Aerodynamic Interaction
Mechanisms Relevant to the
Underbodies of Formula 1 Cars

by

Barnaby John Garrood

Department of Aeronautics
Imperial College London
Prince Consort Road
London SW7 2BY

Thesis submitted for the degree of
Doctor of Philosophy of the
University of London

January 2004
Abstract

This thesis presents an investigation of the flow physics governing aerodynamic interaction mechanisms relevant to Formula One racing cars, in particular the influence of the front wing on the car chassis flat underbody and diffuser.

The research has taken a fundamental approach, with emphasis on identifying the fundamental mechanisms governing the interaction. Wake diagnostic techniques have been used to identify the major flow features of the front wing wake that are likely to influence the flat underbody and diffuser. Models have then been designed to simulate these various flow features.

A simple model was produced with an underbody which is flat apart from a diffuser over the rear 20% of its length. This was used to simulate the basic flow features of the flat underbody and diffuser of a Formula One car. Wind tunnel measurements of force and surface pressure have been made on this diffuser model with the various upstream models simulating aspects of the front wing wake.

It has been found that the downforce generated by the diffuser model is highly sensitive to the configuration of the upstream front wing. Although, for the majority of front wing configurations the downforce generated by the diffuser model was reduced, it was discovered that for certain arrangements the downforce may actually be enhanced.

By piecing together the various results gathered over the course of the research, the underlying flow physics involved in the interaction have been identified. Using this information, suggestions are made for how the configuration of the front wing may be optimised for minimum negative impact on the performance of the car underbody and diffuser.
Acknowledgements

Over the past 3 years a large number of people have provided me with academic, technical and emotional support, without which I would not have been able to complete this work.

First and foremost, I would like to express my deepest gratitude to Professor John Harvey, and Professor Peter Bearman, who have given me the opportunity to work on this project. It has been a true pleasure to work under their supervision and guidance.

Also to Luciano Mariella of Ferrari Gestione Sportiva who has provided me with substantial support and advice, which I gratefully acknowledge.

Financial support for the project has been provided jointly by the Department of Aeronautics at Imperial College London, and Ferrari Gestione Sportiva, for which I am sincerely grateful.

Thanks in particular to Nigel MacCarthy for his extensive advice and support, to Ian James, Alan Smith, Joseph Meggyesi and Roland Hutchings for their skilful building of wind tunnel models and support in the wind tunnel. I am also indebted to Len Parrish, Clive Mott and Russell Carter for their help with electrical systems, to Steve Johnson, Dave Bowler and John Churchwood for their frequent and willing help in the workshop, and to John O’Leary for his invaluable help with photography and flow visualisation.

My sincere thanks also go to John Owen, Maša Branković, Christine Hogan, Fran Huera Huarte and Jon Pegrum for putting up with me in the office over the last three years and to Sean Creighton and Richard Wilden for their extensive advice and support.
Last but by no means least, I would like to express particular gratitude to Nicola Salter, whose patience and encouragement has made the completion of my work infinitely easier, and to my family for their support and understanding over the long course of my education.
In memory of my father.
Contents

Table of Contents v

List of Figures ix

List of Tables xxi

Nomenclature xxii

1 Introduction 1
  1.1 Problem Definition ............................................. 1
  1.2 Aims and Objectives .............................................. 3
    1.2.1 Aims ........................................................ 3
    1.2.2 Objectives .................................................. 4

2 Review of Literature and Relevant Theory 6
  2.1 Introduction .................................................... 6
  2.2 Ground Effect Aerodynamics .................................... 7
    2.2.1 Streamlined bodies in ground effect ......................... 8
    2.2.2 Ground plane simulation and Reynolds Number effects .... 13
    2.2.3 Effect of Endplates ......................................... 15
    2.2.4 Factors Pertinent to Negative Lifting Wings with Endplates in Ground Effect ........................................... 18
    2.2.5 Bluff Bodies in Ground Effect ................................ 20
    2.2.6 Automotive Diffusers ......................................... 22
  2.3 Vortical Flows .................................................. 29
2.3.1 The Structure of an Isolated Vortex ........................................... 29
2.3.2 Vortex Breakdown ................................................................. 33
   2.3.2.1 Description of Vortex Breakdown ...................................... 33
   2.3.2.2 Modes of Vortex Breakdown ............................................ 33
   2.3.2.3 Criteria for Vortex Breakdown ....................................... 35
2.4 Interaction Effects ................................................................. 37
   2.4.1 Basic Interaction Effects .................................................... 37
   2.4.2 Vortex Interactions ........................................................... 39
      2.4.2.1 Behaviour of multiple vortex systems .............................. 39
      2.4.2.2 Interaction between a vortex and a plane boundary .......... 40
      2.4.2.3 Interaction between a vortex and a wing ....................... 44
   2.4.3 Interaction effects relevant to Formula One cars .................. 47
3 Experimental Arrangements and Procedures ................................. 49
   3.1 Wind tunnel facility ............................................................ 49
      3.1.1 Donald Campbell Wind Tunnel ......................................... 49
      3.1.2 Moving Floor ............................................................... 50
      3.1.3 3-Axis Wind Tunnel Traverse .......................................... 50
   3.2 Wind Tunnel Models ............................................................ 51
      3.2.1 Tandem wing pair .......................................................... 51
      3.2.2 Large Wing Model ......................................................... 52
      3.2.3 Diffuser Equipped Bluff Body ......................................... 52
      3.2.4 Front Wing Model ........................................................ 53
      3.2.5 Vertical Wing Models .................................................... 54
   3.3 Model Measurements ............................................................ 55
      3.3.1 Surface Pressure Measurements ....................................... 55
         3.3.1.1 Scanivalve system .................................................... 55
         3.3.1.2 ZOC pressure scanning system ................................... 56
      3.3.2 Body Force Measurements ............................................... 57
         3.3.2.1 Overhead balance ..................................................... 57
         3.3.2.2 Internal balance ...................................................... 58
3.4 Wake Measurements ........................................... 59
  3.4.1 Smoke Flow Visualisation ..................................... 59
    3.4.1.1 Smoke and laser sheet visualisation ..................... 59
    3.4.1.2 Smoke plume .............................................. 60
  3.4.2 Total Pressure Surveys .................................... 60
  3.4.3 Particle Image Velocimetry .................................. 61
    3.4.3.1 LaVision Digital PIV System ............................. 62
    3.4.3.2 TSI Digital PIV System ................................... 63
    3.4.3.3 Experimental Setup and Procedure ....................... 63
    3.4.3.4 Perspective Error ....................................... 64
    3.4.3.5 Data Analysis .................................. 65
  3.4.4 Hot-wire Anemometry ...................................... 66
3.5 Axis conventions .............................................. 68
3.6 Wing Tunnel Corrections ...................................... 68
  3.6.1 Solid Boundary Corrections ................................ 68
    3.6.1.1 Blockage correction ................................... 69
    3.6.1.2 Downwash correction ................................... 69
  3.6.2 Wind Tunnel Calibration ................................... 70
3.7 Numerical Simulations ......................................... 71

4 Aerodynamics of Negative Lifting Wings in Ground Effect .... 75
  4.1 Introduction .................................................. 75
  4.2 Effect of Ground Proximity on the Pressure Distribution of a Low
    Aspect Ratio Wing ............................................. 77
      4.2.1 Two-Dimensional Considerations .......................... 79
      4.2.2 Effects of Three-Dimensionality .......................... 81
  4.3 Effect of Endplates on Wing Performance ..................... 84

5 Aerodynamics of a Diffuser-Equipped Bluff Body in Ground Effect ... 88
  5.1 Introduction ................................................. 88
  5.2 Lifting Behaviour ............................................ 89
  5.3 Reynolds number effects ..................................... 91
5.4 Description of Flow-Field ........................................... 96
5.5 Influences of Flow Three-Dimensionality .......................... 98
  5.5.1 Results of non-constant underbody flow rate ................. 99
  5.5.2 Influence of the Nose Vortex .................................. 101
  5.5.3 Influence of the Diffuser Vortices ............................ 103
5.6 Key Findings ......................................................... 104

6 Front Wing Wake Structure and Behaviour ........................ 106
  6.1 Introduction ....................................................... 106
  6.2 Wake Structure ................................................... 107
  6.3 Low Energy Wake .................................................. 110
  6.4 Trailing Vortex Behaviour in Ground Effect .................... 113
    6.4.1 Preliminary Results for a Vortex in Ground Effect ....... 113
    6.4.2 Front Wing Vortex Experiments ............................. 114
    6.4.3 Vertical Wing Vortex Experiments ........................... 116
  6.5 Key Findings ....................................................... 128

7 Interaction with a Trailing Vortex System ........................ 130
  7.1 Introduction ....................................................... 130
  7.2 Interaction Between a Trailing Vortex and a Wing ............. 132
  7.3 Interaction Between a Trailing Vortex and a Diffuser-Equipped Bluff Body ......................................................... 135
    7.3.1 Influence of Vertical Location of Vortices ............... 136
    7.3.2 Influence of Lateral Separation of Vortices .............. 142
      7.3.2.1 Single Vortex Pair ..................................... 142
    7.3.3 Interaction with Multiple Vortex Pairs .................... 144
      7.3.3.1 Same signed vortices .................................. 145
      7.3.3.2 Opposite signed vortices ............................... 147
  7.4 Key Findings ....................................................... 148

8 Interaction with an Upstream Horizontal Wing .................... 150
  8.1 Introduction ....................................................... 150
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.2 Influence of Low Energy Turbulent Wake</td>
<td>152</td>
</tr>
<tr>
<td>8.3 Influence of Full-Span Horizontal Wing</td>
<td>156</td>
</tr>
<tr>
<td>8.4 Influence of Partial-Span Horizontal Wing</td>
<td>160</td>
</tr>
<tr>
<td>8.5 Key Findings</td>
<td>164</td>
</tr>
<tr>
<td>9 Closing Remarks, Conclusions and Suggestions for Further Work</td>
<td>166</td>
</tr>
<tr>
<td>9.1 Closing Remarks</td>
<td>166</td>
</tr>
<tr>
<td>9.1.1 Applicability of Results</td>
<td>166</td>
</tr>
<tr>
<td>9.1.2 Relevance to Formula One</td>
<td>167</td>
</tr>
<tr>
<td>9.2 Conclusions</td>
<td>167</td>
</tr>
<tr>
<td>9.3 Suggestions for Further Work</td>
<td>171</td>
</tr>
<tr>
<td>Appendices</td>
<td>173</td>
</tr>
<tr>
<td>A Model Drawings</td>
<td>174</td>
</tr>
<tr>
<td>B Aerofoil Co-ordinates</td>
<td>181</td>
</tr>
<tr>
<td>B.1 Tandem wing model</td>
<td>182</td>
</tr>
<tr>
<td>B.2 NASA Langley GA(W)-2 LS(1)-0416</td>
<td>183</td>
</tr>
<tr>
<td>C Internal Wind Tunnel Balance</td>
<td>185</td>
</tr>
<tr>
<td>C.1 design</td>
<td>185</td>
</tr>
<tr>
<td>C.2 Calibration</td>
<td>188</td>
</tr>
<tr>
<td>D One-Dimensional Diffuser Theory with Variable Volume-Flow Rate</td>
<td>190</td>
</tr>
<tr>
<td>Bibliography</td>
<td>192</td>
</tr>
</tbody>
</table>
List of Figures

1.1 Typical Formula One car (Piola, 2000) ........................ 2
2.1 Schematic of image horseshoe vortex system used to simulate ground plane ................................................. 8
2.2 Effect of ground proximity on the effective aspect ratio of a positive lifting wing in ground effect. (Fink and Lastinger, 1961) .............. 10
2.3 Schematic diagram showing likely effect of wing thickness on local flow in ground effect due to image sources .......................... 12
2.4 Illustration showing effect of ground-plane simulation on velocity profiles beneath a body in ground effect ............................. 13
2.5 Lift coefficient of NACA 0010 wing versus ground clearance and angle of incidence .......................................................... 14
2.6 Two-dimensional viscous CFD streamlines for flow around a generic car shape for ground stationary and moving (Sardou, 1986) ............ 15
2.7 Typical results for the effects of endplates on the drag polar for a wing of aspect ratio 6 (Hemke, 1927) ................................. 16
2.8 Schematic diagram of flow field in vicinity of endplates .............................. 17
2.9 Effect of ground clearance on lift curve slopes of low aspect ratio symmetrical wing in ground effect (Stollery and Burns, 1969) ......... 18
2.10 Lift curve slopes for a Formula 1 style wing in ground effect (Zerihan and Zhang, 2000) ................................. 19
2.11 Schematic diagram of the vortex system in the wake of a single element wing with endplates in ground effect ........................ 20
2.12 Lift coefficient versus ground clearance for Fackrell's model at $\alpha = -1, 0, \text{ and } 1^\circ$ (Fackrell, 1975). ................................. 21
2.13 Effect of nose shape on lift-curve slope for symmetrical low aspect ratio wing at h/c=0.3 (Stollery and Burns, 1969). ......................... 22
2.14 Take-off and flip of Mercedes CLR at Le Mans in 1999 due to unstable aerodynamics. ....................................................... 22
2.15 Two-dimensional diffuser geometry. ........................................ 23
2.16 Relationship between area ratio and viscous blockage at inlet, and effective area at outlet (Sovran and Klomp, 1967). ......................... 25
2.17 Diffuser flow regimes (Reneau, Johnston and Kline, 1967). .......... 26
2.18 Typical diffuser performance map for fixed boundary layer thickness at inlet (Sovran and Klomp, 1967). ..................................... 26
2.19 Automotive diffuser geometry definition. .................................. 27
2.20 Components of downforce as identified by Cooper, Bertenyi, Dutil, Syms and Sovran (1998). ..................................................... 27
2.21 Contours of diffuser model downforce from Cooper et al. (1998). ... 28
2.22 Vortex structure forming either side of the sloping base of a bluff body (Morel, 1978). ............................................................... 28
2.23 Comparison of the circulation about, and tangential velocity in a Rankine and a Lamb-Oseen vortex. ........................................... 31
2.24 Static and total pressure distributions through a delta wing vortex core at various locations downstream of the apex Earnshaw (1961). .... 31
2.25 Smoke flow visualisation of vortex breakdown over an F-18 aircraft wing (NASA Dryden Flight Research Centre)............................ 33
2.26 The three types of vortex breakdown of Sarpkaya (1971a): Double-Helix (top); Spiral (middle) and Bubble (bottom). ......................... 34
2.27 Sketches of variation in breakdown pattern with increasing amounts of swirl (Hall, 1972). ......................................................... 35
2.28 Interference effects between symmetrical streamlined and bluff bodies in tandem (Hoerner, 1958). ............................................. 38
2.29 Key stages in vortex merging process for two equal strength co-
rotating (anti-clockwise) vortices (Cerretelli and Williamson, 2003). . 40
2.30 Separation of the cross-flow boundary layer on the ground plane be-
neath a vortex to produce a secondary vortex of opposite sign (Harvey
and Perry, 1971). ............................................................... 41
2.31 Computational iso-contours of vorticity showing generation of multi-
ple secondary vortices (Puel and de Saint Victor, 2000). ............ 42
2.32 Iso-contours of vorticity showing effect of Reynolds number on pro-
duction of secondary vorticity (Türk, Coors and Jacob, 1999). .... 43
2.33 Surface flow pattern underneath vortex path on wing surface (top)
and spanwise pressure distributions at a number of chordwise loca-
tions (Patel and Hancock, 1974). ........................................ 46
2.34 Suggested cross-flow topologies for the interaction of a vortex with
a stationary surface for reducing ground clearances (Mehta and Lim,
1984). ............................................................... 46
2.35 Influence of front and rear wings on centreline pressures in the diffuser
of the 1987 March Indy car (Katz and Dykstra, 1989). ............... 47
2.36 Effect of front wing flap angle on downforce generated by the front
wing ($C_{MW}$) and the whole of a generic Indy car (Katz and Garcia,
2002). ............................................................... 48

3.1 Schematic diagram of identical tandem wing model arrangement. . . 52
3.2 Large wing model with endplates (left) and rounded tips (right). . . 52
3.3 Various views of diffuser model, with key features highlighted . . . 53
3.4 Diagram of front wing model, showing all sections in place to produce
an aspect ratio of 5. ............................................................... 54
3.5 Photograph of front wing and large wing with endplates mounted in
the wind tunnel. ............................................................... 54
3.6 All four vertical wings assembled with their mounting system, allow-
ing adjustment of incidence and horizontal separation. ............... 55
LIST OF FIGURES

3.7 Vertical wings and diffuser model mounted in the Donald Campbell wind tunnel. .......................................................... 55
3.8 Schematic diagram of internal balance load cell layout. .......... 58
3.9 Location of internal balance in diffuser model. ...................... 59
3.10 Schematic diagram of smoke generation system. ................... 60
3.11 Schematic diagram of Kiel type yaw insensitive total head probe. 61
3.12 Schematic diagram of PIV experimental setup in wind tunnel. .... 63
3.13 Arrangement of laser to produce laser sheet in required position and orientation. .......................................................... 64
3.14 Schematic diagram showing origins of perspective error. .......... 65
3.15 Measured perspective error for TSI DPIV setup. .................... 65
3.16 Coordinate system to be used throughout the thesis. .............. 68
3.17 Schematic diagram of trailing vortices from a negative lifting body in ground effect, and the image vortices in the tunnel walls. .......... 70
3.18 Wind tunnel calibration factors versus tunnel reference pressure. 71
3.19 3D grid on symmetry plane around diffuser model. ............... 74
3.20 2D computational grid in vicinity of wing. ......................... 74
4.1 Centreline pressure distribution for aspect ratio 1 wing without end-plates at a range of ground clearances. .......................... 77
4.2 Centreline pressure distributions of Zerihan and Zhang (2000), with nose-lift reversal highlighted. ................................. 78
4.3 Comparison between NASA LS(1)-0413 and 0416 sections. ........ 78
4.4 Comparison between 2-dimensional CFD and experimental centreline chordwise pressure distributions. .......................... 79
4.5 Contours of streamfunction for two-dimensional calculations of a wing in ground effect for values either side of that bisected by the aerofoil. 80
4.6 Inviscid (Chordwise pressure distributions for inviscid and viscous simulations of two-dimensional flow around an aerofoil in ground effect. 80
4.7 Wool tufts on ground at the front and rear of a single aspect ratio 1 wing in ground effect. .................................................. 82
4.8 Spanwise variation in chordwise pressure distributions without endplates for $h/c = 0.05$. 83
4.9 Spanwise variation in chordwise pressure distributions without endplates for $h/c = 0.10$. 83
4.10 Total head contours 10mm downstream of wing at ground clearance of $h/c = 0.05$ and $0.10$. 84
4.11 Spanwise variation in chordwise pressure distributions with endplates. 85
4.12 Total head contours in planes beneath and behind aspect ratio 1 wing. 85
4.13 Chordwise pressure distributions with and without endplates at $h/c = 0.10$ at $y/b = 0.0$ and 0.47. 86

5.1 Contours of downforce on axes of ground clearance, $h/c$, and diffuser angle, $\theta$. 90
5.2 Diffuser model centreline pressure distributions for a number of ground clearances at a fixed diffuser angle, $\theta = 14^\circ$. 90
5.3 Diffuser model centreline pressure distributions for a number of diffuser ramp angles at a fixed ground clearance, $h/c = 0.060$. 91
5.4 Centreline vertical underbody velocity profiles in the diffuser from CFD results. $\theta = 12^\circ$, $h/c = 0.060$. 92
5.5 Displacement thickness along diffuser model underbody from CFD data for $\theta = 12^\circ$ and $h/c = 0.060$. 92
5.6 Plot of downforce coefficient against Reynolds number with and without transition. $\theta = 14^\circ$, $h/c = 0.060$. 94
5.7 Plot of downforce against ground clearance at a number of diffuser angles with and without transition. $Re = 6.8 \times 10^5$ based on model chord (20m/s). 94
5.8 Centreline underbody surface pressure distributions with and without transition. $\theta = 12^\circ$. 95
5.9 CFD flow streamlines on model centreline showing lowered stagnation point, $\theta = 12^\circ$, $h/c = 0.06$. 96
5.10 CFD flow streamlines beneath diffuser model, $\theta = 12^\circ$, $h/c = 0.060$. 97
LIST OF FIGURES

5.11 3D CFD simulation streamtraces showing vortex shed from outer edge of diffuser model nose. $h/c = 0.060, \theta = 14^\circ$ ................................. 97
5.12 CFD flow streamlines beneath diffuser model, $\theta = 7^\circ$. ................................. 98
5.13 Volume flow rate per unit span calculated at centreline from CFD results. $\theta = 12^\circ, h/c = 0.060$. ................................. 100
5.14 Assumed 'lateral volume-flow rate' distribution, $\frac{\partial Q}{\partial x}$. ................................. 100
5.15 Comparison of theoretical and experimental pressure distributions. $\theta = 10^\circ, h/c = 0.02$ and 0.04 ......................................................... 100
5.16 Comparison of total head in wake of diffuser model with two different noses at two different ground clearances. $\theta = 7^\circ, h/c = 0.014$ and 0.080. ......................................................... 101
5.17 Variation in downforce coefficient with ground clearance for the diffuser model two different noses. $\theta = 12^\circ$. ......................................................... 102
5.18 Centreline pressure distributions for the diffuser model with two different noses at $h/c = 0.040$ and 0.080. $\theta = 12^\circ$. ......................................................... 102
5.19 Variation in pitching moment coefficient with ground clearance for the diffuser model two different noses. $h/c = 0.040$ and 0.080. $\theta = 12^\circ$. 102
5.20 Centreline underbody surface pressure distributions with and without fixed transition. $\theta = 12^\circ, h/c = 0.060$. ......................................................... 103
5.21 Spanwise pressure distributions at $x/c = 0.52$ and 0.72 at $h/c = 0.04$, 0.06 and 0.08. $\theta = 12^\circ$. ......................................................... 104
5.22 Centreline chordwise pressure distributions for $h/c = 0.04, 0.06$ and 0.08. $\theta = 12^\circ$. ......................................................... 104

6.1 Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.15$. ......................................................... 108
6.2 Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.25$. ......................................................... 108
6.3 Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.50$. ......................................................... 108
6.4  Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.75$. ........................................ 109

6.5  Total head coefficients distribution along a vertical line at the centre of the wing wake for $h/c = 0.10$ at a number of incidences. ............ 110

6.6  Static pressure coefficients on vertical line one chord length behind wing at $0^\circ$ incidence from 2D CFD. ........................................ 110

6.7  Momentum deficit coefficient in wake of front wing model. .......... 111

6.8  Non-dimensional position of front wing wake centre relative to quarter-chord. .......................................................... 112

6.9  Momentum deficit coefficient for wake of cylindrical rod. .......... 113

6.10 Smoke flow visualisation of lower endplate vortex with and without the moving road, $h/c = 0.40$. ................................. 113

6.11 Estimated vortex breakdown location behind single wing of tandem pair estimated from smoke-flow visualisation and lift curve slope at $h/c = 0.40$. ........................................ 114

6.12 Vortex trajectories relative to endplate trailing edge from front wing model at a number of ground clearances for an incidence of $5^\circ$. .... 114

6.13 Basic vortex system of front wing model in ground effect. ........... 115

6.14 Circulation of endplate vortices calculated from their trajectories. ... 115

6.15 Total head contours in wake of vertical wing at $x/c = 2.57$ at an incidence of $15^\circ$ and a number of ground clearances. ................. 116

6.16 Averaged PIV data at $x/c=2.0$ behind vertical wings at $\alpha = 15^\circ$ at a number of tip ground clearances. Spanwise separation $b/c=4.29$ (other wing to left, and arranged as mirror image). ............... 117

6.17 Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.07$. ............. 118

6.18 Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.14$. ............. 119

6.19 Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.21$. ............. 119
6.20 Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.29$.  
6.21 Power spectral density from hot-wire measurements at $x/c = 2.57$ in vertical wing vortex core at a number of ground clearances.  
6.22 Normalised power spectral density from the hot-wire measurements at $x/c = 2.57$ in vertical wing vortex core at a number of ground clearances.  
6.23 Vortex centre locations at 20 instantaneous points in time as taken from PIV data for the vertical wing vortex at a tip ground clearance of $h/c = 0.21$.  
6.24 Variation in mean velocity normal to hot wire and RMS velocity fluctuation with vertical wing tip ground clearances at time-averaged core centre location.  
6.25 Simulated vortex $y$ location, axial and tangential components of velocity at the hot wire location, and component normal to the wire.  
6.26 Simulated power spectral density and normalised power spectral density.  
6.27 Power spectral density from hot wire measurements in vortex core for vertical wing tip ground clearance of $h/c = 0.21$ at a range of free stream velocities.  
6.28 Mean velocity and RMS velocity fluctuations in vortex core versus tip ground clearance at $x/c = 2.57$.  
6.29 PIV data for moving ground plane at $x/c = 0.86$ and tip ground clearance $h/c = 0.21$.  
6.30 PIV data for moving ground plane at $x/c = 1.71$ and tip ground clearance $h/c = 0.21$.  
6.31 PIV data for moving ground plane at $x/c = 2.57$ and tip ground clearance $h/c = 0.21$.  
6.32 PIV data for stationary ground plane at $x/c = 0.86$ and tip ground clearance $h/c = 0.21$.  
6.33 PIV data for stationary ground plane at $x/c = 1.71$ and tip ground clearance $h/c = 0.21$.  

