td>0.99825</td>
<td>0.00115</td>
</tr>
<tr>
<td>0.10482</td>
<td>0.0977</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>Ref #</td>
<td>Year</td>
<td>General Info</td>
<td>Experimental</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>--------------</td>
<td>---------------</td>
</tr>
<tr>
<td>15</td>
<td>2005</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>2008</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>18</td>
<td>2008</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>16</td>
<td>2002</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>27</td>
<td>2005</td>
<td>x</td>
<td>(a)</td>
</tr>
<tr>
<td>25</td>
<td>1998</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>4</td>
<td>2010</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>13</td>
<td>1997</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>21</td>
<td>2006</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>2005</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>1</td>
<td>2006</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>2000</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>14</td>
<td>2003</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>11</td>
<td>1994</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>9</td>
<td>2009</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>15</td>
<td>2011</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>2008</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>19</td>
<td>2011</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>21</td>
<td>2006</td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>

Notes:
- **General Info**: Experimental, CFD, WISH, Flow, Ground, Wing characteristics, Wing profile, AOA, Speed (m/s), Ground clearance
- **Reference Numbers**: Ref #, Year
- **Wish List**: x indicates presence, blank indicates absence
- **General Description**: Paper, Review Paper
12 References