6.34 PIV data for stationary ground plane at \( x/c = 2.57 \) and tip ground clearance \( h/c = 0.21 \). .......................................................... 126
6.35 PIV data for two sets of vertical wings, \( h/c = 0.21 \). ......................... 127
6.36 PIV data for two sets of vertical wings, \( h/c = 0.29 \). ......................... 127

7.1 Total head contours in the wake of the aspect ratio 1 wing at a ground clearance of \( h/c=0.05 \) with vertical wings upstream at a tip ground clearance of \( h/c_{rear}=0.10 \). .......................................................... 132
7.2 Total head contours in the wake of the aspect ratio 1 wing at a ground clearance of \( h/c=0.08 \) with vertical wings upstream at a tip ground clearance of \( h/c_{rear}=0.10 \). .......................................................... 132
7.3 Total head contours in the wake of the aspect ratio 1 wing at a ground clearance of \( h/c=0.10 \) with vertical wings upstream at a tip ground clearance of \( h/c_{rear}=0.10 \). .......................................................... 132
7.4 Vortex path beneath aspect ratio 1 wing at ground clearances of \( h/c=0.08 \) (blue) and 0.10 (red) for vertical wing tip ground clearance of \( h/c_{rear}=0.10 \). .......................................................... 133
7.5 Change in surface pressure coefficients, \( \Delta C_p \), on the rear wing for an upstream vertical wing ground clearance \( h/c = 0.08 \). ......................... 134
7.6 Change in surface pressure coefficients, \( \Delta C_p \), on the rear wing for an upstream vertical wing ground clearance \( h/c = 0.10 \). ......................... 134
7.7 Effect of vertical wing tip ground clearance, \( h/c_{rear} \), on diffuser model downforce coefficient. \( \theta = 12^\circ, h/c = 0.06 \) ......................... 136
7.8 Effect of vertical wing tip ground clearance, \( h/c_{rear} \), on diffuser model underbody centreline pressures. \( \theta = 12^\circ, h/c = 0.06 \) ......................... 137
7.9 Effect of vertical wing tip ground clearance, \( h/c_{rear} \), on diffuser model underbody centreline pressure at \( x/c = 0.28 \). \( \theta = 12^\circ, h/c = 0.06 \) ......................... 138
7.10 Effect of vertical wing tip ground clearance, \( h/c_{rear} \), on diffuser model pitching moment. \( \theta = 12^\circ, h/c = 0.06 \) ......................... 138
7.11 Effect of vertical wing tip ground clearance on diffuser model downforce with different diffuser ramp angles. \( \theta = 12^\circ, h/c = 0.06 \) ......................... 139
LIST OF FIGURES

7.12 Effect of vertical wing tip ground clearance on diffuser model centre-line underbody pressure distributions for diffuser angles of $\theta = 10^\circ$ and 14°. $h/c = 0.06$. .................................................. 139

7.13 Effect of vertical wing tip ground clearance on diffuser model under-body centreline pressure at $x/c = 0.80$ with different diffuser ramp angles. $h/c = 0.06$. .................................................. 140

7.14 Semi-spanwise pressure distributions at $x/c = 0.28$ with vertical wings at $h/c_{rear} = 0.04$, 0.05, and 0.06. $\theta = 12^\circ$, $h/c = 0.06$. .......... 141

7.15 Effect of vertical wing spanwise separation, $b/c_{rear}$, on diffuser model downforce for $h/c_{rear}=0.01$ and 0.05, with conventional and unconventional vortex configurations. $\theta = 12^\circ$, $h/c = 0.06$. ................. 142

7.16 Semi-spanwise pressure distributions on the underbody at $x/c = 0.28$ for a number of spanwise separations of the vertical wings at tip ground clearances of $h/c_{rear}=0.05$. $\theta = 12^\circ$, $h/c = 0.06$. ............... 143

7.17 Centreline underbody pressure variation at $x/c = 0.28$ and 0.80 versus lateral separation of vertical wings at $\alpha = -10^\circ$ and 15° at a tip ground clearance $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$. ............... 144

7.18 Centreline underbody pressure distributions with and without vertical wings at $\alpha = 15^\circ$, $h/c_{rear} = 0.05$, and $b/c_{rear} = 0.90$. $\theta = 12^\circ$, $h/c = 0.06$. .................................................. 144

7.19 Pitching moment coefficient versus lateral wing separation at $\alpha = -10^\circ$ and 15°, $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$. ............... 145

7.20 Effect of vertical wing spanwise separation, $b/c_{rear}$, on diffuser model downforce for two wing pairs, with $\alpha_1 = 10^\circ$ and $\alpha_2 = 15^\circ$, and for a single wing pair at $\alpha = 15^\circ$, all at $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$. 146

7.21 Effect of vertical wing spanwise separation, $b/c_{rear}$, on diffuser model downforce for two wing pairs, with $\alpha_1 = -10^\circ$ and $\alpha_2 = 15^\circ$, and for a single wing pair at $\alpha = 15^\circ$ and $\alpha = -10^\circ$, all at $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$. .................. 148

8.1 Schematic diagram showing possible distortion of front wing wake. . . 153
LIST OF FIGURES
8.2

Variation in diffuser model downforce with cylindrical rod ground
clearance for a number of rod diameters. 9 = 12°, h/c = 0.06

8.3

^

153

Variation in centreline underbody pressures at xjc = 0.28 and 0.80
with ground clearance of ^jcrcar = 0.016 cylindrical rod. Q = 12°,
h/c = 0.06

8.4

Variation in downforce with ground clearance of (j)/hreaT = 0.016
cylindrical rod for 9 = 10°, 12°, and 14° . h/c = 0.06

8.5

154

155

Variation in centrehne underbody pressures at x/c = 0.28 and 0.80
with ground clearance of 4'/crear = 0.016 cylindrical rod with 6 = 10°
and 6 = 14°. h/c = 0.06

8.6

155

Variation in diffuser model downforce with upstream full-span {h/crear =
1.0) wing ground clearance for a number of wing incidences. 6 = 12°,
h/c = 0.06

8.7

157

Changes in diffuser model underbody centreline pressures oX x/c —
0.28 and 0.80 versus upstream full-span {h/crear = 10) wing ground
clearance at a = 0, 4, and 8°. 9 = 12°, h/c = 0.06

8.8

157

Variation in diffuser model downforce with upstream full-span ih/crear =
1.0) wing ground clearance for a number of wing incidences. 9 = 10°,
h/c = 0.06

8.9

159

Variation in diffuser model downforce with upstream full-span {h/crear =
1.0) wing ground clearance for a number of wing incidences. 9 = 14°,
h/c = 0.06

159

8.10 Variation in diffuser model downforce with upstream partial-span
{h/crear = 0.60) wing ground clearance for a number of wing incidences. 9 = 10°, h/c = 0.06

160

8.11 Variation in diffuser model downforce with upstream partial-span
{h/crear = 0.60) wing ground clearance for a number of wing incidences. 9 = 12°, h/c = 0.06

160

8.12 Variation in diffuser model downforce with upstream partial-span
{h/creaT = 0.60) wing ground clearance for a number of wing incidences. 9 = 14°, h/c = 0.06

160


8.13 Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01, 0.03$ and 0.05 at $\alpha = 4^\circ, \theta = 10^\circ, h/c = 0.06$ .................................................. 161
8.14 Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01, 0.03$ and 0.05 at $\alpha = 4^\circ, \theta = 12^\circ, h/c = 0.06$ .................................................. 161
8.15 Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01, 0.03$ and 0.05 at $\alpha = 4^\circ, \theta = 14^\circ, h/c = 0.06$ .................................................. 161
8.16 Difference in diffuser model underbody centreline pressure at $x/c = 0.80$ for different upstream horizontal incidences versus wing ground clearance for $\theta = 10^\circ$ and $\theta = 14^\circ, h/c = 0.06$ .................................................. 162
8.17 Semi-spanwise pressure distributions on diffuser model underbody at $x/c = 0.28$ with the partial-span horizontal wing at $\alpha = 8^\circ$ and $h/c_{rear} = 0.01, 0.04, 0.08. \theta = 12^\circ, h/c = 0.06$ .................................................. 163

C.1 Photograph of internal wind tunnel balance showing lift and drag cells. 185
C.2 Shear strain (in the same plane as the view) in one of the two balance units when a pure lift and pure drag force is applied. .......................... 186
D.1 Assumed ‘lateral volume-flow rate’ distribution, $\frac{\partial Q}{\partial x}$. ....................... 191
List of Tables

3.1 PIV setup parameters and calculated perspective errors. ............. 65

4.1 Sectional lift coefficients at wing centreline and near wing tip with
and without endplates. .................................................. 86

B.1 Tandem wing model section co-ordinates. ............................ 182
B.2 NASA LS(1)-0416 wing section co-ordinates. ......................... 184
Nomenclature

Roman

\( A \) \hspace{1cm} \text{Area}

\( AR \) \hspace{1cm} \text{Aspect Ratio}

\( AR \) \hspace{1cm} \text{Area Ratio}

\( B \) \hspace{1cm} \text{Blockage ratio}

\( b \) \hspace{1cm} \text{Model span or lateral separation between quarter-chords of vertical wings}

\( C_{Di} \) \hspace{1cm} \text{Induced drag coefficient}

\( C_{La} \) \hspace{1cm} \text{Lift curve slope}

\( C_M \) \hspace{1cm} \text{Pitching moment coefficient}

\( C_p \) \hspace{1cm} \text{Pressure coefficient}

\( C_z \) \hspace{1cm} \text{Coefficient of z-force, or downforce}

\( c \) \hspace{1cm} \text{Wing chord or model length}

\( f \) \hspace{1cm} \text{Frequency}

\( G \) \hspace{1cm} \text{Momentum}

\( h \) \hspace{1cm} \text{Ground clearance}

\( k \) \hspace{1cm} \text{Turbulence kinetic energy}

\( Q \) \hspace{1cm} \text{Volume flow rate}

\( r \) \hspace{1cm} \text{Radius, or distance from a defined point or centre}

\( R_e \) \hspace{1cm} \text{Reynolds number}

\( t \) \hspace{1cm} \text{Time}

\( u_\theta \) \hspace{1cm} \text{Tangential velocity}

\( u_z \) \hspace{1cm} \text{Axial velocity}
$U$ Velocity

**Greek**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Incidence</td>
</tr>
<tr>
<td>$\Gamma$</td>
<td>Circulation</td>
</tr>
<tr>
<td>$\delta^*$</td>
<td>Boundary layer displacement thickness</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Dissipation rate of turbulence kinetic energy</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Diffuser ramp angle</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Molecular viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Diameter</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Vorticity</td>
</tr>
</tbody>
</table>

**Subscripts**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\infty$</td>
<td>Flow conditions far upstream</td>
</tr>
<tr>
<td>1</td>
<td>Conditions at diffuser inlet</td>
</tr>
<tr>
<td>2</td>
<td>Conditions at diffuser outlet</td>
</tr>
</tbody>
</table>

**Abbreviations**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two-dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three-dimensional</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Full Form</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------</td>
</tr>
<tr>
<td>DPIV</td>
<td>Digital Particle Image Velocimetry</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>PSD</td>
<td>Power Spectral Density</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier Stokes Equations</td>
</tr>
<tr>
<td>RMS</td>
<td>Root mean square</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds stress turbulence model</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Problem Definition

Since 1968 aerodynamics has been used in Formula One to increase the adhesion of the wheels to the ground, increasing cornering speeds and braking performance dramatically (Wright, 1983). The aerodynamic performance of a Formula One car is now one of the most important areas of development. For example, an increase in downforce of just 1% can give a reduction in lap-time of 0.1s (for a lap time of approximately 80s) around the Silverstone Grand Prix circuit (Dominy, 1992). This gain is relatively easy to achieve compared to developments in any other areas of the car design (Agathangelou and Gascoyne, 1998) which explains the heavy investment by all Formula One teams in improving the car's aerodynamics, with many teams testing in more than one wind tunnel around the clock in the search for better performance.

Other than optimising the aerodynamic performance of the car, the demands on the aerodynamic design include providing adequate airflows to the various cooling devices (Wright, 1983), creating the correct balance of aerodynamic forces between the front and rear axles, a factor vital to the car's handling (Dominy, 1981), and complying with the strict regulations set down by the FIA (Federation Internationale de l'Automobile who govern the sport) regarding the size and placement of the various components. These tasks are complicated by the presence of the large open
wheels (Agathangelou and Gascoyne, 1998) and the proximity to the moving ground plane.

Modern Formula One cars consist of an elaborate arrangement of aerodynamic devices in order to meet the requirements outlined above. All of these devices interact with one another to varying degrees, and the aerodynamics of the car as a whole are highly complex. For this reason, the aerodynamic design is a complicated task, accomplished on an iterative basis predominantly through extensive wind tunnel testing.

Figure 1.1 shows a diagram of a typical Formula One car with some of the more important aerodynamic devices pointed out. Of particular interest to the present study is the front wing assembly whose wake and trailing vortices pass between the underbody and the ground. The barge boards, further downstream, also generate vortices which pass beneath the car body.

Figure 1.1: Typical Formula One car (Piola, 2000).
The front wing of a Formula One car is perhaps the easiest to optimise in its own performance since it is positioned at the front of the car and operates in an undisturbed air stream. However, many of the components downstream of this are heavily influenced by the front wing. It is therefore vital in the design of a Formula One car that the correct compromise is made between the optimisation of the front wing performance, and minimisation of the detrimental effects that it may have on those parts of the car downstream of it. One of the most important of these is the car underbody and diffuser, which can pay a heavy penalty in loss of performance.

These interaction mechanisms are extremely complex due to the number of aerodynamic devices involved, together with the other components of the car such as the wheels, and the physics of the interactions are not well understood. It would therefore be beneficial to investigate the aerodynamic interaction mechanisms between these components, and in particular between the front wing and the flat underbody and diffuser since these are amongst the most important aerodynamic features.

1.2 Aims and Objectives

The project has taken a fundamental approach, breaking the problem down into simple components, from which an understanding of the governing mechanisms can be constructed. The aims and objectives of the study are as follows.

1.2.1 Aims

1. To improve the understanding of the flow physics governing the behaviour of a Formula One car underbody and diffuser.

Wind tunnel measurements and CFD are used on simplified models representing a Formula One car body with the aim of identifying the aerodynamic behaviour and flow field associated with the flat underbody and diffuser in particular, including their interaction with the ground.

2. To build a qualitative description of the front wing wake.
In order to explain the interaction mechanisms between the front wing and the underbody and diffuser it is vital that the structure and behaviour of the wake of a horizontal front wing in close proximity to the ground plane is appreciated. A variety of wind tunnel techniques are used on a number of simplified models to this end.

3. **To identify the mechanisms involved in the interaction between the front wing and underbody and diffuser of a Formula One car.**

The knowledge gained through the first two aims is combined with wind tunnel experiments on various combinations of models to investigate the flow physics governing the influence of the front wing on the underbody and diffuser of a Formula One car. The ultimate aim is to understand how the wing-body combination can best be configured to optimise the downforce produced by the car body.

### 1.2.2 Objectives

1. **To use a variety of investigative methods to identify the major features of a front wing wake.**

Diagnostic wind tunnel techniques such as PIV and total head measurements are used to investigate the front wing wake. The objective of these tests is to identify the basic components of the front wing wake and the flow field associated with them.

2. **To design models which simulate the identified features of the front wing wake.**

Following the investigation of the front wing wake, wind tunnel models are constructed with the aim of separately generating each of the components of the front wing wake.

3. **To design a model which can be used to simulate the basic features of a Formula One car underbody and diffuser, and use this model to investigate their aerodynamic properties and behaviour.**
To keep the investigation at a fundamental level, a wind tunnel model is required which can be used to simulate the basic aerodynamic features of the underbody and diffuser of a Formula One car in ground effect. This model is investigated in the wind tunnel using a variety of techniques to identify its aerodynamic characteristics and the flow mechanisms governing its behaviour.

4. To use the models described above to study how each of the defined features of the front wing wake influence the rear body.

By placing the models designed to simulate the various features of the front wing wake upstream of the diffuser and measuring the forces and surface pressures on it, the influence of each of the front wing wake components on the diffuser model may be studied.

5. To produce guidelines for how the front wing wake can best be used to minimise its negative influence on the performance of the car underbody and diffuser.

The ultimate objective of the project is to bring together the knowledge gained throughout its course to produce guidelines for how the front wing may be configured to minimise the loss in performance it induces on the car underbody and diffuser.
Chapter 2

Review of Literature and Relevant Theory

2.1 Introduction

The aim of the present investigation is to evaluate the influence of the front wing on the aerodynamic performance of the flat underbody and diffuser. Before embarking on this study, there are a range of aerodynamic topics which require some research in order to provide a background understanding on the subject. A review of the relevant literature and theory is provided in the following chapters.

2.2 Ground Effect Aerodynamics

Formula One cars operate at a very low clearance between the underbody and the road (typical ground clearances being of the order of between a few, and a few tens of millimeters depending on the circuit and car configuration). The close ground proximity has a profound and rather complex effect on the aerodynamics of the car. Whether the aerodynamics are investigated in the wind tunnel, or with CFD, it is essential that the ground plane is modelled accurately, and its effects understood.

A review of the current understanding of the principal ground effects provides a useful background to this study. This will be followed by a deeper look at
the influence of ground proximity on the aerodynamics of wings in general,
negative lifting wings with endplates (such as a Formula 1 car front wing),
bluff bodies (representing the bulk of the car body), and automotive diffusers.

2.3 Vortical Flows

Streamwise vortices are shed from many of the aerodynamic devices on a
Formula 1 car, and those shed from the front wing endplates and flaps, and
the barge boards are likely to have some significant interaction with the car's
flat underside and diffuser (see Figure 1.1). The formation, structure and
stability of these vortices will be discussed.

2.4 Aerodynamic Interaction Mechanisms

An account of the current understanding of a number of basic interaction
mechanisms will be given, from simple two-dimensional interactions, such as
those between cylinders and wings, to interactions more specific to racing cars
such as those between the car wings and body. The interactions of vortices with
stationary and moving boundaries, such as the ground plane, and with wings
will also be discussed in some detail. The aim is to build an understanding
of how the front wing and barge boards are likely to effect the car underbody
and diffuser.

2.2 Ground Effect Aerodynamics

"Ground Effect" is a phenomenon of enormous importance to racing car designers.
by which a body's lift force at fixed incidence is augmented and its induced drag, for
fixed lift, reduced due to its proximity to the ground plane. These effects are used
extensively by many of the aerodynamic components of Formula One cars, from
the wings to the flat underbody and diffuser. A review of the various mechanisms
governing ground effect, together with a careful consideration of the importance of
ground plane simulation both in experimental and numerical investigations, forms
an essential background to this study.
2.2.1 Streamlined bodies in ground effect

The effect of ground proximity on the performance of lifting wings has been studied since early in the 20th Century, with reference to its influence on aircraft performance on take-off and landing (Raymond, 1921; Reid, 1928). Initially, the reduction in induced drag and increase in lift for any fixed positive incidence were the primary concerns of these experimental studies.

Wieselsberger (1922) was amongst the first to offer a means of predicting the ground effect by considering the changes in induced drag for a wing near the ground. He did this by calculating the upwash generated by the trailing vortices at the lifting line. The vortex system was mirrored about the ground plane, ensuring zero normal component of velocity everywhere on it, as shown in Figure 2.1.

![Image of wing and ground plane](image.png)

**Figure 2.1:** Schematic of image horseshoe vortex system used to simulate ground plane.

For a positive lifting wing the image trailing vortices reduce the downwash at the lifting line, and so reduce the induced drag. For a negative lifting wing, the image vortices reduce the upwash, but since the lift is negative the induced drag is still reduced and the theory is equally applicable. The resulting equation for the induced drag is a function of the lift coefficient as follows:

\[ C_{Di} = (1 - \sigma) \frac{C_L^2 S}{\pi b^2} \]

where \( \sigma \), the influence coefficient, is defined as follows to hold between \( 1/30 < h/b < 1/4 \):
\[ \sigma = \frac{1 - 1.22h/b}{1.05 + 7.4h/b} \]

where, \( h \) is the distance between the lifting line of the wing and the ground plane.

Comparing his theory to experiment, Wieselsberger found it to hold only for moderate lift coefficients (below 1.0), a result which he could not explain at the time.

Although not explicitly considered by Wieselsberger, his theory also suggests an increase in lift-curve slope, \( dC_L/d\alpha \). The upwash generated by the image trailing vortices is directly proportional to the lift coefficient and the ground clearance. For small values of the upwash the effective incidence change is therefore also directly proportional to the lift coefficient and ground clearance such that the change in \( dC_L/d\alpha \) is positive and directly proportional to the ground clearance.

Interest in low-flying ground effect vehicles has resulted in a number of works studying the effects of low aspect ratio wings in ground effect. Examples include the experimental studies of Fink and Lastinger (1961) and Carter (1961), who used different simulations of the ground plane. Fink and Lastinger (1961) carried out an extensive experimental survey of lift, drag and pitching moment on wings of varying aspect ratio and positive camber, comparing their results to Weiselsberger's theory. They found that for all aspect ratios tested (1, 2, 4, and 6) the proximity to the ground increases the lift-curve slope and reduces the induced drag, as predicted. However, they found that the angle of attack for zero lift (\( \alpha_0 \)) rises, an effect not predicted by the theory of Wieselsberger. Increasing aspect ratio, meanwhile, has no effect on the angle of attack for zero lift, but increases the lift-curve slope and induced drag. By consideration of the effect of ground clearance on lift to drag ratio \( (L/D) \), the influence of ground proximity is likened to an increase in effective aspect ratio. At low ground clearance, the ground plane 'image vortices' reduce the downwash at the lifting line, effectively reducing the influence of the trailing vortices, as for a greater aspect ratio. Plotting the effective aspect ratio in ground effect as a fraction of that out of ground effect (for which they use the symbols \( A_e/A_{e,\infty} \)) against ground clearance, the results for all aspect ratios collapse onto a single curve. This curve
is in excellent agreement with the theory of Wieselsberger down to the minimum ground clearance for which he states it to be valid \((h/b = 1/30)\). This plot is shown in Figure 2.2. The solid line shows the predictions from the theory of Weiselsberger, with dotted parts indicating the values of \(h/b\) for which he did not consider his theory to be valid \((h/b < 1/30\) and \(h/b > 1/4)\). Note that the effective aspect ratio below this range is actually greater than predicted.

![Figure 2.2: Effect of ground proximity on the effective aspect ratio of a positive lifting wing in ground effect. (Fink and Lastinger, 1961).](image)

The increase in \(\alpha_0\) with reduced ground clearance is not explained by the effective aspect ratio increase, since Weiselsberger’s theory is based on the effect of the image horseshoe vortex system, which of course has zero strength for zero lift. Furthermore, from their pitching moment measurements, Fink and Lastinger found an increase in nose-up pitching moment with reduced ground clearance at low lift coefficients, and an increase in longitudinal static stability at positive angles of attack through a reduction in \(dC_M/dC_L\), although they offer no explanation for it. Similar results were found by Carter (1961), in his experiments towing a low aspect ratio wing over a water channel. For positive angles of attack, the nose-up pitching moment
Chapter 2

becomes more negative with reduced ground clearance. They offered the explanation that the increment in lift comes about through an increase in ‘ram pressure’ on the lower surface which is more uniformly distributed over the lower surface at lower ground clearances, tending to move the aerodynamic centre rearward. This is not an entirely satisfactory explanation since ram pressure is by no means a scientific term. The simple horseshoe vortex methods which they used presume a fixed location of the lifting line, and so are unable to predict changes in loading on a wing in ground effect.

Tani, Taima and Simidu (1938) made considerable enhancements to the model of Wieselsberger (1922) by considering the effects of thickness and chordwise loading on the ground effect. A distribution of vorticity along the chord allows changes in wing loading and thus pitching moment due to its image in the ground plane to be evaluated. Meanwhile, the effect of thickness was modelled by a single doublet and its image. Their model gives the following equation for the change in pitching moment for a 2-D wing section in ground effect:

$$\delta C_M = D C_L - E e$$

where D and E are positive constants depending on the ground clearance, and e is the wing thickness ratio. Here $C_M$ is defined as positive nose-down. The first term represents the effects of the bound vorticity and the second the effects of thickness. The terms have opposing effects for positive lift values, and the net result clearly depends on the amount of lift being generated. At negative and low levels of positive (away from the ground) lift, proximity to the ground increases the nose-up pitching moment, whilst at larger levels of lift the effect is nose-down.

Bagley (1960), following the approach of Tani et al., represented the 2-D wing thickness as a distribution of sources along the chord line, as opposed to a single doublet and its image. This gives a better simulation of the distributed effects of thickness on the ground effect phenomenon.

The consequences of wing thickness may be explained by consideration of a symmetrical aerofoil at zero incidence. Far from the ground, this provides zero lift and may be modelled purely as a distribution of sources. Creating the correct
distribution of thickness requires positive sources near the nose, near-zero sources along the centre, where there is little change in thickness, and negative sources (sinks) towards the trailing edge. In close proximity to the ground the image sources create an upward vertical component of velocity at the leading edge, whilst at the trailing edge the image sinks generate a downward component of velocity. This results in a 'curved' airflow, increasing the effective incidence of the wing at the leading edge, and reducing it at the trailing edge. To maintain the Kutta condition at the trailing edge a circulation is now required, and thus a (negative) lift force generated. A schematic illustration of this effect is shown in Figure 2.3.

![Figure 2.3: Schematic diagram showing likely effect of wing thickness on local flow in ground effect due to image sources.](image)

Bagley described this as 'negative induced camber'. The distribution of vertical velocity component means that the zero-camber wing now has an effective camber relative to the locally 'curved' airflow, producing downforce at zero incidence and increasing the angle of attack for zero lift. By comparison with experiment, Tani et al. and Bagley both found their theories to be in good agreement with experiment but only for relatively low lift coefficients and ground clearances greater than approximately half of the chord. Bagley suggested that this was due to boundary layer growth, and showed that by providing for this (presumably by modification of the aerofoil shape) this error could be removed, although this required prior knowledge of the flow from experiment.

In addition to the trends in the pitching moment coefficient mentioned previously, Bagley also noted a rearward movement of the stagnation point on the lower surface
of the aerofoil. Referring to the schematic diagram of Figure 2.3, this is a result which could be expected from the 'negative induced camber' effect, and is thus an effect of thickness.

Berry (1968) extended the 2-dimensional theory of Bagley to three dimensions by incorporating the effective incidence change due to thickness in 2-D discussed above into the image trailing vortex model for induced drag and lift-curve slope. He compares this theory and the simple trailing vortex model to experiments, which he made over a static and moving ground plane. The agreement between theory and measured lift curve slopes is excellent for the moving ground plane, and less good for the static ground plane. The prediction of the change in effective angle of attack due to thickness is not so good, nor is the prediction of induced drag. However, it is a significant improvement on the simple trailing vortex model, and so demonstrates that taking account of the wing thickness is a necessary consideration in the modelling of ground effect.

2.2.2 Ground plane simulation and Reynolds Number effects

In the previous section, various forms of ground simulation have been mentioned. Ground simulation methods have been the primary subject of a large number of studies (Berry, 1968; Fackrell, 1975; Burkin, Adey and Beatham, 1986; Sardou, 1986; Fago, Lindner and Mahrenholtz, 1991; Kim and Geropp, 1998). Even in the earliest investigations of ground effect a flat plate simulation of the ground plane was thought of as unsatisfactory since it is stationary relative to the model (Raymond, 1921). This led to the introduction of the 'image' method of ground simulation, which involves a second model to 'mirror' the primary model about an imaginary ground plane. This method has the advantage of removing the incorrect boundary layer growth on a stationary plane. However, since this method is also inaccurate as the velocity gradient at the wall is completely removed (Fago et al., 1991). Since the flow in most cases is accelerated beneath the wing, a boundary layer is produced on a ground plane moving at the free stream velocity which is not represented in the
reflection method. Figure 2.4 shows schematic illustrations of the velocity profiles beneath a body in close proximity to the ground for the floor simulation methods mentioned above.

![Figure 2.4: Illustration showing effect of ground-plane simulation on velocity profiles beneath a body in ground effect.](image)

This would suggest that the only way to correctly simulate the velocity profile at the ground in the wind tunnel is by means of a moving ground plane with some form of upstream boundary layer removal. Berry (1968) used a rolling road consisting of a rubber and fabric continuous belt running over two rollers, upstream of which the boundary layer was bled away via a duct. This gave a velocity down to the ground which did not drop below 82% of the freestream. To demonstrate its effectiveness, he conducted experiments on a NACA 0010 symmetrical wing with an aspect ratio of 2.35 over a range of incidences and ground clearances. Curves showing lift coefficients over the range of incidences and ground clearances tested are shown in Figure 2.5, showing considerable differences between the lift curves given by the two ground simulation techniques.

The importance of ground simulation on road vehicle aerodynamics has been the subject of some considerable debate (Fago et al., 1991) and it would seem that the influence of ground plane simulation is to some extent a function of underbody
 geometry, with smooth venturi type underbodies being more dependant than rough underbodies consisting of many mechanical parts (Sardou, 1986). For racing car models which make heavy use of underbody flow to generate downforce, a stationary ground simulation can result in a loss in downforce of up to 100% compared with the moving ground (Burkin et al., 1986). The effect of the ground simulation also varies with body shape, with either gains or losses in both lift and drag with and without the ground plane movement. Kim and Geropp (1998) conducted separate experiments on a cylinder and on a two-dimensional car model in the proximity of a stationary and moving ground plane. They found that the moving plane increases the lift (away from the ground) for the cylinder, whilst reducing it considerably (reversing the sign in some cases) for the car model. Studies of body wakes in ground effect have shown significant changes in the flow structure between stationary and moving ground planes (Davis, 1982; Sardou, 1986). As an example, Figure 2.6 shows streamlines from a two-dimensional viscous numerical simulation of the flow around a car shape with a stationary and a moving ground plane (Sardou, 1986). With the
ground moving, the stagnation point shifts downwards, resulting in separation of flow on the car bonnet, which isn't present with the stationary ground. The wake structure is also considerably different with the ground moving. The effect of the ground simulation on aerodynamic forces is complex, depending very much on the body geometry. A moving ground simulation with upstream boundary layer removal is therefore essential for wind tunnel investigations of the aerodynamics of bodies operating in ground effect. This is equally important for numerical simulations, where the ground plane is simulated as a moving wall.

Reynolds number is another important consideration for road vehicle aerodynamics. It has already been shown that the correct simulation of boundary conditions, and therefore boundary layer growth beneath bodies in ground effect can have an enormous influence on the aerodynamic forces and flow structure. Since boundary layer growth depends on Reynolds number, it is an important consideration for road vehicle aerodynamics. Sardou (1986) noted strong Reynolds number dependence for his saloon car model with a venturi underbody, with a definite transition in flow states at a Reynolds number based on model length of $4 \times 10^6$. Meanwhile Burkin et al. (1986) found strong dependence of the drag of a model of a truck on Reynolds number. In both studies, indications are of no general tendency in the direction of
changes in aerodynamic forces with Reynolds numbers, but more of changes in flow regime.

2.2.3 Effect of Endplates

Endplates are used most commonly on racing car wings (for example see Figure 1.1. Their influence on the aerodynamics of wings therefore needs to be considered as part of the present study of interaction effects relevant to Formula One. Generally, they consist of thin vertical plates mounted at the tips of a wing with their plane perpendicular to its axis. They first appeared in aeronautical applications in the 1920s in an attempt to find an economical means of increasing the effective aspect ratio of a wing by limiting flow leakage around the tips. Reid (1925) conducted wind tunnel tests on a wing of aspect ratio 6, with disk and trapezoidal endplates, finding that they gave the anticipated improvement in induced drag, lift curve slope, maximum lift and maximum L/D. The only detrimental effects were at low lift coefficients where the added parasite drag of the endplates is greater than the improvement in induced drag, resulting in a lower L/D. Hemke (1927) showed that the reduction in drag offered by endplates increases as the aspect ratio is reduced which is in agreement with Reid's description of the endplates as offering an increased effective aspect ratio. A typical drag polar from his results is shown in Figure 2.7, showing the reduction in drag exists only at larger lift coefficients (in this case $C_L > 0.4$). Given that aircraft are designed generally for minimum drag in the cruise, where the lift coefficient (and therefore the improvement in performance offered by endplates) is low, and that they have relatively large wing spans, it is not surprising that endplates have not become common appendages to modern day aircraft wings.

Hemke suggested that fairing the endplate cross-section could be beneficial to the endplate profile drag, and that endplate shape could have some influence on the benefits. Subsequently, Riley (1951) carried out an investigation of a large number of endplates of different shapes, sizes, and cross-sectional profile. Increasing the endplate area improved $C_{L\alpha}$, $C_{L\text{max}}$ and $\partial C_{Di}/\partial C_L^2$. Some of his endplates were
the shapes of contours of constant static pressure around the aerofoil for infinite aspect ratio and zero incidence, and he found these to be more favourable than rectangular endplates, presumably due to a better use of the added wetted area. Similarly, endplates with more of their area on the suction side of the chord line were more beneficial. Furthermore, concentrating area towards the trailing edge appeared to show some small benefit in the induced drag with a small reduction in $\partial CD_i/\partial C_L^2$. In all cases, the maximum lift to drag ratio was reduced by the addition of endplates, but they did have the effect of increasing the lift coefficient at $(L/D)_{\text{max}}$, and increasing the range of lift coefficients over which $L/D$ was at a maximum.

Some of the references quoted above involve some potential flow modelling of the effects of endplates, but with little explanation of the basic mechanisms governing the endplate effects. These can be better understood by considering the flow in the vicinity of the endplates, and contemplating the likely effects on the wing

Figure 2.7: Typical results for the effects of endplates on the drag polar for a wing of aspect ratio 6 (Hemke, 1927).
performance. The illustration of Figure 2.8 is provided as an aid. The endplate shown is a simplification of the complex shapes used on modern Formula One cars (as shown in Figure 1.1, but is fundamentally similar in concept. The pressure is generally more negative on the suction surface than it is positive on the pressure surface, and so there will be a greater flow over the edge of the endplate on the suction side, resulting in a greater leakage on that edge. Subsequently, the stronger of the two endplate vortices will be generated from that edge. It is not surprising therefore that there is a benefit from increasing the height of the endplate on the suction side, as its reduction in leakage will have the greater effect. Furthermore, the effect of the trailing vortices on the induced drag and lift curve slope will be lessened if the vortices are moved further from the lifting line. By increasing the height of the endplate on the suction side, and concentrating that height towards the trailing edge, the stronger vortex is moved away from the wing. This increases the effective aspect ratio, and increases the loading towards the tips by reducing the air leakage at the tip on the suction side, allowing a lower pressure to be maintained.

![Figure 2.8: Schematic diagram of flow field in vicinity of endplates.](image)

Open-wheeled racing car wings typically have low aspect ratios and operate at high lift coefficients. Reid’s findings back in 1925 would suggest that endplates lend themselves very well to racing car wings, and indeed their widespread use as such is a testament to their efficiency. Some of the mechanisms discussed in the proceeding sections, however, are slightly different in the case of racing car wings. Factors contributing to this difference are discussed in the following section.
2.2.4 Factors Pertinent to Negative Lifting Wings with End-plates in Ground Effect.

It has been established that one of the effects of the ground is to introduce a 'negative camber' which results in a negative lift, or downforce at low incidence, and increases downforce at negative incidence. As the ground is approached, the level of negative lift increases, but Stollery and Burns (1969) found that below a certain height, this trend reverses, and a further reduction in ground clearance results in a reduction in downforce. Although they state that this phenomenon depends critically on ground clearance and boundary layer growth, they do not attempt to explain it.

![Diagram](image)

Figure 2.9: Effect of ground clearance on lift curve slopes of low aspect ratio symmetrical wing in ground effect (Stollery and Burns, 1969).

Knowles, Donoghue and Finnis (1994) compared experiments on a negative lifting Formula 1 style wing, comparing the results to two-dimensional inviscid CFD and finding good agreement only at large ground clearances and low incidences. Although the range of ground clearances studied didn’t include the force reduction phenomenon, they did find that under high lift conditions, separation at the rear of the aerofoil limits the level of downforce generated.

Ranzenbach and Barlow (1994; 1996) conducted 2-dimensional viscous calculations on symmetrical and cambered wings at zero incidence and varying ground
clearances with the aim of explaining the force reduction phenomenon. They found that as the ground is approached the boundary layers on the ground and the wing suction surface approach each other and merge. They attributed the force reduction to this merging, since it will tend to block the flow of air beneath the wing. Indeed, the height at which the force reduction phenomenon occurred did correspond to that at which the boundary layers merged in their computations. It was found to be greater for the cambered wing, presumably due to faster boundary layer growth on the suction surface due to the greater adverse pressure gradient towards the trailing edge.

More recently, Zerihan and Zhang (2000) conducted a comprehensive study of a single element Formula 1 wing with endplates in ground effect, including force and surface pressure measurements, and surface flow visualisation. The wider range of data available allowed them to present a more detailed analysis of the force reduction phenomenon. They found the height at which force reduction occurs to be independent of Reynolds number. Since Reynolds number affects the boundary layer thickness, this would tend to suggest that the merging boundary layer theory of Razenbach and Barlow is inaccurate. They found that, for a fixed incidence, as the ground is approached the flow separates on the suction surface, and the separation line moves upstream with increasing ground proximity. At the same time increased ground proximity continues to produce an increasingly large peak suction on the lower surface. The optimum balance between these effects results in a height at which the downforce is at a maximum. The onset of trailing edge separation corresponds to a marked reduction in the lift curve slope, which can clearly be seen, especially at h/c=0.134 at an incidence of between 1 and 2°, on their lift curve slopes, shown in Figure 2.10.

Surface flow visualisation indicates a strong effect of the vortices inboard of the endplate on the suction surface side of the wing. They induce local change in incidence sufficient to keep the boundary layer attached near the wing tips. Zhang and Zerihan (2003) show that this vortex can induce a localised suction on the surface over which it passes, resulting in a non-linear component of downforce produced near the tips.
2.2.5 Bluff Body in Ground Effect

Design constraints on Formula One cars render it impossible to produce a low drag body that is not possible to make from ground effect. These constraints include the need to maintain at least a 6 mm gap at the ground. Jasinski and Selig (1998; Moseley, 1999) show the wake to be dominated by three vortices. Two are shed from the upper and lower edges of the endplate as suggested previously in Figure 2.8. Formula One car front wings commonly have variable chord flaps (see Figure 1.1), and where the flap chord changes there is an abrupt change in downforce, resulting in the shedding of a third trailing vortex. A schematic diagram of this vortex system is shown in Figure 2.11.

Figure 2.10: Lift curve slopes for a Formula 1 style wing in ground effect (Zerihan and Zhang, 2000).

Measurements of the flow field in the wake of a Formula 1 style wing with a flap and endplates close to the ground (Jasinski and Selig, 1998; Moseley, 1999) show the wake to be dominated by three vortices. Two are shed from the upper and lower edges of the endplate as suggested previously in Figure 2.8. Formula One car front wings commonly have variable chord flaps (see Figure 1.1), and where the flap chord changes there is an abrupt change in downforce, resulting in the shedding of a third trailing vortex. A schematic diagram of this vortex system is shown in Figure 2.11.

Figure 2.11: Schematic diagram of the vortex system in the wake of a single element wing with endplates in ground effect.
2.2.5 Bluff Bodies in Ground Effect

Design constraints on a Formula One car chassis, are such that it is not possible to taper it sufficiently at the rear to avoid a large separated wake. These constraints include the technical regulations which also specify that the majority of the car underbody must be flat. There are a number of considerations particular to bodies which are bluff and have flat underbodies which will be considered in this section.

Stollery and Burns (1969) carried out force measurements on a symmetrical rectangular wing sectioned wing of aspect ratio 2.35 which was progressively blunted at the rear by moving portions of the trailing edge. They reported that blunting the wing results in a significant increase in drag, and a minimal reduction in the magnitude of the lift due to a small reduction in lift curve slope (basing $C_L$ on the planform area of the complete wing). This would suggest that increasing bluntness does not have a large effect on the pressure distribution over the remaining wing. As the wing is progressively blunted, the surface pressure at the point on the surface of the wing where the portion is removed gets increasingly negative as it is moved upstream. This determines the base pressure, since the sharp edges at the rear of the blunted wing fix separation at that point. The increasingly negative pressure acting over a growing base area increases the drag. Unfortunately, no pressure data is provided to evaluate exactly how much the pressure distribution is affected by the blunting.

For a body of the form used by Stollery and Burns, the effect of ground clearance on drag is difficult to interpret since it is due to both changes in base pressure and separation on the curved underside towards the rear of the body. Fackrell (1975) carried out surface pressure and force measurements on a wedge shaped body with a flat underside, finding that generally there is a slight decrease in base pressure on the model with reduced ground clearance, resulting in an increasing drag. Fackrell’s plots of lift against ground clearance for incidences of -1, 0, and 1° are shown in Figure 2.12 together with a diagram of the model cross-section.

The lift reversal phenomenon only appears weakly at 0° incidence. As the ground clearance is reduced, the lift becomes more negative down to a minimum, but at
very low ground clearances, lift reversal occurs. For $\alpha = -1^\circ$ lift reversal does not appear over the range of incidences tested. These results would tend to confirm the suggestion of Zerihan and Zhang (2000) that lift reversal is the result of separation at the rear of the underbody, since trailing edge separation would be less likely to creep upstream with reduced ground clearance on the flat lower surface of Fackrells model. Fackrell also reports a downwards shift in the stagnation point on the nose at low ground clearance as noted by Bagley (1960) for his wing in ground effect. As the nose of his model is drooped the lowering of the stagnation point is unlikely to produce much positive lift on the nose, although it probably generates enough to cause the lift reversal at zero incidence. At $-1^\circ$ incidence, the lowering of the stagnation point will have less effect, and lift reversal does not seem to occur. Stollery and Burns
(1969) added drooped and raised noses to their symmetrical model, finding that lowering the nose increases longitudinal stability and reduces downforce. Raising the nose forces more flow beneath the model, increasing the flow velocity on the underside, reducing the pressure there and thus increasing downforce. However, as the stagnation point lowers, an increasing level of positive lift acts on the nose, resulting in an increase in nose-up pitching moment, making it less stable. Their lift-curve slopes for the model with the three noses are shown in Figure 2.13 for a ground clearance, h/c=0.3.

![Figure 2.13: Effect of nose shape on lift-curve slope for symmetrical low aspect ratio wing at h/c=0.3 (Stollery and Burns, 1969).](image)

From the viewpoint of car design, although the high nose increases downforce, it also increases the nose-up pitching moment. This produces a greater load on the rear axle than on the front, which would cause the car to understeer. Furthermore, $\partial C_M/\partial \alpha$ is positive, and greater for the raised nose. This is a potentially unstable situation since a sufficient increase in incidence could result in a nose up moment great enough to lift it further, possibly lifting the car off the ground as happened with disastrous consequences to the Mercedes CLR at Le Mans in 1999, as shown in Figure 2.14.
2.2.6 Automotive Diffusers

A diffuser is an aerodynamic device which produces a rise in pressure. For subsonic flows this is achieved by an increase in cross-sectional area which causes the flow to decelerate. Automotive diffusers (for example see Figure 1.1) have been used on racing cars for some years to augment the downforce generated by the car underbody, and in more recent years have become increasingly common on road cars to reduce drag and/or increase downforce (Howell, 1994).

An automotive diffuser exhausts flow to the rear of the car, where usually the flow is separated, and the local pressure is known as the base pressure. The pressure at the diffuser inlet is therefore reduced below the base pressure, which is relatively insensitive to diffuser angle (Cooper et al., 1998). This reduction in pressure on the diffuser ramp and upstream on the car underbody generates considerable levels of downforce. Although the basis for the automotive diffuser is based on diffuser theory, the mechanisms by which it generates the desired effects are more complicated due to inflow along its sides. However, to start with, the performance of a straight-walled two-dimensional diffuser will be considered as it illustrates several features common to automotive diffusers.

Diffuser performance has been the subject of a large number of investigations (Kline, Abbott and Fox, 1959; Sovran and Klomp, 1967; Reneau et al., 1967). The usual approach is to define some performance parameters as follows. A straight-walled two-dimensional diffuser may be characterised by any two of the following three geometrical parameters: Area ratio \( AR = \frac{W_2}{W_1} \), non-dimensional diffuser
length \((N/W_1)\), and the diffuser angle \((\theta)\) as defined in Figure 2.15.

\[
\text{Figure 2.15: Two-dimensional diffuser geometry.}
\]

The area ratio, \(AR\), being greater than unity results in a deceleration of the flow, and thus a rise in pressure. From this we define a pressure recovery parameter, \(C_p\), based on the mass-averaged velocity at inlet, \(\overline{U_1}\).

\[
\overline{C_p} = \frac{p_2 - p_1}{1/2\rho\overline{U_1}^2}
\]

From a simple one-dimensional analysis of the flow through the diffuser, an expression for the ideal (inviscid) pressure recovery parameter, \(C_{pi}\), may be derived using mass-conservation and Bernoulli’s equation.

\[
C_{pi} = 1 - \frac{1}{AR^2}
\]

This is used to define the effectiveness, \(\eta\), representing the proportion of the one-dimensional ideal pressure rise that is recovered by the diffuser.

\[
\eta = \frac{C_p}{C_{pi}}
\]

There are two reasons why the ideal pressure recovery is not realised in practice, resulting in a diffuser effectiveness below unity. Firstly, viscous losses in the duct reduce the total head of the flow, reducing the recoverable pressure gradient. Secondly, boundary layer growth through the duct, which is enhanced by the adverse pressure gradient, increases the displacement thickness along its length, reducing the effective area ratio. Sovran and Klomp (1967) examined the effects of viscous blockage as follows. Ratios between the effective \((A_E)\) and geometric \((A)\) areas,
$E$, and between the blocked ($A_B$) and geometric areas, $B$, are defined. Consider a cross-sectional area at some point along the diffuser, $A$, with a variable velocity normal to the cross-section, $u$, with a maximum, $U$. The effective area is that over which the constant peak velocity gives the same volume flow rate as for the variable velocity over the geometric area. This effective area is given by,

$$A_E = \frac{1}{U} \int_A u \, dA$$

The effective area ratio is defined as,

$$E = \frac{A_E}{A}$$

and the blocked area ratio as

$$B = 1 - E$$

Following a simple analysis, which will not be repeated here, Sovran and Klomp arrived at an equation for the diffuser effectiveness as follows. The subscripts 1 and 2 signify conditions at the diffuser entry and exit respectively.

$$\eta = \frac{1}{E_1^2} \left[ 1 - \frac{(E_1/E_2)^2}{1 - \frac{1}{AR^2}} \right] - \frac{\omega_m}{1 - \frac{1}{AR^2}}$$

where $\omega_m$ is the coefficient of total head loss between the points of maximum velocity at inlet and outlet, normalised by mass-averaged dynamic pressure at inlet.

This is an interesting result as it shows that if the diffuser has an inviscid core, then the diffuser effectiveness depends entirely on boundary layer growth. Even if this is not the case, owing to the generally large adverse pressure gradients in diffusers, the total head loss term is normally much smaller than the blockage term. In other words, as Sovran and Klomp put it, the loss in diffuser effectiveness is the result of insufficient rather than inefficient diffusion. This is why $\eta$ is called the diffuser effectiveness rather than efficiency: so long as there is a boundary layer at inlet, an inviscid analysis yields a value less than one even though there is no loss in energy.

Since the viscous term is small, the majority of the boundary layer growth is due to adverse pressure gradient effects, rather than viscous effects. Sovran and
Klomp analysed this on the basis that the blockage at exit should be a function of the blockage at inlet and the pressure gradient. The pressure gradient is in fact a function of the area ratio and boundary layer growth. They found a semi-empirical relationship between area ratio, blockage at inlet, and effective area at outlet for a conical diffuser, which fits that for a 2-D diffuser rather well, as shown in Figure 2.16, for near-optimal diffusers (i.e. where the viscous loss term is small). $C_p^*$ is defined at the maximum possible pressure coefficient for fixed non-dimensional length, as described in greater detail further on in this discussion. In this figure, they use $AR$ for the area ratio, $B_1$ for the blockage ratio at the diffuser inlet, $L$ for the diffuser length, and $W_1$ for the diffuser height (proportional to the cross-sectional area here) at inlet.

![Figure 2.16: Relationship between area ratio and viscous blockage at inlet, and effective area at outlet (Sovran and Klomp, 1967).](image)

Kline et al. (1959) defined a number of flow regimes in two-dimensional straight-walled diffuser flow. At low area ratios, they describe ‘no appreciable stall’ in the diffuser, with no large areas of flow separation. With increasing area ratio, large areas of separated flow appear in the diffuser and get “washed out” of it by the flow through the diffuser. This is highly unsteady, with large pressure fluctuations, and designated as the ‘large transitory stall regime’. They found that increased turbulence at the inlet increases the area ratios at which the transition to this
regime occurs. Beyond this, in the 'two-dimensional stall regime' the flow separates near the throat and remains separated along one entire wall, but following the other wall. At very high area ratios, the flow is fully separated from both walls, and the 'jet flow regime' exists. A plot of these regimes in the diffuser length-diffuser angle plane are shown in Figure 2.17 (Reneau et al., 1967). In Figures 2.17, 2.18, and 2.19, \( \phi \) is the diffuser half-angle, for which \( \theta \) is used everywhere else in this thesis.

![Figure 2.17: Diffuser flow regimes (Reneau et al., 1967).](image)

Reneau et al. correlated these regimes against diffuser pressure recovery, finding that the largest pressure recovery occurs in the large transitory stall regime. The maximum effectiveness, however, occurs in the regime of no appreciable stall, at a lower area ratio (or non-dimensional length). The optimum design therefore depends very much on the diffuser requirements. For automotive design, this will be more related to the maximum pressure recovery since the condition of the flow exiting the diffuser is not very important.
Reneau et al. used contour plots of pressure recovery parameter with axes of non-dimensional diffuser length, $N/W_1$, and area ratio, $AR-1$. These are considered to be the best parameters defining a diffuser, since the area ratio defines the ideal pressure recovery, and the non-dimensional length defines the ideal pressure gradient for a fixed area ratio. The diffuser angle, $\phi$, appears as a dependant variable in these plots as diagonal lines of constant gradient. Sovran and Klomp (1967) plotted diffuser performance data on the same axes, together with loci of points defining maximum pressure recovery for fixed non-dimensional length, $C_p^*$ and for fixed area ratio ($C_p^{**}$) as shown in figure 2.18. Both are straight lines, and the locus of points defining ($C_p^{**}$) is at an approximately constant diffuser half-angle of 3.5°, a well-known result for two-dimensional diffusers.

![Figure 2.18: Typical diffuser performance map for fixed boundary layer thickness at inlet (Sovran and Klomp, 1967).](image)

An automotive diffuser differs from a two-dimensional diffuser in that, in the absence of skirts which seal the car sides to the ground, flow is able to leak in or out
of it at the sides. This has a number of consequences for the diffuser flow behaviour. Automotive diffusers do, however, tend to have endplates at either end, but these do not generally extend below the level of the underbody at the diffuser inlet. The geometry of an automotive diffuser is defined in Figure 2.19.

Since the ground clearance of the body, $h$ is not fixed, it is better to define the geometry in terms of the diffuser ramp angle, $\theta$, and the diffuser length, $L$. The diffuser area ratio is, in terms of this fixed geometry and the body ground clearance,

$$AR - 1 = \frac{L \tan \theta}{h}$$

At large ground clearances, where $h \ll L$, $AR$ tends to infinity, and the rear of the underbody can no longer be considered as a diffuser since $\overline{C_{pi}} \to 0$. However, the diffuser angle adds a negative camber to the body, which can therefore still be expected to generate some downforce. Cooper et al. (1998) looked in some detail at automotive diffuser performance, defining three mechanisms of downforce generation; upsweep, which is the negative camber introduced by the diffuser; ground effect, the downforce generated by any body in close proximity to the ground; and diffuser pumping, which is the downforce resulting from the low pressure upstream generated by the pressure drop along it relative to the base pressure (the pressure in the separated wake of the body). Their experimental results for varying diffuser angles at different ground clearances provide an excellent illustration of how these
three effects contribute to the overall downforce of a diffuser-equipped bluff body, as shown in Figure 2.20.

Figure 2.20: Components of downforce as identified by Cooper et al. (1998).

Since the pressure drop across an automotive diffuser is difficult to define owing to three-dimensional effects, they plotted contours of downforce instead of pressure recovery parameter. These were plotted on the same axes of area ratio against non-dimensional diffuser length used by Reneau et al. (1967), an example of which is shown in Figure 2.21. They found that the contours are closed, a result only found for very thick boundary layers at diffuser entrance ($2\delta^*/W_1 \geq 0.05$) by Reneau et al. However, for the automotive diffuser this is due to a combination of boundary layer thickness at the inlet, which may grow excessively if the diffuser is too long, and the leakage of the flow underneath the endplates. This leakage results in a non-constant mass flow, which rises along the diffuser length. For a very long diffuser with sufficient leakage at the sides, it would be possible to have a constant velocity through it, and thus zero pressure drop. However, they did not explicitly examine this effect.
Investigations of the three-dimensional flow structures in a diffuser without endplates by George (1981) showed that separation of the flow leaking from the diffuser sides produces streamwise vortices similar to those forming either side of the sloping backlight of hatchback cars studied by Morel (1978), as shown in Figure 2.22. These vortices have two important effects. The first is to help maintain attached flow on the sloping base, improving the pressure recovery, increasing the base pressure and reducing the drag. Secondly the low pressure associated with the core applies a force normal to the sloping surface which, depending on the angle of slope, generates varying amounts of lift and drag.

George and Donis (1983) compared automotive diffuser models with sides sealed to the ground to those with open endplates, finding that for greater diffuser angles their models with open endplates produce greater downforce because of the vortices maintaining attached flow on the diffuser ramp, and directly inducing low pressures along their path under it.

Senior and Zhang (2001) examined the three-dimensional diffuser flow mechanisms affecting diffuser performance for fixed diffuser angle over a range of ground
clearances. They identified four regimes of diffuser flow which govern the downforce generated by the body. At large ground clearances, as the ground is approached, the downforce rises. The diffuser flow is symmetric about the centreline, with streamwise vortices forming from the separation of flow from the diffuser endplates either side. These vortices generate a considerable drop in pressure near the endplates. The associated low pressure peak in the spanwise pressure distributions moves inboard and disappears towards the diffuser exit as the vortices move inboard and separate from the diffuser ramp. The rate of increase of downforce with reduced ground clearance then arrests as a symmetrical separation bubble appears on the diffuser ramp. The separation line moves upstream with reduced ground clearance. The maximum downforce occurs in this regime. With a further increase in ground proximity the vortex on one side of the diffuser collapses, and the separation line moves up to the diffuser inlet. This corresponds to an abrupt drop in downforce. Finally, at very low ground clearances the downforce maintains a low level with less dependence on ground clearances. They suggested that this flow regime is characterised by a merging of boundary layers on the diffuser ramp and ground plane.
2.3 Vortical Flows

The flow over a Formula 1 car, as with other open-wheel racing cars, is heavily dominated by vortices from the various wings, winglets and barge-boards. The interaction between these components is nonlinear and complicated (Katz and Garcia, 2002) and these vortices are likely to play an important role in this. The structure of a vortex is therefore an important consideration for the interaction mechanism and will be addressed in the following section.

2.3.1 The Structure of an Isolated Vortex

A vortex may be defined as a quasi-cylindrical axisymmetric swirling flow. Viscosity in the core results in the flow rotating as a solid body at the centre, whilst far from the core the flow is essentially irrotational. The simplest model of the tangential velocity distribution associated with a vortex is the so-called Rankine vortex (Saffman, 1995) which assumes the solid body rotation within the core ($r < r_1$), and irrotational motion outside it:

\[ r < r_1 \]

\[ u_\theta = \frac{\Gamma_0 r}{2\pi r_1^2} \]

\[ r > r_1 \]

\[ u_\theta = \frac{\Gamma_0}{2\pi r} \]

Lamb (1932) showed that the Navier-Stokes equations may be re-written for the two-dimensional incompressible circular flow about a point in cylindrical coordinates in terms of the vorticity, $\omega$ as follows

\[ \frac{\partial \omega}{\partial t} = \nu \left( \frac{\partial^2 \omega}{\partial r^2} + \frac{1}{r} \frac{\partial \omega}{\partial r} \right) \]

which is identical to the equation for the radial conduction of heat in two-dimensions. The solution to this, starting at a time $t = 0$ with an instantaneous vortex source of strength $\Gamma_0$ at $r = 0$ is given as follows, with the thermal analogy of an instantaneous heat source at $t = 0$ in an infinite medium.

\[ \omega(r, t) = \frac{\Gamma_0}{4\pi \nu t} e^{-\frac{r^2}{4\nu t}} \]
This is known as the Lamb-Oseen vortex, and since it is a result of the Navier-Stokes equations is a more physical solution than the Rankine vortex. In the same way that heat is conserved, the total circulation of the system is also conserved. A comparison of the vorticity, circulation and vorticity fields for a Rankine and a Lamb-Oseen vortex is shown in Figure 2.23. Note that at $r = r_1$ the circulation for the Lamb-Oseen vortex has not yet reached its maximum, but $\Gamma/r$ (= $u_\theta$ for an axisymmetric vortex) is at a maximum.

![Figure 2.23: Comparison of the circulation about, and tangential velocity in a Rankine and a Lamb-Oseen vortex.](image)

The velocity distribution in a real vortex, however, is more complicated than this, depending on a number of factors such as the distribution of vorticity in the shear layer which rolls up into the vortex (Moore and Saffman, 1973), but for the purposes of the present investigation, the basic form of the velocity distribution as given by the Lamb-Oseen vortex is sufficient.

The pressure field in a vortex is governed by the need to balance the centripetal acceleration of the flow about the vortex centre. A simple consideration of the
required forces results in the following equation for the pressure, $p$, at any radius, $r$, from the centre of a circular vortex, with tangential velocity field $u_\theta(r)$, relative to the static pressure far from the vortex centre, $p_\infty$.

$$\frac{p_\infty - p(r)}{\rho} = \int_r^\infty \frac{u_\theta^2}{r} dr$$

This results in a low pressure at the vortex centre to sustain circulatory motion about it. Earnshaw (1961) made static and total pressure measurements in a delta-wing vortex, and noted extremely low pressures at the vortex centre, with minimum pressure corresponding to a pressure coefficient below -6, as his plots, displayed here in Figure 2.24, show. He also recorded very low total pressure coefficients of almost -1.0. Despite this low total head, the very low static at the vortex centre is indicative of a raised axial velocity, peaking at over double the free-stream velocity.

![Figure 2.24: Static and total pressure distributions through a delta wing vortex core at various locations downstream of the apex Earnshaw (1961).](image)

Batchelor (1964) showed that this distribution of static pressure through the vortex results in the necessity for an axial velocity distribution. He considered Bernoulli's equation along a streamline from far upstream of the wing, where the
velocity is $U_\infty$, to a point in the vortex. Since this streamline may pass through the boundary layer, and there is viscous action in the vortex core, there may be some loss in total head, accounted for by $\Delta H$. Writing $C = ru_\theta$, the following equation results for the axial velocity, $u_z$ at any point in the vortex, $r$.

$$u_z = U_\infty + \int_r^\infty \frac{1}{r^2} \frac{\partial C^2}{\partial r} \, dr - 2\Delta H$$

Neglecting the total head loss, since in the core $\frac{\partial C^2}{\partial r}$ is greater than zero, the axial velocity everywhere in the core is greater than the freestream velocity. This effect is most commonly found in delta wing vortices, which tend to have highly condensed cores with less total head deficit due to interaction of the streamline with the boundary layer. Conventional unswept wings tend to have a greater total head loss term due to interaction of the streamlines with the boundary layer, which tends to result in an axial velocity deficit (for example, the results of Devenport, Rife, Liapis and Follin (1996)).

The diffusive nature of the core results in a slow decay in the tangential velocity, such that $\frac{\partial C^2}{\partial r}$ decays. This results in a pressure which increases along the core with streamwise distance (Batchelor, 1964) that will tend to decelerate the axial flow at the vortex centre.

Devenport et al. (1996) measured the flow in a trailing vortex, with efforts to take account of the 'vortex wandering' phenomenon. This is a process by which the location of the vortex at a given distance downstream of the wing fluctuates with time, and is not fully understood. Their measurements suggested that the vortex core is laminar, but moves by approximately 30% of the core radius. They proposed that the motion is due to buffeting by the surrounding turbulent flow. If this is not accounted for when taking time-averaged measurements then it can have a significant affect on the results, increasing the measured core radius and decreasing the measured peak tangential velocity. Vortices are also highly sensitive to intrusive measurement probes (Green, 1995) which together with the wandering makes time-averaged probe-taken data unreliable. This problem has been overcome in recent years with the introduction of non-intrusive techniques such as Particle Image Velocimetry (PIV).
2.3.2 Vortex Breakdown

2.3.2.1 Description of Vortex Breakdown

Vortex breakdown involves a brutal and abrupt disorganisation of a vortex core, such that, for the majority of practical flows, its structure is completely destroyed. Vortex breakdown was first observed by Peckham and Atkinson (1957) who noticed through flow visualisation that one of the vortices over their gothic wing appeared to abruptly 'bell-out' at some point, downstream of which the core appeared to lose its coherent structure. Following this, a large amount of work has been conducted on vortex breakdown, as its consequences are important for a number of engineering applications. In particular, vortices over delta wings provide a large amount of the lift at high angles of attack. When one of the vortices breaks down, it can lead to a large and abrupt loss of lift and control (Althaus, Bruker and Weimer, 1995). An example of a vortex breakdown flow visualisation over the wing of an F-18 aircraft is shown in Figure 2.25.

![Figure 2.25: Smoke flow visualisation of vortex breakdown over an F-18 aircraft wing (NASA Dryden Flight Research Centre.](image)

As previously mentioned, the wakes of the front wing and barge boards are likely to be dominated by trailing vortices, which may pass underneath the car,
forming an important part of the interaction process. The persistence or possible breakdown of these trailing vortices is therefore likely to have a strong influence in the interaction mechanisms. Vortex breakdown is a widely studied field that still lacks full understanding, and a comprehensive review of the current level of understanding will not be given here. However, some discussion of the modes of vortex breakdown, and factors influencing breakdown would be beneficial.

2.3.2.2 Modes of Vortex Breakdown

A number of studies have been conducted on a vortex in the so-called vortex tube. This consists of a long tube through which air is drawn, upstream of which are a series of vanes to impart swirl on the flow, generating a vortex which convects down the tube whose strength may be altered by modification of the vane angles. By injecting dye into the centre of the vortex, the core may be visualised. Harvey (1962) noted a form of breakdown characterised by a hemi-spherical bubble headed by a free stagnation point, which remained virtually stationary in the tube. Later, Sarpkaya conducted a series of similar tests in a diverging vortex tube, so as to impose an adverse pressure gradient on the flow (Sarpkaya, 1971a; Sarpkaya, 1971b; Sarpkaya, 1974). He found three modes of vortex breakdown: the ‘bubble’ mode, which is of the same form as that found by Harvey (1962); the ‘spiral’ mode; and the ‘double helix’ mode. His flow-visualisations of these modes are shown here in Figure 2.26.

The double-helix mode of breakdown was only found to exist at low Reynolds numbers, and is extremely unlikely to occur outside the controlled environment of the vortex tube. We will therefore only consider the bubble and spiral types of breakdown, since these are the only two forms that are likely to occur in the flows that are the topic of this thesis. It would seem from the results of Sarpkaya (1974) that the axisymmetric bubble type of breakdown is more likely at higher swirl angles than the non-axisymmetric spiral type. However, there is a range of swirl levels over which either form can occur (Leibovich, 1978) with the flow switching from one mode to the other with no apparent change in conditions.

For the spiral form of breakdown, a kink forms abruptly in its path, downstream
Figure 2.26: The three types of vortex breakdown of Sarpkaya (1971a): Double-Helix (top); Spiral (middle) and Bubble (bottom).

of which it spirals about its original axis before eventually being broken up by large scale turbulence. Hall (1972) made sketches based on the results of Sarpkaya (1971a) of the development of vortex breakdown from the spiral to the bubble type of breakdown with increasing swirl angle, as shown in Figure 2.27.

As the swirl angle is increased, the spiral becomes more 'compact' with a greater winding angle, with progressively greater distortion of the vortex filament, which Hall (1972) stated has a tendency to turn back towards the kink. Ultimately, at large swirl angles, the spiral structure completely disappears, leaving the bubble form of breakdown. The bubble-type breakdown consists of a single vortex ring which gyrates around the centreline, resulting in a strong periodicity of the flow field, with a slight asymmetry in the bubble shape for which the stagnation point rotates around the centreline (Althaus et al., 1995). There is some question over how much the spiral and bubble type breakdown modes differ, with some arguing that the bubble type is a compressed spiral (Lucca-Negro and O'Doherty, 2001), and
in many respects this is still not a fully resolved issue. However, for the purposes of this thesis it is only important that vortex breakdown is recognised qualitatively, and that its aggravating factors are identified, as discussed in the following section.

2.3.2.3 Criteria for Vortex Breakdown

Since its discovery, there have been a large number of theories which have emerged in the attempt to explain vortex breakdown. One of the first theories on the subject was that of Squire (1960) who analysed the conditions necessary for long standing waves to be sustained in the flow field, arguing that this would allow disturbances to be propagated upstream from some downstream source. He found that the swirl ratio, \( \tau = \frac{u_\theta}{u_x} \) lay between 1.00 and 1.20, depending on the form of the tangential velocity profile. This corresponds to swirl angles, \( \gamma = \tan^{-1} \tau \), between 45° and
50.2°. Flow visualisations of Harvey (1962) found the swirl angle just upstream of breakdown to be between 50.2° and 51° which is in remarkably good agreement with Squire’s theory.

Brooke Benjamin (1962) formulated a theory along similar lines to Squire, interpreting breakdown as the transition between two conjugate states of swirling flows, using the analogy of the hydraulic jump in channel flow. He found the conditions for the two states to exist: ‘supercritical’, where standing waves are not possible in the flow; and ‘subcritical’ where they are. The boundary between these two states is the ‘critical’ state, where it was proposed that breakdown would be initiated. He shows that increasing the swirl increases the likelihood that the flow will make the jump to the subcritical state. Experiments have shown that the flow just upstream of vortex breakdown is always supercritical, and downstream is always subcritical (Leibovich, 1978), which tends to support this theory.

Some other theories of vortex breakdown follow the analogy of boundary layer separation by applying quasi-cylindrical approximations similar to the boundary layer approximations, in which changes in the axial direction are slower than in the transverse (radial) direction. Solving the quasi-cylindrical equations of motion using an x-marching technique, the solution diverges where vortex breakdown is likely to occur. Delery (1994) used the example of a vortex of fixed swirl in an adverse pressure gradient, for which solution divergence ultimately occurred. This is analogous to Goldstein’s singularity, for the divergence of the solution to the boundary layer equations near the point where the boundary layer separates. This is in agreement with the experiments of Sarpkaya (1974), which showed that vortex breakdown can occur at lower swirl angles for increased adverse pressure gradients. This is not surprising since arguments similar to those used by Batchelor (1964) can be used to show that the pressure along the axis of a vortex is amplified in accordance with the following equation (Delery, 1994).

\[
\left(\frac{\partial p}{\partial x}\right)_{(axis)} - \left(\frac{\partial p}{\partial x}\right)_{r_0} \sim \alpha \rho \Gamma^2_0 \frac{r_0^3}{c^3}
\]

Delery reviews a number of different experiments and theories, finding that all have a critical swirl parameter, \(S^*\) all roughly equal to 1.37, which equates to \(\gamma \approx\)
50°. The swirl parameter, $S$, is defined as

$$S = \frac{\Gamma_0}{r_c V_{x(z)}}$$

Flows with $S \leq S^*$ are supercritical, and its development is slow and steady. In an adverse pressure gradient, the axial velocity is decelerated until $S$ reaches $S^*$ at which point breakdown occurs. He also pointed out that experiments in which the axial velocity profile could be modified, increasing the axial velocity reduces the susceptibility of the vortex to breakdown, and vice versa.

In all there is no simple way of determining breakdown, but reduced axial velocity at the core, increased swirl and adverse pressure gradients are all factors which increase the likelihood of breakdown in a vortex.

2.4 Interaction Effects

The present study is of interaction mechanisms relevant to Formula 1 cars, in particular, the effects of upstream aerodynamic devices such as the front wing and bargeboards on the underbody and diffuser.

These interactions involve a number of rather complex non-linear aerodynamic effects, comprising the low energy, turbulent and vortical components of the upstream body wakes. To build an understanding of the major considerations of the various interaction mechanisms, it is of benefit to begin with a review of some of the more basic known aerodynamic interaction effects. Starting with the simplest two-dimensional inviscid and viscous interaction effects between lifting and non-lifting bodies, our discussion will move on to more complex vortical interactions, finishing with a review of those effects more pertinent to open-wheeled racing cars, such as a Formula One car.

2.4.1 Basic Interaction Effects

The flow in the wake of a body has a momentum deficit due to viscous losses, resulting in a reduction in total head. This has two consequences for a downstream body: a reduction in stagnation pressure, which may reduce drag; and a reduction
in the flow velocity, increasing the pressure, for example, on its flat lower surface in the case of a Formula One car. Biermann and Herrnstein (1933) studied interference effects between streamline struts. With the struts in tandem (one behind the other) they found that as the separation is reduced, the drag on the forward strut is lowered, whilst the drag of the rear strut rises. The high pressure on the rear strut raises the pressure at the rear of the forward strut, reducing its drag. Meanwhile, earlier separation on the rear strut increases its drag, despite there being a lower stagnation pressure (Hoerner, 1958).

For bluff bodies, the effect is quite different. For two bluff bodies in tandem, the effect of reduced separation on the front body is negligible, whilst the drag on the rear strut is greatly reduced, and can even result in a net upwind force on the rear body. This is because the recirculation in the wake of the front body produces a lower stagnation pressure (if in fact the flow stagnates at all) on the rear body upstream face. Meanwhile, the base pressure of the front body is largely unaffected and the drag unchanged. This effect is far greater than any change in separation point, and so the drag is reduced. These effects are summed up in Figure 2.28, taken from Hoerner (1958).

Clearly, the overall effect is highly dependant on the geometry involved. Wind tunnel measurements on NASCAR racing saloon car models in tandem made by Romberg, Chianese, Jr. and Lajoie (1971) showed that at a separation of one car length, the drag on the rear car is reduced by 37%. Meanwhile the drag on the forward car is only reduced at very low separations (less than $1/10^{th}$ of a car length), with a reduction of 30% at separation distances near zero. For the rear car this can be highly beneficial on straight sections of track. Romberg et al. quote a reduction in power of 133 HP required to maintain a speed of 190mph. However, a Formula One car relies on downforce for cornering, and the detrimental effect of an upstream car on downforce can lead to losses in grip, making overtaking more difficult.

Simple inviscid analysis of two-dimensional positive lifting wings in tandem shows that the upstream wing generates a downwash at the trailing wing, reducing its lift for fixed incidence, whilst the downstream wing generates an upwash at the upstream wing, increasing its lift (Katz, 1995).
2.4.2 Vortex Interactions

2.4.2.1 Behaviour of multiple vortex systems

As we have seen in the previous discussion, the wake of a Formula 1 front wing is dominated by a number of vortices, and so it is necessary to consider how these vortices interact.

At the simplest level, we can consider how a number of point vortices move under the velocities induced between them. Batchelor (1970), by taking moments of vorticity, showed that a group of vortices of strength $\Gamma_i$ at positions $(x_i, y_i)$ will orbit a point at position $(X, Y)$, defined as

$$X \sum_i \Gamma_i = \sum_i x_i \Gamma_i, \quad Y \sum_i \Gamma_i = \sum_i y_i \Gamma_i$$

with a velocity of each vortex calculated as the sum of the induced velocities from
all of the other vortices:

\[
\frac{dx_{ij}}{dt} = -\frac{1}{2\pi} \sum_{i \neq j} \frac{\Gamma_i (y_j - y_i)}{r_{ij}^2}, \quad \frac{dy_{ij}}{dt} = \frac{1}{2\pi} \sum_{i \neq j} \frac{\Gamma_i (x_j - x_i)}{r_{ij}^2}
\]

where \(r_{ij}\) is the separation between the centres of vortices \(i\) and \(j\). For the simple case of a single vortex pair separated by a distance between their centres, \(d\), the angular velocity of the pair about the centre of vorticity is

\[
\omega_{\text{pair}} = \frac{\Gamma_1 + \Gamma_2}{2\pi d^2}
\]

The results of these equations are that two vortices of equal sign and strength will rotate about a point midway between the two. If one vortex strength is greater then the centre of rotation is shifted towards that vortex. If the vortices are of opposite sign and equal strength, then they will not rotate about one another, but will simply translate along a path perpendicular to the line joining the vortex centres.

For a single pair of three-dimensional vortices in otherwise irrotational flow, for large separation distances, their paths are governed well by this irrotational point vortex theory (Groenenboom and levalts, 1977; Cerretelli and Williamson, 2003). However, for small separations ‘vortex merging’ may occur for either like signed vortices, or opposite signed vortices of unequal strength (Groenenboom and levalts, 1977), in which the vortices rapidly merge together after some orbiting of one-another. Experimental vorticity plots of two co-rotating (anti-clockwise) vortices undergoing merging are shown for four points in time in Figure 2.29, illustrating the key stages of the merging process (Cerretelli and Williamson, 2003).

Bertényi and Graham (2000) showed that the distance between production and merging of the vortices is dependant on the initial vortex separation, relative vortex strength and core size, finding that the merging process is rapid compared with their previous orbiting. For example, for two particular vortices of nominally equal strength, merging occurred 98 chord lengths downstream of the generating wing, but was completed within one chord length. They found that for unequal vortices the vorticity from the weaker vortex bleeds into the stronger vortex, and this occurs earlier with greater difference in strength, although the merging process lasts longer.
An excellent account of the physics of the merging process is given by Cerretelli and Williamson (2003) who described four stages to the merging process. In the first, viscous (or turbulent) diffusion causes the vortices to grow in size, with a fixed distance separating their centres, such that their vorticity fields begin to overlap. Following this, the induced velocity field between the two causes spiral vortex filaments to appear, which Smits and Kummer (1985) described as giving the appearance of 'spiral galaxies'. These spiral filaments, which Cerretelli and Williamson described as the antisymmetric components of vorticity 'push' the vortex cores together. Finally further viscous and turbulent diffusion results in a merging of the two vortices into a single continuous vortex core.

2.4.2.2 Interaction between a vortex and a plane boundary

The interaction between vortices and the ground plane has been studied widely in the context of aircraft trailing vortices. Lamb (1932) considered the motion of a
vortex pair of equal and opposite strength, oriented such that they descend towards the ground plane. He showed that they follow a hyperbolic path, diverging as the ground plane is approached, governed by the following equation:

\[ b_0^2(h^2 + y^2) = h^2y^2 \]

where \( h \) is the (variable) ground clearance of the vortices, \( y \) is half of the lateral separation, and \( b_0 \) is half of the initial lateral separation of the vortices far from the ground, and the height which the vortices asymptotically approach.

It is a well known fact (Barker and Crow, 1977) that in reality, a descending vortex pair will follow this path down to a certain ground clearance, below which they 'rebound' and rise again. Harvey and Perry (1971) provided evidence of a separation of the cross-flow boundary layer beneath the vortices, into a secondary vortex of opposite sign to the primary vortex, as shown in Figure 2.30.

![Figure 2.30: Separation of the cross-flow boundary layer on the ground plane beneath a vortex to produce a secondary vortex of opposite sign (Harvey and Perry, 1971).](image)

Barker and Crow (1977) carried out experiments on vortex pairs approaching both a fixed and a free surface (where the shear stress vanishes) in water. They calculated the circulation of the vortex from experimental vortex trajectories using the following equation

\[ \Gamma = 4\pi q \frac{yh(y^2 + h^2)}{\sqrt{y^6 + h^6}} \]

where \( q \) is the vortex propagation speed, \( q = \sqrt{\dot{y}^2 + \dot{h}^2} \). They found that the circulation calculated from this method resulted in a series of oscillations which was
repeatable in both amplitude and frequency. These findings were the same for both the free and fixed surfaces. This suggested that there was an extra mechanism at work, which they thought to be an inviscid phenomenon due to the finite vortex core size.

Two-dimensional viscous Navier-Stokes calculations of a vortex in ground effect were performed by Peace and Riley (1983) using no-slip and slip-free boundaries to simulate the ground plane. With the no-slip boundary, secondary vorticity was created, but did not separate from the surface. However, with the slip-free boundary (synonymous with the free-surface of Barker and Crow (1977)), clearly no secondary vorticity is created. Instead, the surface acts as a vorticity sink, lowering the vorticity in the finite sized core nearest the boundary, moving the position of peak vorticity upwards, and so giving the impression of rebound. However, Liu, Hwang and Srnsky (1992) point out that it has been demonstrated that only extremely clean surfaces can fulfill the free-surface condition, and surface contamination can cause the free-surface to act as a solid one during the interaction with a vortex. This is a more likely explanation of the findings of Barker and Crow, but the mechanism does not seem to be fully understood.

Liu et al. (1992) demonstrated that by considering the influence of a secondary vortex on the primary vortex, the rebound phenomenon can be simulated. The secondary vortex can, depending on its strength relative to the primary vortex, orbit about the primary, resulting in a periodic motion, which could explain the repeatable oscillations in the calculated circulation of Barker and Crow. Later two-dimensional viscous calculations (Zheng and Ash, 1996; Türk et al., 1999; Puel and de Saint Victor, 2000) all demonstrate that at greater Reynolds numbers \(= \Gamma_0/\nu \) for the primary vortex than those of Liu et al. secondary vorticity separates into secondary vortices, which orbit around the primary vortex, causing it to loop, or spiral in three-dimensions. Further vortices may also form from the separation of the secondary vorticity, as shown in the iso-vorticity contours of Puel and de Saint Victor in Figure 2.31 at three points in time of the primary clockwise vortex \(T_1\) with secondary anti-clockwise vortices \(S_2\), \(S_3\) and \(S_4\).

The computations at greater Reynolds numbers by Türk et al. (1999) show an
even more complex flow pattern as the Reynolds number is increased, as shown in their iso-vorticity contours of Figure 2.32. They found that with increasing Reynolds number less vorticity is lifted from the ground, but that vorticity is raised higher by the primary vortex. At the lowest Reynolds numbers, the primary and secondary vortices touched and partially eliminated each other, resulting in a faster decay of the primary vortex.

In summary, it is known that a vortex pair approaching a solid boundary will rebound owing to the production of secondary vortices from separation of the cross-
flow boundary layer beneath the primary vortices. This can result in a number of secondary vortices which then orbit the primary vortex, causing it to convect in a spiral path downstream. Since the production of secondary vorticity is directly governed by the proximity of the primary vortex, which in turn is governed by the location of the secondary vortices, the interaction effects are extremely complex. Prediction of the vortex path is therefore very difficult without solving the Navier-Stokes equations.

Finally, it is also known that a turbulent boundary layer is more resistant to separation than a laminar boundary layer, which delays the production of secondary vortices and thus delays rebound (Puel and de Saint Victor, 2000). The state of the cross-flow boundary layer is therefore an important consideration. This is also likely to be strongly affected by the velocity of the ground plane relative to the freestream, although no literature could be found in which the cases for moving and stationary boundaries were compared directly.

A similar process occurs when a vortex pair interact with a stationary (with respect to the model in a wind tunnel over which the air flows) plane (Cutler and Bradshaw, 1993a; Cutler and Bradshaw, 1993b). Flow divergence between the descending vortex pair causes a local thinning of the boundary layer, with convergent flow on the up-flow sides of the vortices causing boundary layer thickening. At low clearances between the plane and the vortices, this results in a three-dimensional separation line, from which low momentum fluid is lifted from the surface and entrained into the primary vortex, resulting in a gradual decline in its circulation. All measurements were time-averaged, and the lifting of discrete secondary vortices was not detected.

Experiments on wings similar to a Formula 1 car front wing show that at low ground clearances, the main vortex shed from the lower edge of the endplate (see Figure 2.11) tends to burst. Zhang and Zerihan (2003) showed that, for their arrangement, the vortex grows in strength over reduced ground clearances down to that for maximum downforce, below which the vortex bursts. More detailed PIV measurements taken by Moseley (1999) in vertical planes perpendicular and parallel to the freestream showed vortex breakdown to occur at low ground clearances in
either the spiral or bubble mode, with spiral winding angles anywhere between $45^\circ$ (spiral mode) and $90^\circ$ (bubble mode). Comparing these angles to the convection velocity ($6.66\,m/s$), he showed this to correspond to a rotational frequency $= V_\theta / 2\pi r$ of between 111 and $167\,Hz$. Normalising this as $f^* = f_c / u_{\infty}$ gave values between 2.22 and 3.3. Comparing these values to those of other researchers, he found that his mode of breakdown involved a more tightly wound spiral (greater winding angle and rotational frequency) than for vortex breakdown far from a ground plane. He therefore suggested that the ground plane has a strong effect on the breakdown process, and resulting flow structure.

2.4.2.3 Interaction between a vortex and a wing

The interaction between a trailing vortex and a wing involves a number of effects which are relevant to the topic of study being discussed in this thesis. Vortex interactions with wings are present in a large number of practical engineering applications including blade-vortex interactions of helicopter rotors, propeller vortex-wing interactions on aircraft, canard-wing aircraft configurations, and straked wing designs. Much of the blade-vortex interaction problem involves complex three-dimensional (with the vortex either aligned perpendicular, parallel, or at an angle to the blade axis) and unsteady effects which are not directly related to the current problem. Therefore the only studies which are considered are those addressing problems where the vortex is aligned with the free-stream and passes over a rigid wing.

There are a number of key effects which are addressed in these studies which may be classed into four categories:

1. Locally induced changes in incidence.
2. Locally induced pressure variations.
4. Influence of the wing on the vortex structure.

Early studies involved inviscid considerations of the locally induced velocity flow field. Smith and Lazzeroni (1960) considered local changes in incidence along the
span of the wing resulting from the vertical component of velocity induced by the vortex. Good agreement with experimental results were achieved by calculating the effects of the vertical component of velocity induced by an inviscid vortex with finite core size (to avoid infinite velocities at the centre) on the spanwise incidence and thus lift distribution. This behaviour is in agreement with flow visualisations showing stall to be promoted on one side of the vortex and delayed on the other due to the corresponding increase and decrease in effective incidence either side of it (Patel and Hancock, 1974; McAlister and Tung, 1984). Local spanwise changes in incidence are therefore one of the most important influences, and this is a largely inviscid phenomenon.

It is known that a vortex passing close to a surface induces a low pressure, which is often implemented to augment the lift generated by, for example, a delta wing (Polhamus, 1971). Passing a trailing vortex close to a wing surface may therefore be expected to induce a low pressure locally. This was found to be the case by Patel and Hancock (1974), as shown in the Figure 2.33 showing a schematic diagram of their surface flow visualisation and corresponding spanwise pressure distributions. The vortex location is shown by the herringbone pattern, where low surface pressures were measured at locations 2, 3 and 4. At the most upstream location, a large low pressure peak is created by the local increase in incidence induced by the vortex. The strength of this low pressure track created by the vortex increases as it is brought closer to the wing surface (Mahalingam, Funk and Komerath, 1997).

The tangential component of velocity induced by the vortex at the wing surface introduces a strong three-dimensionality in the boundary layer, as shown in the sketched surface flow visualisations of Patel and Hancock in Figure 2.33. This illustrates the herringbone streaklines at the surface, with a three-dimensional separation line clearly visible on the up-going (left in the figure) side of the vortex. This is due to the separation of the boundary layer into a secondary vortex, as discussed for vortex-boundary interactions in the previous section. At very close proximity of the vortex to the wing, this secondary vortex is strong enough to cause a low pressure track of its own. Similar behaviour was found in the flow visualisations of Mehta and Lim (1984) who suggested cross-flow topologies for reducing vortex-
Figure 2.33: Surface flow pattern underneath vortex path on wing surface (top) and spanwise pressure distributions at a number of chordwise locations (Patel and Hancock, 1974).

wing surface clearance up to the generation of a single secondary vortex as shown in Figure 2.34.

The comprehensive flow visualisations of McAlister and Tung (1984) showed that the vortex may be rendered unstable by its interaction with the wing. For interactions where the vortex impinges on the leading edge of the wing, a spiral mode of breakdown was reported some small distance upstream of the leading edge, as also documented by Patel and Hancock. They also noted a spiral instability in the vortex as it passed close to the separated wake of the wing at high incidences, which they proposed was due to buffeting by the unsteady separated flow. Meanwhile, the visualisation study of Mahalingam et al. (1997) showed their vortex not to break down during impingement with the leading edge of the trailing wing, but to be 'cut' into two vortices, one which passes beneath, and the other over the wing.
Figure 2.34: Suggested cross-flow topologies for the interaction of a vortex with a stationary surface for reducing ground clearances (Mehta and Lim, 1984).

Unfortunately direct comparisons of the vortex swirl angles between these studies is not possible due to the limited data available, but it is likely that breakdown is more likely in the stagnation region where there are high swirl angles in the vortex.

2.4.3 Interaction effects relevant to Formula One cars

We have seen in Figure 1.1, the Formula One car is made up of a number of aerodynamic components, all of which interact with one another to some degree. These interactions are highly non-linear and difficult to predict, and results can often appear counter-intuitive and confusing. For example, Hurst (1994) reports that a 1° change in camber angle of the front wheels of an Indy car can change the downforce
of the car by 2%.

We have already seen that an automotive diffuser has a strong influence on the downforce generated by the flat underbody of a car by lowering the pressure relative to the approximately constant base pressure. This base pressure may be lowered by the use of a rear wing (Katz and Dykstra, 1989) which has a large effect in lowering the pressure through the diffuser, as shown in Figure 2.35.

![Figure 2.35: Influence of front and rear wings on centreline pressures in the diffuser of the 1987 March Indy car (Katz and Dykstra, 1989).](image)

Figure 2.35 also illustrates how the addition of the front wing to the model has a detrimental effect on the diffuser performance, with greater pressures through the entire diffuser with the front wing (no rear wing) compared to the no wings case. This is because viscous interaction with the front wing lowers the energy of the flow, which then passes through the underbody and diffuser. It is also possible that the upwash generated by the front wing diverts air away from the underbody, reducing the mass-flow rate through the underbody, thus increasing the pressure.

The front wing receives clean airflow and operates close to the ground, and so it is easy to create large amounts of downforce with it. However, the main wing element of a front wing is usually set to relatively low angles of attack, as it can otherwise have a strong detrimental effect on the underbody and diffuser flow, and can even reduce the performance of the rear wing (Katz and Garcia, 2002). For example, increasing the wing flap angle causes an increase in downforce on the wing, but at high angles can reduce the total downforce acting on the car, as shown in Figure 2.36. This results in a forward shift in the balance of the car, with far more downforce generated at the front axle than at the rear. This would result in a
poor driving condition with the car prone to oversteer.

![Graph showing the effect of front wing flap angle on downforce generated by the front wing ($C_{LW}$) and the whole of a generic Indy car (Katz and Garcia, 2002).]

In conclusion, front wings are placed on the car to balance the aerodynamic forces produced by it, but they are inevitably detrimental to those components of the car downstream of it. The flow physics governing the interaction is not well understood as the overall flow field is extremely complex and non-linear. An improved understanding of how the front wing wake affects the performance of the rest of the car, in particular the underbody flow, would therefore be of benefit to the racing car designer.
Chapter 3

Experimental Arrangements and Procedures

3.1 Wind tunnel facility

3.1.1 Donald Campbell Wind Tunnel

The experimental investigations were carried out in the Donald Campbell low speed wind tunnel in the Department of Aeronautics, Imperial College London. The tunnel is of the closed-return, closed section type, with a contraction ratio of 4.92 and a working section of 1.37m x 1.21m, 2.98m long. The turbulence intensity throughout the working section is approximately 0.15%, and the maximum flow velocity is 45m/s. A breather slot at the downstream end of the working section equalises the test section pressure with atmospheric, but due to boundary layer growth on the tunnel walls there is a pressure drop along the working section of less than 3% of the dynamic head, and the maximum velocity deviation across the section is 0.75%. A detailed description of the tunnel flow may be found in Bearman, Harvey and Gardner (1976).
3.1.2 Moving Floor

The wind tunnel is equipped with a moving floor system to simulate the correct ground boundary conditions necessary for the study of ground vehicle aerodynamics. The moving floor consists of an endless belt 0.91m wide, passing over 0.3m diameter rollers set 1.52m apart. The maximum speed at which the belt can run is 40 m/s.

In order to prevent the belt from lifting up due to the low pressure beneath a model, suction is applied underneath the belt. The belt runs over a perforated platen, beneath which is a suction box containing a number of compartments. Valves within the suction box allow the suction level distribution to be varied to ensure that the belt does not lift locally beneath the model. This avoids having to apply a large suction level over the entire platen, thereby keeping friction between the belt and platen to a minimum.

Suction is also applied through a perforated sheet just upstream of the moving floor, and over the belt as it comes over the front rollers. This removes the boundary layer growing upstream of the moving floor, and the retarded flow entrained by the belt as it enters the working section. Pitot-static pressure measurements close to the ground have identified the optimum level of suction, which gives a velocity within 0.5% of the freestream down to 2mm above the ground.

Limitations of the capability of the underfloor and boundary layer suction fans effectively limit the velocity at which the floor can be operated to less than 30m/s.

3.1.3 3-Axis Wind Tunnel Traverse

The Donald Campbell wind tunnel is equipped with a 3-axis traverse which allows a probe to be manoeuvred in all three dimensions around the test section. In order to make use of this, a complete refurbishment of it was undertaken by the author, involving the installation of all motors and drivers. The traverse is driven by 4 Zebotronics 1.8° stepping motors, driven by Parker L25i intelligent drives which allow a resolution of 4000 revolutions per step. The motors are connected to two 200 pulse encoders which operate in post-quadrature to give an encoder resolution of 800 steps per revolution. The gearing in the traverse is such that one full motor
revolution results in 2mm of movement in the streamwise (X) and horizontal cross-stream (Y) axes, and 5mm in the vertical (Z) axis. The encoder resolution therefore gives a theoretical traverse resolution of 2.5\(\mu\)m in the X and Y axes, and 6.25\(\mu\)m in the Z axis. Due to compliance of the traverse structure under wind loading and backlash in the mechanical gearing, the true spatial accuracy is not as high as this. Hamidy (1991) found the spatial resolution to be better than 0.1mm, with a positional accuracy better than 0.5mm over a one metre length. The traverse is controlled by a personal computer via an RS232 link, and a program was developed in Visual Basic which allows large surveys across streamwise planes to be carried out automatically.

3.2 Wind Tunnel Models

As the project evolved, the models used to simulate the aerodynamic interactions being studied evolved, from a simple pair of identical wings in tandem, to a flat bottomed diffuser equipped bluff body with a model front wing. All of these models were designed specifically for the project, and built in the workshop of the Department of Aeronautics at Imperial College. A description of all of these models is given in the sections which follow.

3.2.1 Tandem wing pair

An initial set of experiments was carried out in which the forces were measured on the rear wing of a tandem pair. Both wings featured a symmetric 18\% thick section, had an aspect ratio of 2, and were fitted with endplates. The section coordinates may be found in Appendix B.1, and a dimensioned drawing in Appendix A. The downstream wing was hung from the tunnel's 3-component balance, 3 chords downstream of the other wing, such that both wings could be adjusted in ground clearance and incidence. The tests were conducted at a flow velocity of 20m/s, giving a Reynolds number based on wing chord of 3.4 \(\times\) \(10^5\). Figure 3.1 shows a diagram of the arrangement.
3.2.2 Large Wing Model

In order to produce a more realistic reproduction of the relative proportions of a front wing and car body combination, a large wing with a chord of 500mm and an aspect ratio of 1 was manufactured for use with the front wing model described below. The wing was of a NASA GA(W)-1 type wing section, with a 16% maximum thickness. These co-ordinates are listed in Appendix B.2. The wing was fitted with endplates for some of the experiments and rounded wing tips for others, for reasons which will be described later on in this report. Dimensioned drawings of the wing for both cases are provided in Appendix A.

Copper tubing was set in the surface and along the span of the wing at a number of chordwise locations on both upper and lower surfaces such that it ran from one wing tip to the mid-span. By drilling holes in the tube at a number of spanwise locations, and covering all but one up on each, a set of pressure tappings around the wing surface at any spanwise location could be created.

Figure 3.2 shows the wing model with the endplates and rounded tips. For these experiments, the incidence of the wing was fixed at 0° and the ground clearance is defined as the clearance between the undersurface of the wing and the ground plane.
3.2.3 Diffuser Equipped Bluff Body

The underside of a Formula 1 car may be characterised as having a flat bottom, with diffusers at the rear which exhaust to the bluff rear end of the car. It is therefore important that the model used to simulate the interaction mechanisms being studied includes these characteristics. To this end, the model on which the majority of the experiments were conducted consists of a bluff body, with an underbody which is flat apart from a single variable angle diffuser with endplates over the rear 20% of the chord. The diffuser ramp angle may be varied from $0^\circ - 20^\circ$ in $1^\circ$ increments. In order to suppress flow separation on the forebody, the nose is elliptical in shape, and the upper side edges are rounded. Figure 3.3 shows a diagram of this model. A scale dimensioned drawing is provided in Appendix A.

There are a number of pressure tappings on the model surface: 20 along the underbody centreline; a further 62 tappings arranged in spanwise rows both on the flat underbody and on the diffuser ramp; and 8 pressure tappings on the rear surface of the model to allow the base pressure to be measured. As will be described later, only 64 pressures could be measured, so 64 of these 90 tappings were chosen after
some experimentation to best capture the trends in surface pressures on the model.

### 3.2.4 Front Wing Model

A true Formula 1 car front wing is highly complex, with spanwise variations in chord and twist, with flaps, flow deflectors, and complex endplates. In order to keep the study at a fundamental level, a simplified model of a Formula 1 car front wing was produced incorporating the most basic key features of a horizontal cambered wing with endplates. The wing section is the same as that used for the large rear wing mode described in Section 3.2.2. From observations of models used in other studies of Formula 1 front wings, this aerofoil section was found to best represent the aerofoil sections generally used.

The wing was machined from solid aluminium in 5 blocks, each of chord and span 100mm, so that any aspect ratio from 1 to 5 in steps of 1 could be created. Figure 3.4 below shows a scaled diagram of the wing, set up for an aspect ratio of 5. A dimensioned drawing is provided in Appendix A.

A photograph of the front wing model placed upstream of the large rear wing with endplates is shown in Figure 3.5.
3.2.5 Vertical Wing Models

During the course of the project, it was found that the trailing vortices shed by the front wing model play a very important role in the interaction process. It was therefore desirable to look at the effect of a set of trailing vortices on the rear body, without the other components of the front wing wake, in particular the momentum deficit in the wake. In order to achieve this, a number of different methods of creating an isolated trailing vortex system were tested, and the best method was found to be a pair of vertical wings. Placed far enough upstream so that the vortices are
fully developed (i.e. complete roll up of the vortex sheet), the vertical wings each produce a single vortex at the lower tip, whilst the body of the low energy wake and other trailing vortex pass over the rear body.

Four vertical wings were manufactured, all with a wing section similar to that of the front wing, i.e. a 16% thick NASA GA(W) section, but with a chord of 70mm. The wings have a span of 210mm. The wings are suspended from a vertical plate with mounting blocks designed to allow the horizontal cross-stream separation of the wings, and their incidence to be altered. The mounting plate is suspended from two vertical struts in the wind tunnel so that the height of the whole assembly above the ground may be adjusted as required.

Figure 3.6 shows a diagram of the four wings mounted on the assembly described above, whilst Figure 3.7 shows a photograph of the vertical wings with the diffuser model mounted in the wind tunnel. A dimensioned drawing of the vertical wing assembly is provided in Appendix A.

![Diagram of vertical wings](image)

Figure 3.6: All four vertical wings assembled with their mounting system, allowing adjustment of incidence and horizontal separation.

### 3.3 Model Measurements

In the sections to follow, the measurement techniques employed in the experimental investigations of this study are described.
3.3.1 Surface Pressure Measurements

Surface pressure measurements were made on both the large wing and the diffuser models. In the case of the large wing model, a Scanivalve was used in conjunction with two pressure transducers, whilst in the second case, a ZOC 32 channel scanning pressure transducer was used. These two methods are described below.

3.3.1.1 Scanivalve system

A 48 port Scanivalve mechanical pressure scanning valve was used to multiplex the 48 pressure tappings to a single central port. This central port is then connected to a Honeywell 163PC01D36 differential pressure transducer, which measures the pressure at each tapping relative to the tunnel static pressure. A second identical pressure transducer is used to measure the tunnel reference pressure. Since the pressure transducer was outside the tunnel, and the Scanivalve inside the model,
long lengths of tube were required. This results in a significant settling time (time between the Scanivalve switching to a new port, and the start of the transducer output measurement) each time the Scanivalve switches ports, and for all of the measurements, a settling time of 2 seconds was used. The outputs of both transducers (one for the pressure tapping, and one for the tunnel reference pressure) were read simultaneously at a rate of 200Hz for 2 seconds using a PCI-DAS1602/16 16 bit analogue-digital card mounted in a personal computer. The long scan time was required for optimum repeatability due to pressure fluctuations and electrical noise resulting in a fluctuating signal from the pressure transducers.

The pressure transducers have a maximum range of ± 5.0 inches H2O, giving a span of 2490 Pa, with a repeatability and hysteresis accuracy of 0.25 % full range. With the wind tunnel running at a velocity of 20 m/s, the dynamic head equals 245 Pa, approximately 10 % of the transducer range. The transducer is accurate to 6.2 Pa, and thus the transducers give a pressure coefficient accuracy better than ± 0.026.

3.3.1.2 ZOC pressure scanning system

Pressure measurements on the diffuser model surface were made using a ZOC 22B TCU /32PxX2 electronic pressure scanning module. The unit consists of 32 differential pressure transducers with pneumatically controlled duplexing valve calibration valves. The duplexing valve allows one of two pneumatic inputs for each transducer to be selected, allowing a total of 64 pneumatic inputs in total. The calibration valve allows a single calibration pressure to be applied to all of the 32 transducers, with a common reference pressure input.

The module transducers have a maximum range of 10 inches HgO, equivalent to 2490 Pa, with an accuracy of 0.20 % full scale range (FS), equivalent to 5.0 Pa. For a wind velocity of 20 m/s, as described above, this results in a pressure coefficient accuracy better than ± 0.021.

The pressure scanning module is mounted in a thermal control unit, which maintains a temperature of 40° ± 0.2° in order to minimise the zero and scale drift during use. The zero and scale temperature sensitivity of the module is 0.25 % and 0.10 %
FS/°C. In the worst case, this reduces the accuracy of the unit by a further ± 1.7 Pa, giving an overall accuracy better than ± 6.7 Pa, resulting in a pressure coefficient accuracy better than ± 0.028 at 20 m/s.

By using the wind tunnel reference pressure to supply the ZOC module with a calibration pressure, measured using a Betz manometer, the transducers were individually calibrated at least every 2 hours using at least 5 calibration points by the method of least squares. Since the temperature sensitivity of a transducer's zero is greater than that of its scale, zeros were taken before every set of measurements.

3.3.2 Body Force Measurements

Two different balances were used during the course of the study to measure body forces. The balances and their methods of use are described in the two following sections.

3.3.2.1 Overhead balance

The Donald Campbell wind tunnel is equipped with a weigh beam type 3-component balance, designed to measure lift, drag and pitching moment with a resolution of 0.01 lbf (0.044 N), 0.001 lbf (0.0044 N) and 0.001 ft lbf (0.015 Nm) respectively. This accuracy is not achieved in practice since the model is attached to the balance via struts, which are within a fairing for the most part, but have some area exposed, on which the aerodynamic forces augment the model forces measured by the balance.

The overhead balance was used to measure the drag on the downstream wing of the tandem wing pair. Due to the setup required to move the model supports far enough back for this purpose, it was not possible to measure pitching moment. An attempt was made to quantify the forces on the struts by suspending a 6mm diameter cylindrical bar between the two struts, and assuming a drag coefficient of 1.2 for the cylinder. The tare forces using this method were found to be negligible in lift, and approximately 1.5 - 1.6 N in drag, depending on the model ground clearance. It is unlikely that these values are accurate to less than 0.1 N, but they are unlikely to vary for a fixed ground clearance, and so the changes in drag due to an upstream
wing for a fixed rear wing ground clearance may be found with an accuracy better than 0.1 N, although absolute values are significantly less accurate than this.

### 3.3.2.2 Internal balance

In order to eliminate tare forces, a 5-component balance was designed and manufactured. The balance has 6 shear webs, each with shear strain gauges fitted either side to form a full bridge, arranged so that each measures force in one direction only: 4 in lift; 2 in drag. A schematic view of the balance load cell arrangement is shown in Figure 3.8.

![Figure 3.8: Schematic diagram of internal balance load cell layout.](image)

Appendix C contains a more detailed description of the balance design and calibration methodology, examples of the finite element analysis used in its design, a photograph and technical drawing.

The balance calibration has shown the accuracy of the balance for each component of force to be as follows:

- Lift $\pm 0.6\%$
- Drag $\pm 1.1\%$
Pitch  ±0.3%
Roll  ±0.3%
Yaw  ±4.4%

The yaw component accuracy is poor, as was expected from the balance design, since the principle aims of the design were to measure lift, drag, and pitching moment. However, the balance is able to give a qualitative measure of any yawing moments, which would generally not be anticipated for the measurements conducted in this study.

A diagram of the balance mounted inside the diffuser model is shown in Figure 3.9.

![Figure 3.9: Location of internal balance in diffuser model.](image)

The forces are resolved to a point in the horizontal plane at the centre of the balance on its lower surface. This translates to a point laterally in the model centre, at \( x/c = 0.454 \), measured downstream from the model nose, and \( z/c = -0.076 \), measured from the lower surface of the model (\( z \) positive vertically downwards).

### 3.4 Wake Measurements

To enhance the understanding of the flow structures involved in the study, a number of wake measurement techniques were employed, all of which are described below.
3.4.1 Smoke Flow Visualisation

Two methods of smoke flow visualisation were utilised to visualise the flow structure and behaviour. The first involves illuminating a cross-stream plane with a laser light sheet, and injecting smoke into the flow upstream of the model, whilst in the second a small diameter smoke plume is generated.

3.4.1.1 Smoke and laser sheet visualisation

Non-persistent smoke from a Concept Spirit 900 smoke is introduced into the flow in the contraction. Since the wind tunnel test section is at atmospheric pressure, the contraction is at a pressure above atmospheric. The smoke generator is outside the contraction, in order to minimise disturbance of the flow, and so itself needs to be pressurised. A further problem common to non-persistent smoke generation is excessive condensation of the smoke oil between the smoke generator and the point of injection into the contraction. A unit was designed in which the smoke generator could be placed which would both pressurise the smoke generator, and allow the smoke to pass through a chamber in which any large smoke oil droplets could condense. A schematic diagram of this unit is shown in Figure 3.10.

![Figure 3.10: Schematic diagram of smoke generation system.](image)

The fan blows air through a pipe with a contraction between the vents for the two compartments such that the compartment containing the smoke generator is
at a higher pressure than the settling chamber to ensure that no smoke enters the generator chamber. The fan speed may be adjusted to regulate the pressure difference between the settling chamber and the wind tunnel contraction, which controls the rate of flow of smoke into the tunnel.

A cross-flow plane in the flow is illuminated by a laser sheet generated by a Lexel Model 95 Argon Ion 6 W laser. The beam is spread into a light sheet by passing it through a cylindrical lens.

3.4.1.2 Smoke plume

An Aerotech SGS 10 smoke generator is used to generate a smoke plume. The generator consists of a power supply, which provides power for a vaporiser tip at the end of a metal tube through which a persistent smoke oil is pumped by a peristaltic pump. This results in a dense plume of smoke which can be positioned manually within the tunnel, and is clearly visible across the range of flow speeds at which the experiments were carried out. The smoke plume is useful for identifying local flow directions, vortices, and separation bubbles.

3.4.2 Total Pressure Surveys

Total pressure surveys were made using the 3-axis wind tunnel traverse described previously, to which a yaw-insensitive total head probe similar to the type developed by Kiel (1935) is attached. The probe works on the principle that for an axisymmetric tube of suitable length in yaw, to a certain extent, the streamlines entering the tube are deflected along its axis. By forming a venturi along the inside of the tube, and creating a sharp edged inlet, this effect is enhanced by suppressing separation of the flow entering it. A conventional total head probe is placed within the venturi. A calibration of the probe used in these experiments showed it to be insensitive to up to 35° of yaw. A schematic diagram of the construction of the probe used is shown in Figure 3.11.

The probe pressures and tunnel reference pressure were measured relative to the test section static pressure with the same Honeywell pressure transducers used with
the Scanivalve, described above. For each total head reading, a 2 s settling time was allowed before reading the pressure transducer output for 2 s at 100 Hz. The resulting pressure readings were averaged, and the mean and standard deviation of the pressure coefficient recorded. To account for any changes in the tunnel reference pressure over the course of a run, which could last several hours, the total head and tunnel reference pressure were scanned simultaneously for each pressure reading.

3.4.3 Particle Image Velocimetry

Two different digital particle image velocimetry (DPIV) systems were used during the course of the study. The first, LaVision system was borrowed from the department of Mechanical Engineering at Imperial College, courtesy of Dr. Andy Heyes, whilst the second was used as part of a demonstration from TSI, courtesy of Dr. Martin Hyde. The systems are similar but with a number of small differences which are discussed in the following two sections. The procedure for capturing the data is similar for the two systems and discussed in the third section, followed by a description of the data analysis technique.

Both of the DPIV systems use specific DPIV CCD cameras which are capable of capturing two separate images in rapid succession. A cross-correlation technique is then used to interrogate the images, and derive the velocity vector field from them.

3.4.3.1 LaVision Digital PIV System

The 2D LaVision FlowMaster consists of a FlowMaster 3S camera, having a spatial resolution of 1280 x 1024 pixels, with a dynamic range of 12 bits. The camera was used in conjunction with a Nikon 105 mm zoom lens to maximise the spatial
resolution without requiring it to be too close to the laser sheet, and so reducing the perspective error. Illumination was provided by means of a New Wave Research Solo PIV-120 Nd:YAG laser. This contains two laser heads, each capable of producing very short duration (5-10 ns) laser pulses up to 120mJ in power. These pulses are passed through a series of optics to produce a light sheet. The FlowMaster computer software synchronises the exposure of the camera images and the pulsing of each laser head to produce two images, with a small time separation between the two, $\Delta t$.

The time separation between the two images is calculated to give a prescribed probability, $P$, of a seeding particle remaining within the light sheet for both laser pulses. This is calculated for a velocity normal to the sheet, $u_N$ and a laser sheet thickness, $t_{LS}$ from the following equation:

$$
\Delta t = \frac{t_{LS}(1 - P)}{u_N(P + 1)}
$$

The data was taken with a flow speed of 15 m/s, with a light sheet thickness of approximately 1.5mm, for which a laser pulse separation of $\Delta t = 10\mu s$.

Flow seeding was provided by means of the smoke generator described in section 3.4.1.1. The smoke was introduced to the flow by means of two plastic pipes, each of approximately 1.5 m in length, aligned in the streamwise direction in the wind tunnel contraction, through which holes had been drilled around the circumference along their length to allow the smoke to flow out of them. They were held by free-standing retort stands, keeping them just off the wind tunnel floor. It was found that correctly positioning the smoke was extremely difficult to achieve due to the boundary layer suction upstream of the moving floor, and the resulting seeding was found to be very 'patchy', with many images having to be discarded due to poor seeding quality.

3.4.3.2 TSI Digital PIV System

The TSI Powerview DPIV system uses the same laser as for the LaVision system with a Powerview 4M camera. This camera has a greater spatial resolution of 2000 x 2000 pixels, and a 12 bit dynamic range, giving an improved spatial resolution
over the LaVision system. The system was used in conjunction with a TSI model 9306A 6-jet atomiser. From a supply of compressed air and a reservoir of olive oil, the atomiser produces 0.35 μm droplets, the rate of production of which may be governed by modifying the supply pressure and/or number of jets being used. The atomiser droplets were found to persist for longer than the smoke, for possibly 5-10 circuits of the tunnel, as opposed to less than one for the smoke. This means that the correct positioning of the smoke in the tunnel was less critical, and so better results were achieved with this system.

3.4.3.3 Experimental Setup and Procedure

The camera is placed some distance downstream of the light sheet, and arranged with the aim of having the optical plane of the camera normal to and centred on the light sheet field of view. A schematic diagram of the PIV setup is shown in Figure 3.12

![Figure 3.12: Schematic diagram of PIV experimental setup in wind tunnel.](image)

The laser sheet light intensity is strongest in the centre of the sheet, which diverges as it progresses away from the laser head. Ideally, the laser sheet is required as close to the ground plane as possible, whilst having the maximum intensity centred in the field of view. In order to achieve this the laser is tilted upwards, and a sheet of card used to block the lower portion of the beam, so that the lower edge of the laser
sheet is parallel, and as close as possible, to the ground plane. The digital cameras may be damaged by excessive exposure to the laser light, so reflections from any objects, in particular the ground plane, have to be avoided. A schematic diagram of the laser sheet arrangement is shown in Figure 3.13.

Figure 3.13: Arrangement of laser to produce laser sheet in required position and orientation.

The camera is focused on the laser sheet, which is achieved by first focusing the camera on a plane placed in the laser sheet, and then refined using some form of seeding (deodorant spray was used here) in the area of interest so that the optimum focus of the seeding particles is achieved. Finally, both systems have automated calibration systems which involve placing a calibration plate, on which a series of points of known spacing are printed, in the plane of the laser sheet in the area of interest. A picture of the plate is then taken with the camera, and the PIV software then determines the calibration functions required to map the camera pixels to physical two-dimensional space.

A typical experimental run would consist of the following steps:

1. Set up model in required configuration.

2. Turn off tunnel lights, run tunnel up to required speed, switch on seeding and arm PIV laser and camera.

3. Take some images to check that the seeding is adequate. This can be validated by running an interrogation on an image pair to produce vectors, which takes a matter of seconds.
4. Automatically capture a number of images. Depending on the quality of seeding, between 20 and 50 pairs of images were captured at a time.

### 3.4.3.4 Perspective Error

The largest single source of error in the PIV measurements is what is known as the perspective or parallax error. It comes from the fact that the line of sight from the camera to the viewing plane is not normal to the plane at all points. Figure 3.14 shows an exaggerated schematic diagram which helps in the explanation of this effect.

![Figure 3.14: Schematic diagram showing origins of perspective error.](image)

Consider a particle travelling normal to the light-sheet, passing through it at a velocity, $U$, between points A and B. Because the camera is viewing it at an angle to the normal to the sheet, it appears to have an in-plane velocity, $V = U \tan \theta$. The closer the camera is to the sheet, and the bigger the viewing area, the larger the perspective error is at the edges of the sheet. A contour plot of perspective error for the TSI setup is shown in Figure 3.15, which was taken from the wind tunnel with no model in the test section. The contours are of the in-plane velocity due to
perspective error as a percentage of the free-stream velocity. The maximum error is approximately 12%.

![Perspective error graph](image)

**Figure 3.15: Measured perspective error for TSI DPIV setup.**

A comparison of the setups for the two systems, with a calculated maximum of the perspective error is given in Table 3.1. For the TSI perspective error data this is in good agreement with the results.

<table>
<thead>
<tr>
<th></th>
<th>$L$ (mm)</th>
<th>$r_{max}$ (mm)</th>
<th>Max. % Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>TSI DPIV setup</td>
<td>1060</td>
<td>126</td>
<td>11.9%</td>
</tr>
<tr>
<td>LaVision setup</td>
<td>1030</td>
<td>62</td>
<td>5.8%</td>
</tr>
</tbody>
</table>

**Table 3.1: PIV setup parameters and calculated perspective errors.**

### 3.4.3.5 Data Analysis

The DPIV images were analysed within the interrogation software from the same DPIV system as for which the data was captured. The interrogation software in both cases performed cross-correlation between the images using $32 \times 32$ pixel interrogation cells, with a 50% overlap between cells. This means, for example, that for the TSI data, which was taken with a $2000 \times 2000$ pixel camera, a maximum of $125 \times 125$ velocity vectors could be extracted.
CHAPTER 3

The data was postprocessed using the in-house software developed at Imperial College by Elliott (2000). The postprocessing involved the following steps:

1. Attempt to eliminate rogue vectors, firstly by setting a limit on the in-plane velocity, and secondly, by application of a shear filter, which limits the degree of shear and is applied to the horizontal and vertical components of velocity separately using the following equation:

\[ U_{\text{min/max}} = U_{AV} \pm (1 - C_s)U_{\text{Max}} \]

where \( U_{AV} \) is the average of the surrounding vectors, and \( U_{\text{Max}} \) is the maximum velocity as described above and set by the user, and \( C_s \) is a coefficient varying between 0 and 1, also set by the user. Any vectors falling outside these criteria are removed. The values used require some level of expectation of what may be reasonable, together with some experimentation so that the maximum number of (apparently) false vectors are removed, whilst minimising the number of good vectors removed.

2. The vorticity field is calculated using Richardson's extrapolation (as described in greater detail in Elliott (2000)), which reduces noise in the resulting vorticity field by reducing the truncation error in the calculation.

3. A routine was written by the author to identify the centres of the vortices, and their radius. This allows any number of vortices to be identified by first locating a user-specified number of vorticity peaks, and then searching around these peaks for the vortex centre. The methodology used is discussed in greater detail in Section 6.4.3 of this thesis.

3.4.4 Hot-wire Anemometry

A single hot-wire was used in some of the experiments, using the WOMBAT constant temperature anemometer which has been built in-house (Hoffmann, 1981). The wires are approximately 1 mm in length and 5 \( \mu \)m in diameter, and made form Wollaston Platinum wire. The anemometer operates at an overheat ratio of 1.8,
which, assuming an ambient temperature of 293 K, gives an operating temperature of 527 K, or 254° C. Data was captured using a PCI-DAS1602/16 data capture card installed in a personal computer.

The rate at which the hot-wire was signalled is calculated from the highest theoretical frequency that the wire was likely to be exposed to. The smallest scale it can resolve is equal to the wire length, 1 mm. The frequency of the signal produced by this scale is then assumed to be the free-stream velocity multiplied by this scale. For this scale to be resolved in the power spectra, a sampling frequency of double this frequency is required. A total of $2^{24}$ data points were gathered for each run. The sampling frequency, $f_s$, is calculated in terms of the wire length, $l$, and the freestream velocity, $U_\infty$ by

$$f_s = 2 \frac{l}{U_\infty}$$

Calibration of the probe was performed in the wind tunnel by moving the hot-wire, attached to the wind tunnel traverse for all of the tests, into the free-stream, and calibrating it against a pitot-static probe. The hot-wire signal was measured for 5 different flow velocities, and a King’s law (Bruun, 1995) was fitted to the data. This law has the form

$$E^2 = A + BU^n$$

where $E$ is the anemometer bridge voltage, $U$ is the velocity normal to the wire, and $n$ is taken as 0.45. $A$ and $B$ are constants, calculated from the method of least square, to form a linear relationship between $E^2$ and $U^{0.45}$.

The data was processed in Matlab using the 'pwelch' routine. This uses Welch type hamming windows (Press, Teukolsky, Vetterling and Flannery, 2001) to reduce the noise in the resulting spectrum, with $2^{14}$ data points in each window. This leaves $2^{10}$ hamming windows which are averaged together to give $2^{13}$ points in the (single-sided) power spectral density.
3.5 Axis conventions

In the context of racing cars, one of the primary objectives is to generate negative lift, i.e. a vertical force directed towards the ground to increase the normal force between the tyres and the ground. This force is generally increased with increasing nose-down incidence of the car. It is therefore useful to define a coordinate system in which the forces being discussed are positive along the relevant axes, and for which positive changes in incidence result in positive changes in the vertical force. Furthermore, to avoid constant reference to 'negative lift' the term 'downforce' shall be used for the vertical component of force generated by a body directed towards the ground plane. The axis system to be used throughout this thesis, unless otherwise specified, is shown in the diagram of Figure 3.16.

![Coordinate system to be used throughout the thesis.](image)

3.6 Wing Tunnel Corrections

There are a number of corrections which may be applied to the wind tunnel data, not all of which have been used, but which should be mentioned all the same. These are discussed in the following two sections.
3.6.1 Solid Boundary Corrections

The solid walls of the wind tunnel impose constraints on the tunnel flow. This has two main effects: firstly, the blockage of the model and its wake in the test section result in a flow velocity that is higher than it would be in unconstrained flow; secondly, the tunnel walls restrict the velocity normal to them to zero, forcing conditions on the flow that would not occur in the free-stream, resulting, for example, in a reduction in downwash in the wake of a positive lifting wing.

3.6.1.1 Blockage correction

A number of models for the effects of solid and wake blockage on the pressure distribution and body forces exist (Davis, 1982; Barlow, Rae and Pope, 1999; Mercker, 1986) to account for the fact that the tunnel walls constrict the flow, causing it to accelerate. This results in a dynamic head, $q_c$, which is greater than that which would exist for the same tunnel settings with the test section empty, $q_0$. Mercker (1986) developed a semi-empirical method for calculating a correction factor which gives a ratio between the $q_c$ and $q_0$ developed for automotive car testing in close-section wind tunnels, and is therefore relevant to the experimental setup used here. This is applied to the diffuser model, having the greatest frontal area and so greatest blockage, which for a typical configuration and gives:

$$\frac{q_c}{q_0} = 1.005$$

This value is small enough to be within the level of experimental error. Furthermore, since the experimental data is only provided for comparison between configurations, all of which have the same blockage levels, and there is some controversy over which blockage correction is best, it was decided that no blockage correction would be applied.

3.6.1.2 Downwash correction

Considering the trailing vortex system from a wing in the wind tunnel, the walls of the wind tunnel may be simulated as image vortex systems, reflected about the
walls. With the model in the centre of the tunnel, this tends to reduce the upwash in the wake of the model, also reducing the induced drag. However, when a model is being studied in ground effect, this influence of the tunnel walls is minimised. A schematic diagram of the trailing vortices from a negative lifting wing in ground effect is shown in Figure 3.17 together with some of the tunnel wall image vortices. In reality these image vortices continue to infinity due to reflections of reflections, and only the closest sets of image vortices are shown here.

![Diagram of trailing vortices from a negative lifting body in ground effect, and the image vortices in the tunnel walls.](image)

Figure 3.17: Schematic diagram of trailing vortices from a negative lifting body in ground effect, and the image vortices in the tunnel walls.

Since the trailing vortices are in ground effect, the image trailing vortices in the ground are extremely close to them. Therefore their images in the other walls are also close to the images of the ground plane image vortices, reducing their induced velocity at the location of the model. For example, the distance from the trailing vortices to their image in the top boundary is much greater than the distance between the two sets of image vortices in the top boundary. The influence of these four vortices on the flow at the physical trailing vortices is therefore minimal. For this reason, this effect has been disregarded.
3.6.2 Wind Tunnel Calibration

The wind tunnel reference pressure is used to set the wind tunnel velocity, and to non-dimensionalise all forces and pressures as coefficients. The reference pressure is the difference between the static pressure upstream \( p_1 \) and downstream \( p_2 \) of the tunnel contraction. The pressure downstream of the contraction is also used as a reference for the differential pressure transducers. Due to pressure gradients in the tunnel, which are complicated by the presence of the boundary layer suction which acts as a sink, \( p_2 \), does not equal the test section static pressure, \( p_s \), at the position of the model. Clearly, \( p_1 \) does not equal the total pressure, \( p_0 \), in the test section either. Therefore, two calibration factors are required to recover the true coefficients.

The conversion from the measured pressure coefficient \( C_{pm} \) to the true pressure coefficient \( C_{pc} \) is derived as follows.

\[
C_{pc} = \frac{p - p_2}{p_1 - p_2} = \frac{p - p_s}{p_0 - p_s} \times \frac{p_1 - p_2}{p_1 - p_2} \times \frac{p_1 - p_2}{k_1} + \frac{p_2 - p_s}{k_2}
\]

Two calibration factors are therefore required: \( k_1 \), the ratio between the true dynamic pressure and the tunnel reference pressure; and \( k_2 \), which represents the difference between the tunnel static pressure and the true static pressure at the location of the model.

The definitions of \( k_1 \) and \( k_2 \) pose some small problem, since the static pressure in the tunnel will depend on the location in the tunnel, and the presence of the model support struts. For the purposes of the experimental results presented in this report, calibrations were taken at the position of the model on which forces were being measured, with the support strut cowlings but without the model itself. These were measured with the rolling road, and boundary layer and underfloor suction running.

Typical values for the two coefficients versus the tunnel reference pressure, \( p_1 - p_2 \), are plotted in Figure 3.18.
3.7 Numerical Simulations

During the course of the project a number of numerical simulations of the flows being studied were performed using the commercial CFD package, Fluent. Fluent is a highly versatile finite volume Navier Stokes solver. It incorporates a number of different flow solvers and models covering a wide range of fluid problems. The aim of this exercise was to compare CFD data from an industry standard package with the experimental results. Ultimately, a full comparison between numerical and experimental data proved to be beyond the scope of the project owing to the time taken for three-dimensional mesh generation and for the solution to run to convergence. A full evaluation should include mesh convergence tests, and ultimately the scale of this task proved to be beyond the capacity of the project.

A full description of the methodology and theory used by Fluent is therefore not provided here, but a brief description of some of the more important features of the computational set-up and models used will be given.

For the majority of numerical data presented in this report, the Reynolds-averaged Navier Stokes (RANS) solver was used with the Reynolds Stress turbu-
lence model, so called because it models each individual component of Reynolds Stress. A small number of inviscid runs were also carried out for an evaluation of viscous effects, as described in the relevant chapter. The RANS turbulence model was used because it models the individual components of Reynolds stress, therefore attempting to capture the anisotropy of the flow, whereas the eddy-viscosity models assume isotropy of the turbulence. This is an extremely important factor for a number of flows involving stagnation, separation and recirculation (Hanjalić and Jakirlić, 2002), and diffuser flows (Apsley and Leschziner, 1999), all of which occur in the flows being studied in the present investigation.

Fluent has two methods available for treating the flow next to a wall: the near-wall treatment, in which the boundary layer is fully resolved down to the viscous sublayer; and the wall function approach, which uses semi-empirical functions to tie the solution in the near wall cells to that at the wall by assuming a velocity profile between the two. For the boundary layer to be fully resolved, a greater mesh resolution is required, with the first point in the mesh positioned at $y^+ \approx 1$, with 10 cells at $Re_y < 200$. Meanwhile, the wall functions require the first point to be at $y^+ \approx 30$. This cuts down the required mesh density significantly, but due to the assumptions involved, the accuracy of the solution is reduced, particularly where the flow conditions are significantly different to those assumed in the wall function formulation. These include strong pressure gradients resulting in flow separation, and highly 3D boundary layers. Both of these, particularly the strong pressure gradients leading to separation are encountered in the flows being studied, but computational resources were not sufficient to use the near-wall treatment. A $y^+$ of approximately 30 was achieved over the majority of solid boundaries in the simulations using an initial 'guess' mesh, from whose results the required mesh density over those surfaces were calculated. In some cases, such regions of stagnation, slow-moving or separated flow this was not possible.

Due to the use of wall functions and turbulence models, and the likelihood of separation in the diffuser, the solution was highly sensitive to the mesh refinement and topology and mesh convergence for the 3D diffuser simulations was not achieved. As will be discussed in due course, the diffuser model forces are highly sensitive to
boundary layer growth and separation due to the adverse pressure gradient through
the diffuser. Prediction of flow separation under these circumstances, as stated
above, is a particularly difficult problem for wall functions and turbulence models.
However, a number of flow features appeared in the numerical simulations which
were confirmed by various means in experiment. The CFD data has therefore been
used to expand the understanding of how some of these features form, behave and are
likely to affect the results only where the observed flow features were also confirmed
by experiment.

A viscous hybrid mesh was used for all of the CFD runs, with a structured mesh
close to all solid boundaries, and the central symmetry plane: symmetry of the flow
was assumed to reduce the size of the required mesh. A structured grid was also
used in the wake of the diffuser model to reduce numerical diffusion as much as
possible. The remainder of the mesh was filled with tetrahedra.

The mesh was constructed to give a maximum blockage of 1%, extending 5 model
lengths upstream and 10 downstream of the diffuser model. The ground plane was
simulated as a wall moving at the free-stream velocity, in the same way as the
rolling road in the wind tunnel. The plane bisecting the centreline of the model,
the boundary opposite it, and the boundary opposite the ground were all simulated
as symmetry boundaries. This specifies zero velocity normal to the wall, and zero
gradients of all quantities normal to those walls. The velocity-inlet and pressure-
outlet boundaries were used on the inlet and outlet planes. The first specifies a
constant velocity normal to the boundary and turbulence quantities (length scale
and intensity), whilst the latter specifies a constant static pressure at all points on
that plane. The turbulence quantities were set to the same as for the wind tunnel,
with a turbulence intensity of 0.15%.

The mesh density on the model surface was set with the aim of having adjacent
hexahedral cells with an aspect ratio of 5 (normal to the wall), with the first cell
centroid at \( y^+ \approx 30 \). The cell resolution in the wake was made as fine as possible
without exceeding the computational resources available. The computations were
conducted on a dual AMD Athalon MP 1800+ processor PC with 2 Gigabytes of
RAM. The resulting mesh has just over \( 2 \times 10^6 \) cells. Figure 3.19 shows the grid
construction on the symmetry plane around the model.

![Figure 3.19: 3D grid on symmetry plane around diffuser model.](image)

For the 2D calculations on the wing in ground effect, a structured C-grid was used around the model and in the wake, with the remainder of the domain filled with triangles to reduce the number of cells. Figure 3.20 shows a typical computational grid in the vicinity of the wing.

![Figure 3.20: 2D computational grid in vicinity of wing.](image)
A Formula One car chassis is effectively a low aspect ratio negative lifting body, complicated by the various aerodynamic devices attached to it. To begin this investigation with the simplest possible model, the aerodynamics of a low aspect ratio wing in ground effect will be considered.

The principal effects of ground proximity and endplates on wing performance, as discussed in Chapter 2, may be summarised for the negative lifting case as follows:

1. Proximity to the ground plane reduces the upwash generated at the lifting line by the trailing vortices. This reduces the induced drag and increases the lift-curve slope. This is equivalent to an effective increase in aspect ratio.

2. At low ground clearances the wing thickness distribution results in an induced 'negative camber'. This increases the incidence for zero lift, resulting in a negative lift at zero incidence for a symmetrical section wing, and increases the nose-up pitching moment.

3. Downforce increases with ground proximity up to a maximum, below which it falls. This results from the conflicting effects of increasing peak suction, due
to constriction of the flow between the wing and the ground, and increasing trailing edge separation, due to the growing adverse pressure gradient towards the rear of the lower (suction) surface.

4. Endplates increase the lift-curve slope, reduce the induced drag, and increase loading at the tips of the wing by reducing the three-dimensionality of the flow.

5. The trailing vortex forming from the lower endplate edge delays flow separation near the wing tips by re-energising the flow, and providing an upwash inboard of it.

In order to improve the understanding of the aerodynamics of low aspect ratio wings in ground effect, experiments were conducted on a wing of aspect ratio 1.0 with and without endplates, as described in Section 3.2.2. These results of these experiments are presented and discussed in the following sections:

4.2 Effect of Ground Proximity on the Pressure Distribution of a Low Aspect Ratio Wing.

Surface pressures from wind tunnel measurements on the large wing model with rounded tips are presented for a variety of ground clearances to investigate how the three-dimensional pressure distribution is affected by the presence of the ground plane.

4.3 Two-Dimensional Considerations

Two-dimensional CFD data is presented for a variety of ground clearances to examine the two-dimensional effects governing the behaviour of the wing.

4.4 Effects of Three-Dimensionality

The results of flow visualisations and total head surveys of the wake of the large wing model are compared to the three-dimensional pressure distributions to explain the three-dimensional mechanisms involved in the wing aerodynamics.
4.5 Effect of Endplates on Wing Performance

Finally, surface pressure and total head surveys in the wake of the large wing model with endplates are examined and compared to the results for the round-ed tips to investigate how endplates modify the flow behaviour, and wing performance.

4.2 Effect of Ground Proximity on the Pressure Distribution of a Low Aspect Ratio Wing

There is very little surface pressure data for negative lifting wings in ground effect in the open literature. To improve the understanding of three-dimensional effects, surface pressure measurements were made over a large number of points on the aspect ratio 1 wing at 3 different ground clearances. The centreline pressures from these tests are shown in Figure 4.1.

![Figure 4.1: Centreline pressure distribution for aspect ratio 1 wing without endplates at a range of ground clearances.](image)

All three pressure distributions have an unusual ‘figure-of-eight’ shape which is
caused by the influence of the ground in lowering the stagnation point towards the suction side of the nose, raising the pressure over the lower nose surface, and lowering it over the upper side. The resulting reversal of lift on the nose, increases the nose-up pitching moment, reducing the pitching stability of the wing. The lowering of the stagnation point was one of the effects of ground proximity discussed in Section 2.2.1, explained as a purely inviscid effect of the wing thickness: its reflection in the ground plane inducing a ‘negative camber’. However, although many researchers have reported a lowering of the stagnation point, none report an effect as strong as that shown in Figure 4.1. For example the surface pressure data of Zerihan and Zhang (2000) are shown in Figure 4.2, where the stagnation point lowering is only mildly visible at the lowest ground clearance of h/c=0.067.

Figure 4.2: Centreline pressure distributions of Zerihan and Zhang (2000), with nose-lift reversal highlighted.

The wing used by Zerihan and Zhang was a modified NASA LS(1)-0413 section, whereas the section used here was the same LS(1)-0413 scaled to give an increased thickness of 16%, referred to here as the LS(1)-0416 section. A comparison between the two sections is given in Figure 4.3. This shows the difference in height between the lower surfaces to be approximately 2% of the chord. At a fixed ground clearance for the two wings, the ratio between underbody cross-flow areas at the trailing edge and at the lowest part of the wing undersurface will be greater for the greater thickness. For example, a ground clearance of h/c=0.10 for the two wings will give
an area ratio of 2.06 for the 0416 wing, and 1.86 for the 0413. Since this area ratio dictates the ideal (i.e. with no boundary layer growth or separation) pressure gradient, it will be greater for the LS(1)-0416 wing, and so separation becomes more likely.

Figure 4.3: Comparison between NASA LS(1)-0413 and 0416 sections.

The other difference between the wing used here, and that used by Zerihan and Zhang, was their greater aspect ratio and use of endplates, both of which increase flow two-dimensionality. It may be either, or a combination of both the greater two-dimensionality and the less thick section of the wing used by Zerihan and Zhang that accounts for the higher stagnation point exhibited in their results. Since the location of the stagnation point dictates the amount of air flow beneath the wing and thus the downforce, it is important that the factors affecting it are investigated. This investigation will begin with an analysis of two-dimensional effects.

4.2.1 Two-Dimensional Considerations

Two-dimensional simulations of the wing in ground effect were carried out using the commercial Fluent CFD package, as described in Section 3.7. Running the simulation in 2-D is not only simpler to perform in terms of grid generation, but also enables the two-dimensional effects to be identified and separated from the influence of three-dimensionality in the experimental results. The simulations were run at the same ground clearances as for the results of Figure 4.1, together with a simulation far from the ground. Viscous RANS simulations used the RSM turbulence model which, as mentioned in Section 3.7, is be more accurate for these kinds of flow. Chordwise pressure distributions for these simulations are shown (dotted lines) with the experimental data (solid lines) in Figure 4.4.
Figure 4.4: Comparison between 2-dimensional CFD and experimental centreline chordwise pressure distributions.

Separation of the flow on the underside of the wing is difficult to predict, since correct modelling requires an accurate representation of the boundary layer and the turbulent quantities within it. However, comparison between the CFD and experimental results can help to identify some of the mechanisms at work, even though absolute values may be inaccurate. The general trends between the experimental and computational pressure distributions are the same: lowering of the stagnation point, increasing peak suction, and increased trailing edge separation with reduced ground clearance. This is less obvious from the measured pressure distributions, but was confirmed by smoke-flow visualisation experiments. No photographs are shown of the smoke-flow since obtaining clear images was found to be too difficult. Contours of streamfunction from the 2D CFD simulations, with the limits of the contour levels set just either side of that bisected by the wing surface, are shown in Figure 4.5, showing the extent of trailing edge separation and the flow angle approaching the wing at a number of ground clearances. For comparison, inviscid calculations were also performed using Fluent. Numerical diffusion introduced by the discretisation scheme results in an implicit application of the Kutta condition.
at the trailing edge, imposing a circulation on the wing which would not exist in a truly inviscid flow. Comparison between the two sets of results can be used to illustrate the effect of boundary layer growth and flow separation on the flow field.

Figure 4.5: Contours of streamfunction for two-dimensional calculations of a wing in ground effect for values either side of that bisected by the aerofoil.

Interestingly there is virtually no lowering of the stagnation point with ground proximity for the inviscid case, and the vertical component of velocity for the flow approaching the wing remains positive (directed towards the ground). For the viscous case, however, there is a reversal in the vertical flow velocity at the nose at low ground clearance, lowering the stagnation point as discussed above. The cause of this is linked to the trailing edge separation which grows with reduced ground clearance. This results in an effective reduction in negative camber of the wing, reducing the level of downforce produced. To understand better how this affects the flow field surrounding the body, it is necessary to consider the channel between the lower surface of the wing and the ground as a diverging duct, or diffuser. To aid this description, a plot of the inviscid and viscous pressure distributions is given in Figure 4.6.
Figure 4.6: Inviscid (Chordwise pressure distributions for inviscid and viscous simulations of two-dimensional flow around an aerofoil in ground effect.

For each set of computations, the pressure at the trailing edge is relatively insensitive to the ground clearance. Changing the proportion of the flow which passes over the upper surface of the wing does not have a large effect on the pressure distribution towards the rear of the upper surface, as this flow is unbounded (unlike the lower surface, which is bounded by the ground plane). This effectively fixes the static pressure at the trailing edge, where the pressure is approximately matched to the flow at the trailing edge of the lower surface. Assuming a fixed trailing edge pressure and inviscid flow through the channel, the rate of change in cross-sectional area through the channel determines the velocity distribution, and thus the pressure distribution. In the inviscid case, there is no boundary layer growth, and no separation. The flow is therefore slowed through the diverging channel towards the rear of the lower surface according to the geometric rate of change of cross-sectional area. The predicted pressure gradient towards the trailing edge is not physical, and in the more realistic viscous case the boundary layer thickens, and the flow may separate, reducing the effective area ratio through which the flow passes. Since the velocity
and therefore pressure distribution is fixed relative to that at the trailing edge by the effective rate of change in cross-sectional area through the underbody passage, the mass flow rate is determined. The mass flow rate is reduced for the viscous case due to boundary layer growth and trailing edge separation, and so a greater proportion of the flow approaching the wing is diverted over it. As more flow passes over the wing, the vertical velocity of the flow approaching the wing away from the ground increases, lowering the stagnation point on the leading edge.

The performance of the wing is therefore heavily dominated by the effective area ratio through the rear section of the under-wing channel. As the ground clearance is reduced, and the boundary layer growth and region of trailing-edge separation grows, not only is the peak velocity, and thus the peak suction reduced, but more flow is diverted over the wing. This lowers the stagnation point, increasing the pressure on the lower side of the nose and reducing the pressure on the upper surface, reversing the sign of the lift generated at the nose.

4.2.2 Effects of Three-Dimensionality

Comparison of the experimental and computed pressure distributions of Figure 4.4 reveals some considerable discrepancies between the two-dimensional computed and three-dimensional experimental results. As previously mentioned, simulation of this kind of flow is difficult because of the limitations of RANS schemes in predicting flow separation from curved surfaces. More significantly however, at an aspect ratio of unity, the flow around the real wing is likely to be highly three-dimensional, which will account for the majority of the differences between the computational and experimental results.

As discussed in Section 2.2.1, by considering the body thickness as a distribution of sources, the ground plane image source distribution was found to cause a local change in flow incidence by means of an induced upwash at the front of the model, and a downwash at the rear. Extending this idea to three dimensions, the three-dimensional source distribution would cause an outboard component of flow near the nose, and an inboard component of flow towards the rear. This inboard component
at the rear would be augmented by the influence of the low pressure beneath the wing.

In order to investigate this a grid of wool-tufts was attached to the ground plane beneath the model. This required the ground plane to be stationary, but smoke flow visualisation showed this to have little effect on the flow directions at the ground plane. Figure 4.7 shows pictures of these wool tufts beneath the wing at the front and rear of the model, showing the outboard component of flow at the leading edge, and strong inboard component at the rear.

![Figure 4.7: Wool tufts on ground at the front and rear of a single aspect ratio 1 wing in ground effect.](image)

The three-dimensionality associated with this spanwise velocity distribution introduces the consideration of a non-constant mass-flow between the wing and the ground at the centreline. Since flow escapes from the sides of the wing just downstream of the nose, the peak flow velocity under the wing will be less than that for the two-dimensional case, thereby reducing the peak suction. This effect is apparent in Figure 4.4, where there is a delay in the pressure decrease downstream of the nose in the experiments compared to the two-dimensional case, particularly at \( h/c = 0.08 \) and 0.10. This is less prominent at \( h/c = 0.05 \). As the ground is approached, the aspect ratio of the channel formed between the suction surface and the ground rises; at \( h/c = 0.05 \) this aspect ratio is twenty. There must therefore be some form of balance between the rising aspect ratio of the underwing channel, which reduces
three-dimensionality, and the increasing ground effect, which increases the leakage of flow from the wing tips.

Figures 4.8 and 4.9 show measured chordwise pressure distributions at five spanwise locations for a ground clearance of $h/c = 0.05$ and $h/c = 0.10$ respectively. In the latter, arrows have been placed on the figure to show the general trends for both ground clearances with increased distance from the centreline. $x/c$ is defined as before, and $y/b$ is the distance from the model centreline as a fraction of the span. At the lower ground clearance there is far more similarity between the centreline pressure distributions at $y/b = 0.00$ and 0.25 than for the large ground clearance. This shows a greater degree of two-dimensionality at the centreline for the lower ground clearance, as suggested above.

Figure 4.8: Spanwise variation in chordwise pressure distributions without endplates for $h/c = 0.05$.

The effect of flow leakage at the tips is to reduce the peak suction in the chordwise pressure distributions, as for a conventional wing out of ground effect. This applies to the peak suction on both the upper and lower surfaces of the wing, and it is not possible to tell the change in stagnation position along the span from this data alone. The pressure at the trailing edge is higher at the centre of the wing than
Figure 4.9: Spanwise variation in chordwise pressure distributions without endplates for $h/c = 0.10$.

at the tips. This is most probably an effect of the wing trailing vortices, and the low pressure associated with them. Total pressure measurements were made 10mm downstream of the wing trailing edge for wing ground clearances of $h/c = 0.05$ and $0.10$. These results are shown in Figure 4.10 below. The location of the trailing edge is drawn on the diagram in black.

At the trailing edge, the vortex is just inboard of the wing tips, on the suction side. This vortex brings in high energy flow inboard from the wing tips. In these experiments the vortex also draws in some of the low energy flow from the wake of the struts either side of the wing, visible as a band of lower energy flow just outboard of the wing tip. For the lowest ground clearance, at the wing centre there is evidence of a thick boundary layer on the lower surface of the wing. At $y/b \approx 0.35$ there is an abrupt thinning of this band of low energy flow. This is the result of re-energisation by the trailing vortex bringing high energy flow in from the tips and thinning the boundary layer. This has two conflicting effects: The first is an increase in effective area ratio between the wing and the ground towards the trailing edge due to a thinner boundary layer; the second is of an increasing mass-flow rate
along the expanding channel between the lower surface and the ground. The first effect will tend to improve the pressure recovery towards the trailing edge, whilst the second will tend to reduce it. These combined effects probably account for the apparent two-dimensionality exhibited in the pressure distributions at \( y/b = 0.50 \) and 0.25, and the change in behaviour of the distributions outboard of \( y/b \approx 0.35 \) for \( h/c = 0.05 \).

4.3 Effect of Endplates on Wing Performance

Endplates are used on Formula One car wings in order to reduce the flow three-dimensionality. They have been shown to increase the effective aspect ratio of the wing, increasing loading and delaying stall at the wing tips (see Section 2.2.3).
limited set of measurements were made on the aspect ratio 1 wing with endplates. Chordwise pressure distributions at a number of spanwise locations for the wing with endplates at a ground clearance of $h/c = 0.10$ are shown in Figure 4.11. It should be noted that the spanwise locations are different to those used for the wing without endplates in order to highlight the change in shape of the pressure distribution near the tips.

![Figure 4.11: Spanwise variation in chordwise pressure distributions with endplates.](image)

These pressure distributions show a significant rearward shift in the location of the peak suction near the tips with endplates. Furthermore, the peak suction at $y/b = 0.25$ is greater than that at the centreline. These trends indicate a significant change in the flow field, beyond that of an increase in flow two-dimensionality. In fact the suction peak increase and rearward shift near the tips would suggest some extra three-dimensional effect. This is the result of a vortex growing from the lower edge of the endplate. The inboard flow leakage beneath the endplate separates from the sharp lower edge, forming a shear layer that rolls up into a strong vortex which convects downstream just inboard of the endplate. The total head contours of Figure 4.12 illustrate the vortex roll-up and position at $x/c = 0.65, 0.85$, and $1.05$. It was found that the flow angles in the area of the vortex exceeded $40^\circ$ in some places, and it was necessary to yaw the total head probe by $15^\circ$ to optimise its working.
range. Some testing was done with different yaw angles, and it is believed that the measurements presented here are not affected by the large yaw angles of the flow.

The high flow angles are indicative of high tangential velocities in the vicinity of the vortex. This reduces the static pressure locally, forming a track of low pressure on the wing lower surface above the vortex path. This creates the low pressure peaks aft of the point of maximum thickness on the wing near the tips. As the vortex grows towards the trailing edge, it moves inboard. This moves the peak suction inboard towards the trailing edge, which is why the lowest surface pressure at $y/b = 0.45$, for example, is aft of that at $y/b = 0.47$ in Figure 4.11.

Looking at the influence of endplates on the surface pressure distribution in more detail, Figure 4.13 shows a direct comparison between the chordwise pressure distributions at a ground clearance of $h/c = 0.10$ at $y/b = 0.0$ and 0.47 with and
without endplates.

Figure 4.13: Chordwise pressure distributions with and without endplates at $h/c = 0.10$ at $y/b = 0.0$ and 0.47.

At the centreline ($y/b = 0.00$) the suction peak on the upper (pressure) surface is greater without endplates, suggesting a lower stagnation point. With endplates, the higher stagnation point is indicative of a greater mass flow rate beneath the model, increasing the peak suction on the lower (suction) surface. Near the tips, at $y/b = 0.47$, again there is a lower peak suction on the pressure surface with endplates, indicating a higher stagnation point. This is probably due to the endplates lowering the strong trailing vortices towards the ground, such that their influence diminishes, reducing the upwash ahead of the wing. Downstream of that, the negative pressure gradient increases dramatically with the endplates as the lower endplate vortex appears. The effect of this vortex on the loading at the wing tip downstream of $x/c = 0.4$ near the tips is considerable. In order to quantify these effects, the measured pressures may be integrated over the aerofoil surface to give an estimation of the section lift coefficient. The results of this integration are given in Table 4.1.

These integrated lift coefficients show the effect of endplates in this case to be highly beneficial, increasing the loading at the wing tips beyond that at the
Endplates originated in an attempt to reduce tip-leakage and increase the effective area ratio (and thus the flow two-dimensionality). The increase in downforce at the centreline would suggest an increase in the two-dimensionality of the flow, and a reduction in induced incidence created by the trailing vortices. This is in agreement with the findings of previous researchers, as discussed in Section 2.2.3. What has not been reported previously is the potentially strong increase in suction brought about by the lower endplate vortex, which can increase the wing loading at the tips beyond that at the centreline.

<table>
<thead>
<tr>
<th>$y/b$</th>
<th>Rounded Tips</th>
<th>Endplates</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>-0.44</td>
<td>-0.62</td>
</tr>
<tr>
<td>0.47</td>
<td>-0.33</td>
<td>-0.72</td>
</tr>
</tbody>
</table>

Table 4.1: Sectional lift coefficients at wing centreline and near wing tip with and without endplates.
Chapter 5

Aerodynamics of a Diffuser-Equipped Bluff Body in Ground Effect

5.1 Introduction

The aim of the current chapter is to describe the flow physics governing the aerodynamics of the diffuser model used in the present investigation. There are a number of important mechanisms involved, which have been investigated by various means, including body force and surface pressure measurements, smoke-flow visualisation, and CFD. The presentation and discussion of the data has been divided into a number of sections as follows:

5.2 Lifting behaviour

Results are presented showing the variation in downforce over a wide range of diffuser ramp angles. Centreline surface pressure measurements for ramp angles and ground clearances are also provided. The results presented here are without forced transition.

5.3 Reynolds Number Effects
Force measurements over a range of Reynolds Number were made without fixed transition, and it was found that the model behaviour is heavily Reynolds Number dependant. Force and surface pressure measurements with transition fixed were made and are compared to the transition-free case, with a discussion of the influence of boundary layer state and growth on the model performance. All results used after this point are with transition fixed as they show less dependence on Reynolds Number.

5.4 Description of Flow-Field

A qualitative account of the three-dimensional flow field is provided using the results of CFD, smoke-flow visualisation and total head surveys. Key flow features are identified which will be used later in the chapter to explain the physics behind the behaviour described in the first section.

5.5 Influences of Three-Dimensionality

The flow underneath the model is highly three-dimensional, as described in section 5.4. The influences of this three-dimensionality are considered to build an understanding of the flow physics governing the aerodynamic behaviour of the model.

5.2 Lifting Behaviour

Force and pressure measurements were made on the diffuser model over a wide range of ground clearances and diffuser angles in order to map the basic lifting performance. It has been shown (Sovran and Klomp, 1967) that defining the geometry of a diffuser in terms of diffuser length, non-dimensionalised by the height at inlet, and area ratio is the best way of describing its performance. As described in Section 2.2.6, Cooper et al. (1998) showed the downforce of a car-like body with an underbody d-diffuser to be generated by ground-effect, diffuser-pumping and underbody-upsweep. Taking this into account, it was felt that parameterising the overall performance by ground clearance and diffuser angle is the clearest way to present the data, and those parameters will be used throughout this report to present the data.
A contour map of downforce coefficient, $C_z$, is plotted on axes of ground clearance and diffuser angle in Figure 5.1. As discussed in Section 2.2.6, owing to boundary layer thickening and flow separation from the diffuser ramp limiting the maximum attainable pressure recovery through the diffuser, an optimum ground clearance exists for each diffuser angle. A locus of these optimum ground clearances is shown as a solid line on the plot.

![Figure 5.1: Contours of downforce on axes of ground clearance, h/c, and diffuser angle, $\theta$.](image)

Surface pressure measurements were made for a wide range of diffuser model configurations to improve the understanding of the mechanisms governing the aerodynamic forces. The configuration giving maximum downforce in Figure 5.1 is a diffuser angle, $\theta$, of 14° and a ground clearance, h/c, of 0.060. Centreline surface pressures for the diffuser model at a fixed ramp angle of $\theta = 14^\circ$ at a number of ground clearances either side of the optimum are shown in Figure 5.2.
There are profound changes in the form of the centreline pressure distributions as the ground clearance is increased. At the lowest ground clearance, the minimum pressure appears some distance upstream of the diffuser entrance. It moves to the diffuser entrance in a weak form at $h/c = 0.028$ and becomes more pronounced as the ground clearance is further increased up to and beyond that resulting in maximum downforce. Meanwhile, upstream of the diffuser entrance, on the flat underbody towards the nose, the pressure falls with reduced ground clearance down to a minimum at $h/c = 0.020$, below which it rises again rapidly. The peak downforce occurs at a ground clearance representing the optimum compromise between the pressure drop across the diffuser, and the suction levels on the upstream portion of the flat underbody: the former rising and the latter falling with increasing ground clearance.

The shape of the pressure distribution also varies with ground clearance. At very low ground clearances, there is almost a linear reduction in pressure from the most upstream pressure tapping with distance downstream, with a very definite change in slope at some position upstream of the diffuser. At greater ground clearances
there is a curvature in the pressure distribution on the flat underside of the model, suggesting a possible secondary suction peak on the underbody near the nose in addition to the one at the diffuser inlet.

Figure 5.3 shows centreline pressure distributions for a fixed ground clearance of $h/c = 0.060$, where the peak downforce occurs, at a number of diffuser ramp angles either side of the optimum value of $\theta = 14^\circ$.

![Graph showing pressure distributions](image)

Figure 5.3: Diffuser model centreline pressure distributions for a number of diffuser ramp angles at a fixed ground clearance, $h/c = 0.060$.

Variation in the diffuser angle has less effect on the basic shape of the centreline pressure distribution shape. The peak suction at the diffuser inlet increases with diffuser angle up to a maximum at $\theta = 14^\circ$. This peak suction appears to lower the pressures considerably upstream of the diffuser, such that variations in diffuser angle influence the centreline pressures right up to the most upstream pressure tapping at $x/c = 0.28$. There is a very slight reduction in the pressure gradient towards the rear of the diffuser ramp at the greatest ramp angles which smoke flow visualisation showed to be the result of flow separation. The pressure gradient over the separated region is still positive, however, and so a positive but reduced rate of change of effective area is still maintained through the diffuser.
5.3 Reynolds number effects

The growth of the boundary layer beneath a body is likely to be more important where the body is in close proximity to a ground plane, since the boundary layer displacement thickness changes the local effective ground clearance. To help illustrate this, Figure 5.4 shows velocity profiles through the diffuser (0.80 ≤ x/c ≤ 1.00) taken from CFD results for θ = 12°, h/c = 0.060.

The boundary layer growth through the underbody, and in particular through the diffuser makes up a considerable proportion of the local ground clearance. In order to give an approximate value for the displacement thickness, δ*, the peak velocity of the profile is assumed as the boundary layer edge velocity, u_e, in the following integral which is evaluated numerically at each x/c location between the underbody z = Z_u and the ground, z = Z_g:

$$\delta^* = \int_{Z_u}^{Z_g} \left(1 - \frac{u}{u_e}\right) \, dz$$

The displacement thickness variation along the diffuser centreline calculated using the above method is plotted as a proportion of the local ground clearance in

---

**Figure 5.4:** Centreline vertical underbody velocity profiles in the diffuser from CFD results. θ = 12°, h/c = 0.060
Figure 5.5 using the CFD results for $\theta = 12^\circ$ and $h/c = 0.060$.

Figure 5.5: Displacement thickness along diffuser model underbody from CFD data for $\theta = 12^\circ$ and $h/c = 0.060$

The simulation of the boundary layer growth by the CFD is unlikely to be very accurate since the boundary layer is not fully resolved, and wall functions are used in the viscous sublayer. However, it does gives a qualitative description of the likely rate of growth, particularly in the diffuser.

There are likely to be two main effects of the boundary layer growth. On the flat underbody, it constricts the flow, causing it to accelerate. Assuming an inviscid core, which is reasonable since, in the CFD data at least, the boundary layers do not merge, this will cause the pressure to drop. Within the diffuser, meanwhile, it reduces the effective area ratio, thereby reducing the deceleration of the flow through the diffuser and the resulting pressure rise. Referring back to Figure 5.2, at very low ground clearances, where the effects of three-dimensionality are less prominent, there is an almost linear lowering in pressure along the flat underbody up to a position where the rate of change of pressure changes abruptly. This is likely to be a result of the boundary layer growth, with merging occurring at the point where the abrupt change in pressure gradient occurs.
As discussed in Section 2.2.6 the boundary layer growth in the diffuser is very heavily dependent on the pressure gradient, and is so probably relatively insensitive to Reynolds Number. However, on the flat underbody, where the pressure gradients are far smaller, the boundary layer growth is likely to be more Reynolds Number dependant. Only a small change in displacement thickness, increasing the effective cross-sectional area, is required to give rise to a large change in pressure, since the pressure is proportional to the velocity squared, which is inversely proportional to the local ground clearance for a fixed flow rate. It has been shown that increasing blockage at inlet not only reduces the pressure recovery but also reduces the area ratio and/or non-dimensional diffuser length for peak pressure recovery. Therefore, the more Reynolds Number dependant flow on the flat underbody has a strong influence on the diffuser performance.

Tests were conducted to examine the Reynolds number dependency of the model with and without various means of transition. As already mentioned, the flow beneath the model is likely to be highly sensitive to small changes in local ground clearance, and boundary layer thickness. It is therefore important that a boundary layer trip which causes transition with a minimal thickening of the boundary layer is used. Versmissen (1974) conducted experiments using various transition fixing methods with the aim of causing minimum drag penalty. He found that small glass beads (ballotini) were the best solution. Following Versmissen's guidelines, a row of ballotini of 0.5-1.0mm diameter spaced approximately 5mm apart were fixed just downstream of the nose, at x/c=0.21, on the underbody by means of superglue. This transition method was tested against Reynolds number, together with a 0.5mm diameter wire fixed along the underbody, and a strip of electrical tape. The tests were conducted with the diffuser in the optimum transition-free configuration of $\theta = 14^\circ$, $h/c = 0.060$. The downforce results are shown in Figure 5.6 below.

Transition is clearly an extremely important issue since, at the optimum configuration for the transition-free case, fixing transition causes a drop in downforce of almost 30%. Furthermore, for the transition free case, the downforce is a heavily dependant on Reynolds number. The electrical tape has some influence, but does not appear to have the full desired effect. The wire reduces Reynolds number depen-
Figure 5.6: Plot of downforce coefficient against Reynolds number with and without transition. $\theta = 14^\circ$, $h/c = 0.060$

dence, but not as well as the ballotini, presumably due to it creating a greater flow disturbance. It is therefore the ballotini which was felt to best fix transition with the minimum level of disturbance to the boundary layer. Ballotini were used throughout the rest of the experiments for transition fixing. Figure 5.7 below shows how the diffuser model downforce varies with diffuser ramp angle and ground clearance with and without transition.

Overall, transition seems to lower the maximum attainable downforce, and shifts the configuration for maximum downforce to a slightly greater ground clearance and lower diffuser angle. For a wing, one would expect transition fixing to delay stall as the turbulent boundary layer is more resistant to separation. For the diffuser model, however, the reverse would appear to be true, with a lower peak downforce being generated in configurations resulting in a smaller ideal adverse pressure gradient (i.e. greater ground clearance or lower diffuser angle) than for the transition-free case.

Zerihan and Zhang (2000) found similar results for their wing in ground effect, with transition fixing reducing the maximum downforce and causing the maximum to occur at a greater ground clearance. They suggested that the thicker boundary
layer entering the adverse pressure gradient for the transition-fixed case is more likely to separate. They add that this effect is highlighted in ground effect, where transition is much more significant than in the free-stream case.

For a zero pressure gradient flat-plate boundary layer, a turbulent boundary layer grows at a rate proportional to $\text{Re}_x^{-1/5}$ compared to $\text{Re}_x^{-1/2}$ for a laminar boundary layer (Schlichting, 1968). With transition fixing, the greater length over which the boundary layer is turbulent on the model underside is likely to result in a thicker boundary layer at the diffuser entrance. With transition free, meanwhile, the boundary layer still is likely to be turbulent by the time it reaches the diffuser entrance, but will not be as thick owing to the greater distance over which it has been laminar. As already discussed, viscous blockage in the form of boundary layers at inlet is known to be detrimental to the diffuser performance, and so accounts for the difference in performance with transition fixing.

Figure 5.8 shows the influence of transition on the centreline pressure distributions for a diffuser angle of $\theta = 12^\circ$ at $h/c = 0.04, 0.06, 0.08$. This indicates that
the most heavily affected area is the diffuser flow, with a considerable reduction in pressure recovery, especially at the lower ground clearances. The change in performance is therefore strongly governed by the changes in diffuser performance, i.e. a reduction in effective area ratio that is more pronounced at lower ground clearances.

![Graph showing pressure distributions with and without transition.](image)

Figure 5.8: Centreline underbody surface pressure distributions with and without transition. $\theta = 12^\circ$

Sovran and Klomp (1967) showed that the effective area at diffuser outlet can be related directly to the parameter $ARB_1^{1/4}$, where $B_1$ is the blockage at inlet, and $AR$ is the area ratio, with the effective area at outlet falling with increased area ratio and inlet blockage. Meanwhile the results of Reneau et al. (1967) suggest that thicker boundary layers are more prone to separation. This would account for why the flow on the diffuser ramp is separated, and the pressure distribution flattens towards the rear at $h/c = 0.04$ for the transition-fixed case.

Since transition-fixed results show greater Reynolds Number independence and better represents the case of a Formula One car at higher Reynolds number, unless otherwise stated, all results displayed from this point forward will be with fixed transition using the strip of ballotini described above.
5.4 Description of Flow-Field

In order to explain in greater detail the various features of the behaviour discussed in the previous section, some description of the flow field is necessary.

Although only a limited number of CFD simulations have been carried out, and the force data is not entirely in agreement with experiment, the data have highlighted a number of flow features which aid the explanation of the surface pressure and force data.

The CFD data presented here are from an unsteady 3D Navier-Stokes calculation. It was found that this gave force results closer to experiment than from the steady simulations. This is not surprising since a bluff body such as the diffuser model is likely to have an unsteady wake at the Reynolds number at which the experiments and CFD were conducted.

In previous sections, the lowering of the stagnation point on a body with increasing ground proximity has been discussed. This applies equally to the diffuser model, as shown in Figure 5.9 and confirmed with smoke-flow visualisation.

![Figure 5.9: CFD flow streamlines on model centreline showing lowered stagnation point, $\theta = 12^\circ$, $h/c = 0.06$.](image)

As shown for the wing in ground effect of the previous chapter, as the ground is approached, the flow near the nose beneath the model has an increasing outboard component of velocity due to blockage effects. Towards the rear of the model, this lateral flow component reverses and grows into a significant inboard leakage. The same phenomenon appeared in the experiments with the diffuser model, with a significant outboard flow component up to approximately $x/c = 0.40 - 0.50$ and an
inboard component of flow downstream of that. This was confirmed by smoke flow visualisation, and is shown here from the CFD results in Figure 5.10.

The limits of the colour scale have been reduced to give a clearer view of the variation in lateral velocity. For this array of streamtraces the peak outboard component of flow exceeds 12m/s, and the peak inboard component beneath the vortex exceeds 17m/s for a free-stream velocity of 20m/s. The size of these lateral velocities illustrate the extent of the flow three-dimensionality. The effects of this on the body forces and pressure field are discussed in the next section.

It was found in the experiments that the outboard component of flow results in a separation of the flow from the sharper lower edge of the nose sides, forming a vortex which runs down the side of the model. This vortex was found in the CFD results, and is shown in Figure 5.11 by two streamtraces released from the model nose. Smoke flow visualisation suggested that this vortex was stronger for the experiments than shown in the CFD. Its strength is dependant on the radius of the rounded outer edge of the nose. The nose of the experimental model was hand-made from wood, and the consequently the accuracy in the reproduction of the radius was somewhat compromised. Meanwhile, the CFD is unlikely to be accurate in this area since it involves separation from a curved surface, and so requires accurate simulation of the boundary layer state, growth and separation point. Since wall functions are used near the wall and the boundary layer is assumed to be turbulent, the resulting
vortex is unlikely to be a true representation of the physical nose vortex.

Figure 5.11: 3D CFD simulation streamtraces showing vortex shed from outer edge of diffuser model nose. $h/c = 0.060, \theta = 14^\circ$.

Early on in the experiments, a set of total head measurements were made in the wake of the diffuser model at a number of ground clearances with a diffuser ramp angle of $7^\circ$ to examine how the nose vortex behaves as it convects downstream. Figure 5.12 presents these total head measurements as contour plots of total pressure coefficient. The model centreline is at $y/c = 0$, and lines are drawn on the plot showing the horizontal rear edge of the diffuser ramp at approximately $z/c = -0.025$, and the diffuser endplate at $y/c = 0.5$, with the lower edge at $z/c = 0.0$. The nose vortex is clearly visible for all ground clearances. At the lowest ground clearance, the vortex remains at the side of the model. As the model is raised from the ground, the vortex is pulled underneath the model sides and into the diffuser. Since the upward and outboard motion of the flow diminishes with increased ground clearance, the strength of the nose vortex also diminishes.

5.5 Influences of Flow Three-Dimensionality

In the previous section a number of three-dimensional flow features have been identified, and are likely to have important consequences for the aerodynamics behaviour of the diffuser model. The influence of these features will be discussed in the sections to follow, and are categorised as follows.

1. Non-constant underbody volume flow rate.
The outboard and inboard leakage of flow over the upstream and downstream portion of the model underbody from its edges results in a non-constant volume flow rate beneath the model. The influence of this non-constant flow rate on the centreline pressure distribution is considered in a simple one-dimensional analysis.

2. Nose vortex

The outward and upward motion of flow at the nose causes a separation from the outer lower edges of the nose, resulting in vortices which run down the model sides. The effects of this nose vortex on the underbody flow will be briefly discussed.

3. Diffuser vortices

Flow leaking underneath the model sides into the underbody flow separates
from the its edges and diffuser endplates, forming a strong vortex which runs through the diffuser just inboard of the diffuser endplates. Some description of the influence of these ‘diffuser vortices’ on the underbody flow field will be given.

5.5.1 Results of non-constant underbody flow rate

As described in Appendix D, assuming one-dimensional inviscid flow the pressure, $C_{p2}$, at any point on the underbody with local ground clearance, $h_2$, may be related to that at any other point, $C_{p1}$, with local ground clearance, $h_1$, as follows.

$$C_{p2} = 1 - (1 - C_{p1}) \left( \frac{h_1}{h_2} \right)^2$$

Smoke flow visualisation in the experiments indicated an outboard lateral component of flow up to approximately $x/c = 0.4 - 0.5$ downstream of which there is an inboard component of flow which is approximately constant in the diffuser. CFD centreline velocity profiles integrated along vertical lines give approximations of the mass-flow rate per unit span. The results from this are shown in Figure 5.13, showing an approximately quadratic variation upstream up to the diffuser entrance, downstream of which the variation is approximately linear. Separation in the diffuser just downstream of $x/c = 0.95$ in the CFD simulation accounts for the slightly erroneous value at $x/c = 1.0$ (see velocity profiles of Figure 5.4).

From the centreline volume flow rate distribution, it would not be unreasonable to assume a linear rate of change of volume-flow rate at the centreline of the model underside upstream of the diffuser, and a constant rate of change in the diffuser itself. The minimum volume-flow rate occurs at $x = x_c$ in Figure 5.13. An illustration of this lateral flow leakage distribution, termed the ‘added volume-flow rate’, $\frac{\partial Q}{\partial x}$ is shown in Figure 5.14.

The equation for the mass flow rate through the diffuser at the centreline becomes

$$Q(x) = Q_0 + \int_{x_0}^{x} \frac{\partial Q}{\partial x} dx$$
Figure 5.13: Volume flow rate per unit span calculated at centreline from CFD results. $\theta = 12^\circ$, $h/c = 0.060$.

![Graph showing volume flow rate per unit span](image)

Figure 5.14: Assumed ‘lateral volume-flow rate’ distribution, $\frac{\partial Q}{\partial x}$.

where

$$\frac{\partial Q}{\partial x} = ax/c + b \quad 0 < x/c < x_d/c$$

$$= ax_d/c + b \quad x_d/c < x/c < 1.0$$

$$= 0 \quad x/c = x_c/c$$

This modified version of the one-dimensional diffuser theory, to be termed the
'quasi-one-dimensional' theory, requires two boundary conditions, effectively defining the constants $a$ and $Q_0$. These conditions are taken from the conditions ($C_p$ and $h$) at the most upstream and most downstream pressure tappings, i.e. at $x/c = 0.28$ and 0.98. The pressure coefficient is then recovered from the following equation.

$$C_p = 1 - \left( \frac{Q}{U_\infty h(x)} \right)^2$$

For the full formulation, see Appendix D. A comparison of the results from the one-dimensional and quasi-one-dimensional models are shown in Figure 5.15. In this case $x/c$ is set equal to 0.4 to give better agreement with the experimental result.

![Figure 5.15: Comparison of theoretical and experimental pressure distributions. $\theta = 10^\circ$, $h/c = 0.02$ and 0.04](image)

The quasi-one-dimensional theory is a heavily simplified formulation for what is in reality a far more complex problem, depending on ground clearance, diffuser angle and boundary layer growth. However, it illustrates how the leakage from the sides of the model influence the pressure distribution, in particular by reducing the pressure rise through the diffuser, and creating the curvature in the pressure distribution on the flat underbody visible in the majority of the experimental results.
5.5.2 Influence of the Nose Vortex

In an attempt to avoid the nose vortex, a second nose was made for the model with a greater radius on the lower outer edge to try and suppress separation. Although the vortex could not be entirely eliminated, the new nose did result in a much weaker vortex. Total pressure contours in the wake of the diffuser with the two different noses are shown in Figure 5.16.

![Total pressure contours in the wake of the diffuser model with two different noses at two different ground clearances.](image)

Figure 5.16: Comparison of total head in wake of diffuser model with two different noses at two different ground clearances. $\theta = 7^\circ$, $h/c = 0.014$ and 0.080.

These total head data alone do not give a quantitative evaluation of the relative
nose vortex strengths, but they do show qualitatively that the nose vortex is weaker with the new, blunter nose. As mentioned before, and highlighted in Figure 5.16, the vortex strength also diminishes with increased ground clearance.

Model lift and surface pressure measurements were taken on the model for a range of ground clearances either side of the optimum at a diffuser ramp angle of $\theta = 12^\circ$ with both of the noses to give an indication of how the nose vortex is likely to influence the model behaviour. Figure 5.17 shows a comparison of the downforce coefficient variation with ground clearance for $\theta = 12^\circ$ for the two noses.

![Graph showing variation in downforce coefficient with ground clearance for two noses.](image)

Figure 5.17: Variation in downforce coefficient with ground clearance for the diffuser model two different noses. $\theta = 12^\circ$.

The effect of the nose vortex appears to be rather small with a small increase in peak downforce, and a slight reduction in optimum ground clearance. Figure 5.18 shows centreline pressure distributions for the same diffuser angle at ground clearances of $h/c = 0.040$ and $0.080$ for the two noses, showing that over the majority of the pressure tappings the pressures with the old (less-blunt) nose are lower. This seems to stem from the greater pressure recovery through the diffuser with the old nose.

This does not explain, however, why at the lower ground clearances there is
more downforce generated by the new, blunter nose. Looking at the most upstream pressure tappings, at the lower ground clearance, the pressure with the new nose is slightly lower than that with the old nose. This suggests that the pressures further upstream with the new nose are lower than with the old nose. There are indications of the same trend for the larger ground clearance as well. It is probably these lower pressures near the nose which make the downforce with the new nose greater at low ground clearances. If this was the case, then the pitching moment with the new nose would be more positive, i.e. more nose-down because of the greater proportion of downforce generated at the nose. Figure 5.19 shows a comparison of the variation in pitching moment with ground clearance for the diffuser model with the two noses.

The pitching moment curves show an almost constant positive offset between the data for the old and new noses, which would be in agreement with the hypotheses put forward above. The stronger vortex of the old nose, together with its image in the ground plane, will tend to induce an outboard component of flow at the diffuser sides. This will, as discussed previously, reduce the velocity and increase the pressure on the flat underbody, whilst increasing the pressure rise through the diffuser. At
the nose, where the influence of the diffuser is less, the net result is a lower pressure for the new nose, where the nose vortex is weaker. Further downstream where the diffuser has greater effect on the underbody flow, the pressure is lower for the older nose and stronger nose vortex.

5.5.3 Influence of the Diffuser Vortices

The diffuser vortices produce strong three-dimensional variations in the spanwise pressure distribution on the underbody. Spanwise pressure distributions for the optimum transition-fixed configuration ($\theta = 12^\circ$, $h/c = 0.060$) at a number of streamwise locations are shown in Figure 5.20. At the most upstream position, $x/c = 0.28$, there is a small rise in pressure towards the edge of the model at $y/b = 0.5$, as one would expect, for example for a finite aspect ratio wing. At $x/c = 0.52$, the pressure at the most outboard tapping is not as great as the trend towards the edge should suggest, indicating a possible mechanism which reduces the pressure locally. At $x/c = 0.72$ there is a definite drop in pressure towards
$y/b = 0.5$, which is greatest at $x/c = 0.84$, just inside the diffuser. This low pressure peak reduces in size and moves inboard at $x/c = 0.92$.

![Figure 5.20: Centreline underbody surface pressure distributions with and without fixed transition. $\theta = 12^\circ$, $h/c = 0.060$.](image)

The low pressure peak in the spanwise pressure distributions is caused by the diffuser vortex which rolls up from the shear layer separating from the model sides due to the inboard flow leakage. Similar low pressure peaks were reported by Senior and Zhang (2001) but only on their diffuser ramp. In the results of the present investigation, there are signs that the vortex could first appear upstream of $x/c = 0.52$ as the inboard flow leakage separates from the sharp model edges. Figure 5.21 shows the spanwise pressure distributions at $x/c = 0.52$ and $x/c = 0.72$, which highlight an interesting feature of the flow behaviour.

At the lowest ground clearance of $h/c = 0.04$, at $x/c = 0.52$ there is no sign of the low pressure at the model edges signifying inboard flow leakage, although there is some sign of this at $h/c = 0.06$ and 0.08. However, further downstream at $x/c = 0.72$ the low pressure at the model edges is greater at the lowest ground clearances, signifying a stronger vortex. This would suggest that the reversal from outboard to inboard flow leakage occurs further downstream at lower ground clearance, whilst the
strength of the streamwise vortex forming at the diffuser edges is ultimately stronger at the lower ground clearances, a phenomenon also found by George and Donis (1983). Looking at the chordwise pressure distributions for the same configurations, shown in Figure 5.22, the 'flatter' pressure distribution upstream of the pressure minimum at the lowest ground clearance also suggests a later initiation of the inboard flow leakage. This is not a surprising result since the outboard leakage is stronger at lower ground clearances, and so likely to persist for a greater proportion of the model length.

5.6 Key Findings

The key findings regarding the aerodynamic behaviour of a diffuser-equipped bluff-body in ground effect may be summarised as follows:

1. The underbody and diffuser flow is highly sensitive to the state of the boundary layer, since the boundary layer growth in the diffuser depends on the thickness at inlet. The greater the growth in the diffuser, the smaller the effective area
ratio. For this reason, the underbody flow is more dependant on Reynolds number than for a wing far from the ground. Forcing transition can be severely detrimental to the diffuser performance since it results in a thicker boundary layer.

2. Since the sides of the model are not sealed to the ground, the flow is highly three-dimensional, with outboard leakage from the underbody near the nose, and inboard leakage further back towards the diffuser. Outboard leakage on the flat underbody reduces the flow velocity at the centreline, increasing the pressure. The reverse is true for inboard leakage. Inboard leakage in the diffuser increases the volume flow rate through the diffuser such that the deceleration of flow through the diffuser is reduced. This results in a lower pressure rise through the diffuser, and a greater pressure at inlet.

3. The so-called ‘nose vortex’ forms when the outboard and upward flow approaching the nose edges separates from the lower side-edge of the nose. The strength of this vortex grows with reducing ground clearance. At the lowest ground clearances it runs along the sides of the model, and as the ground
clearance is increased it is pulled beneath the model edges by the inboard flow leakage and into the diffuser. The effect of the nose vortex is to reduce the inboard leakage of flow, which tends to reduce the pressure on the flat underbody, whilst increasing the diffuser effectiveness. The increased effectiveness of the diffuser also results in lower pressures over a large proportion of the underbody. These effects increase the nose-up pitching moment.

4. The flow in the diffuser near the endplates is dominated by streamwise vortices forming from separation of inboard leakage flow from the model sides. The vortex roll-up appears to begin upstream of the diffuser entrance. The strength of these vortices increases with increasing diffuser ramp angle and reduced ground clearance. Locally they induce a low pressure ‘track’ on the flat underbody and particularly on the diffuser ramp over which they pass.
Chapter 6

Front Wing Wake Structure and Behaviour

6.1 Introduction

In order to understand the interaction mechanisms between a front wing and a downstream body, it is crucial that the structure and behaviour of the wing's wake are well understood. The aim of the current chapter is to identify the various key components of the front wing wake, their associated flow fields and behaviours, both qualitatively and quantitatively. This will provide a base from which investigation of the interaction mechanisms may proceed.

Within this chapter, the study of the front wing wake is divided into the following sections:

6.2 Wake Structure

The wake of the front wing model in a number of configurations was mapped by total pressure surveys on cross-flow planes behind the wing. This data is presented to give a qualitative description of the flow field in the wake of the wing, and to identify its main components.

6.3 Low Energy Wake
One of the key components of the wake is the low energy flow, or momentum deficit behind the wing due to the viscous interaction between the wing surface and the flow. Total head measurements are used to describe quantitatively the momentum deficit and vertical translation of the wake at the wing centreline. This momentum deficit is compared to that of a cylindrical rod, which are used in later experiments to investigate the effect of that component of the wing wake.

6.4 Trailing Vortex Behaviour in Ground Effect

Another of the major components of the front wing wake is the trailing vortices. The interaction between these vortices and the ground is rather complicated and sometimes destructive, promoting their breakdown. The behaviours of the trailing vortices from both horizontal and vertical front wing models are investigated using smoke-flow visualisation, total head surveys, PIV, and hot-wire measurements, including both time-averaged and instantaneous data. This is used to build an understanding of the interaction between the vortices and the ground plane. Some data is also presented with a stationary ground plane as this is a case of particular interest, relevant to the interaction between a vortex and a stationary boundary, such as the surface of the downstream body to be used later on in the study.

6.2 Wake Structure

Total head measurements were made in the wake of the horizontal front wing model to gain a qualitative appreciation of the wake structure, and a quantitative definition of the total head loss. As described in Section 3.4.2, a traverse mechanism was used with a 2 s settling time between successive data acquisition points, and a 2 s scanning time. Probe movement would take an average of approximately 1 second, giving a total of 5 seconds per data point. Gaining adequate grid resolution over the required areas would involve many hundreds or thousands of data points, resulting in run times of the order of several hours. Minimising the run times was
Data is therefore limited to small areas (up to 20% of the wing span) inboard of the endplates, to capture the tip vortices, and linear traverses to quantify the head loss at the centreline at the position of the rear model nose.

Total head measurements were made over four cross-stream planes downstream of the wing at $x/c = 0.0, 1.0, 2.0$ and 3.0 measured from the endplate trailing edge, with the model at either $5^\circ$ or $10^\circ$ incidence, and ground clearances of $h/c = 0.15, 0.25, 0.50$ and 0.75. Contours of total head coefficients are plotted for all of these configurations in Figures 6.1 to 6.4. These plots are provided to give a qualitative description of the wake only, and a scale for the contour levels is not shown. However, the scale is such that the darkest blue represents total head coefficients below -0.2, whilst red represents coefficients up to 0.9. Total head coefficients above 0.9 are omitted from the plot to improve visualisation the wake shape. The lower rear right-hand (facing upstream) endplate edge is at $x = 0, y = 0, z = 0$.

![Figure 6.1: Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.15$.](image)

These figures show the wake to comprise, on each side of the centreline, a strong vortex shed from the endplate lower edge, and the momentum deficit in the wake of the wing and endplate. The low total head in a diagonal line for $y/c < 0$ in some of the plots is from the wake of the wing support. The vortices move inboard and upward under the velocities induced by their counterpart at the other end of the wing span, and image in the ground plane. The wake moves upward near the
Figure 6.2: Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.25$.

Figure 6.3: Total head coefficients in wake of full-span front wing. $\alpha = 5^\circ$ (left) and $\alpha = 10^\circ$ (right), $h/c = 0.50$.

centreline, wrapping around the vortex further outboard.

At the lowest ground clearance of $h/c = 0.15$ the wake of the wing is much thicker than at the other ground clearances, and does not rise as much as it convects downstream. The vortices are also much less well defined. There are two possible reasons for this. The first is strong vortex wandering under the influence of secondary vortices shed from the cross-flow boundary layer on the ground. Since these results are time-averaged, the vortex appears bigger and less intense, with greater total head at the centre, similar to the results of Devenport et al. (1996) for a vortex far from the ground. The second is that ground proximity is causing the vortex to
break down, as found by Moseley (1999).

At $h/c = 0.25$ the vortices are more distinctly defined, and as the ground clearance is increased further they remain distinct but decrease in intensity, suggesting a reduction in downforce generated by the wing. Meanwhile, the amount that the wake lifts increases with increasing ground clearance, due to the diminishing influence of the ground plane, whose image vortex system tends to reduce the upwash in the wake.

There are three main components of the front wing wake which are likely to affect a downstream body: Upwash; wake momentum deficit; and trailing vortices. The upwash and momentum deficit in the wake could be expected to reduce the downforce generated by a body downstream of it, whilst the influence of the trailing vortices is likely to be more complex, depending on their path over it. The behaviour of wake momentum deficit and trailing vortices is discussed in the following two sections.

### 6.3 Low Energy Wake

Total head measurements were taken along a vertical line at the centre of the horizontal wing, one chord length downstream of its trailing edge, at a number of ground
clearances and incidences, in order to quantify the momentum deficit at the centreline. A typical set of distributions are shown for the lowest ground clearance of $h/c = 0.10$ in Figure 6.5, with $z/c$ measured from the ground plane as a fraction of the chord.

![Graph showing total head coefficients distribution](image)

**Figure 6.5**: Total head coefficients distribution along a vertical line at the centre of the wing wake for $h/c = 0.10$ at a number of incidences.

For the purposes of reducing the results to an easily comparable and meaningful format, a momentum deficit and deficit centre are defined. Since the total head distributions are at the centreline, and not through any vortices, it is assumed that the static pressure along the traverse line is constant and equal to the pressure far upstream. Any changes in total head along the line are therefore taken to be purely due to changes in velocity, from which the momentum may be calculated. Figure 6.6 shows the static pressure coefficient distribution at $x/c = 1.0$ behind the wing from the 2D CFD results presented in Chapter 5. This data shows the assumption of constant static pressure at this location not to be too inaccurate, with the maximum deviation being approximately 2% of the dynamic pressure.

Firstly, the momentum deficit per unit span, $\Delta G$, is defined as
Figure 6.6: Static pressure coefficients on vertical line one chord length behind wing at 0° incidence from 2D CFD.

\[
\Delta G = \int_{-\infty}^{0} \rho (U_{\infty}^2 - U^2) \, dz
\]

Assuming the static pressure to be constant and equal to that far upstream of the body gives the following relation between the total pressure coefficient, \( C_{p0} \) and velocity relative to that far upstream

\[
C_{p0} = \left( \frac{U}{U_{\infty}} \right)^2
\]

which is then used to formulate an equation for the momentum deficit per unit span in terms of the total pressure coefficient

\[
\Delta G = \rho U_{\infty}^2 \int_{-\infty}^{0} (1 - C_{p0}) \, dz
\]

This is non-dimensionalised by \( \rho U_{\infty}^2 c \), to give the momentum deficit coefficient,

\[
C_{\Delta G} = \frac{1}{c} \int_{-\infty}^{0} (1 - C_{p0}) \, dz
\]

The momentum deficit centre, \( z_{\Delta G} \), which is effectively a measure of the height above the ground of the centre of the wake, is found by taking moments of the momentum deficit about the ground (\( z = 0 \)).
\[ z_{\Delta G} = \frac{\rho U^2}{\Delta G} \int_{-\infty}^{0} (1 - C_{p0}) zdz \]

The momentum deficit coefficient results are shown in Figure 6.7 below, whilst Figure 6.8 shows the non-dimensional wake vertical location relative to the quarter-chord. As would be expected, the momentum deficit grows with increasing incidence of the wing and reduced ground clearance, owing to an increased adverse pressure gradient on the lower surface. At the lowest ground clearances, there is a reduction in momentum deficit between the incidences of 0° and 2°. The reason for this is boundary layer growth on the moving floor, as seen in Figure 6.5. At \( h/c = 0.10 \) for example, at an incidence of 0° the wake of the wing has merged with the ground boundary layer. Meanwhile at an incidence of 2° there is a region of low momentum deficit between the ground and wing. It is likely that at the greater incidence more flow is ‘pumped’ underneath the wing, resulting in a lower momentum deficit.

![Figure 6.7: Momentum deficit coefficient in wake of front wing model.](image-url)

In general, the vertical location of the wake relative to the wing quarter-chord rises with increasing ground clearance and increasing incidence. Increasing the incidence increases the upwash in the wake of the wing, resulting in a greater vertical
displacement of the wake. Meanwhile, it also increases the boundary layer thickness on the lower surface of the wing, which would tend to reduce the measured height of the wake, as in the case of the larger incidences. The net result depends on the ground clearance, and the pressure gradient on the suction surface: a greater adverse pressure gradient increasing the boundary layer thickness. Increasing the ground clearance of the wing reduces the influence of the ground, and its effect of reducing the upwash in the wake. It also reduces the pressure gradient beneath the wing towards the trailing edge, resulting in a thinner boundary layer on the suction surface and ground. Both of these effects increase the height of the wake above the ground.

In later experiments, in an attempt to separate the various effects of the wake, the horizontal wing was replaced by a cylindrical rod. The purpose of the rod is to create the momentum deficit in the wake of the wing without the associated trailing vortices or upwash. Similar total head surveys were made along the centreline in the wake of a number of rods of differing diameters, and the same calculations performed on the measurements. Unsurprisingly, the vertical location of the wake relative to the
rod does not vary with rod diameter or ground clearance by a measurable amount. The momentum deficit is also virtually constant with ground clearance, except at very low ground clearances, where the wake growth is such that it reaches the ground level. The momentum deficit was found to vary linearly with diameter (here non-dimensionalised by the front wing chord for comparison, $\phi/c$) over the range tested, indicating no change in flow regime over this range of Reynolds number. The data is shown in Figure 6.9, with a linear trendline fitted to the results.

![Figure 6.9: Momentum deficit coefficient for wake of cylindrical rod.](image)

### 6.4 Trailing Vortex Behaviour in Ground Effect

#### 6.4.1 Preliminary Results for a Vortex in Ground Effect

In the total head traverse plots of Figure 6.1 there is evidence that the trailing vortex is either wandering, or has broken down, because of its less well defined structure. To investigate the behaviour of these vortices near the ground a video recording was made of smoke-flow in the wake of one of the wing models described in Section 3.2.1, using a laser sheet to illuminate a cross-flow plane. One of the most striking features
of the visualisations was a breakdown of the vortices for certain configurations which occurred more readily with the moving floor than with the floor stationary. Two typical images from this video showing the left hand (facing upstream) vortex are shown in Figure 6.10. With the road off, the vortex is clearly visible, with a smoke-free centre from which the smoke has been centrifuged due to the strong coherent core. There is no such smoke-free centre with the road moving, which is taken to indicate the loss of the coherent vortex structure, and therefore probable breakdown.

![Road Off](image1.png) ![Road On](image2.png)

Figure 6.10: Smoke flow visualisation of lower endplate vortex with and without the moving road, \(h/c = 0.40\).

Further smoke-flow visualisations were carried out to identify where breakdown occurred for different ground clearances and incidences of the wing with and without the road moving. These showed that breakdown is more likely for reduced ground clearance and increased incidence. These two factors increase the loading on the wing, which should strengthen the vortex, and as discussed in Section 2.3.2.3, a stronger vortex having greater swirl angle is more prone to breakdown. However, more careful examination of the results shows that this does not give a completely satisfactory explanation of the phenomenon. Figure 6.11 shows plots of vortex breakdown location in number of chords downstream of the trailing edge, and the lift curve slopes for the wing at a ground clearance of \(h/c = 0.40\). This suggests that the change in downforce with and without the moving floor is not sufficient to explain the earlier bursting of the vortices with the moving floor. There must therefore be some other mechanism causing vortex breakdown to occur more readily.
with the moving ground plane.

Figure 6.11: Estimated vortex breakdown location behind single wing of tandem pair estimated from smoke-flow visualisation and lift curve slope at \( h/c = 0.40 \).

### 6.4.2 Front Wing Vortex Experiments

In addition to the total pressure traverse measurements taken in the wake of the front wing model (see Section 3.2.4) shown in Figures 6.1 to 6.4, further measurements were taken over small areas, 10mm square (0.1c) with grid spacings of 2mm (0.02c) covering the vortex cores over a greater number of \( x \)-planes. The aim of these measurements was to track the vortex, and it is believed that this enables a prediction of the time-averaged trajectory to an accuracy of approximately \( \Delta y/c = \Delta z/c = 0.01 \). The resulting trajectories of the vortices for the wing at 5° incidence and a number of ground clearances are shown in Figure 6.12.

Using the same method as Barker and Crow (1977) the circulation based on the vortex trajectories may be calculated. Firstly, consider the basic form of the vortex system of the front wing model in ground effect, as shown in Figure 6.13. The total head measurements showed no trace of a vortex shed from the top endplate edge,
so only the lower endplate vortex is considered. To further simplify the problem, the upwash generated by the bound vortex will be ignored as its effect is likely to be small relative to that generated by the other vortices in the system.

For a simple analysis, the system of 4 endplate vortices (two real, two image) are
considered in the same way as by Lamb (1932), who calculated the velocity induced on one vortex by the other three. Considering the motion induced on the right hand wing endplate vortex in Figure 6.13, the equations of motion are

\[ \dot{y} = -\frac{\Gamma y^2}{4\pi y(h^2 + y^2)} \quad \dot{h} = -\frac{\Gamma h^2}{4\pi h (y^2 + h^2)} \]

Combining these two equations to form an equation for the vortex speed, \( u = \sqrt{\dot{y}^2 + \dot{z}^2} \), the vortex circulation may be solved, in the same way as that of Barker and Crow (1977):

\[ \Gamma = \frac{4\pi u h y(h^2 + y^2)}{\sqrt{h^6 + y^6}} \]

The results from this calculation carried out on the vortex trajectories shown in Figure 6.12, with the wing at an incidence of 5°, are shown in Figure 6.14. The increasing calculated circulation with ground clearance is most probably due to the fact that the bound circulation has not been taken into account. At low ground clearances, where it is close to its image in the ground plane which tends to act against it, it is less important. As the ground clearance is increased, however, the image bound vortex becomes less dominant, and the assumption that it can be neglected becomes less valid.

Similar to the findings of Barker and Crow (1977) there are considerable fluctuations in the calculated circulation, which would not be accounted for by consideration
of the bound vorticity. Since the trajectories are from time-averaged and repeatable data, this is not due to any unsteady vortex wandering effects. This would therefore suggest that there is a mechanism at work outside the considerations of the model, which is most likely the production of secondary vorticity. As discussed in Section 2.4.2.2 this is known to produce complex flow structures, inducing motion of the main vortex which is stronger than that induced by the other system vortices.

6.4.3 Vertical Wing Vortex Experiments

The behaviour of vortices in ground effect was further examined using the vertical wing models described in Section 3.2.5. Since these consist of simple wings without endplates, the wake is much less complex than for the wing with endplates. Furthermore, the wake of the wing itself interacts far less with the ground plane than for a horizontal wing in ground effect, and is so less likely to change its influence on the trailing vortex at different ground clearances.

Total head traverses were carried out with the vertical wings at an incidence of 15° and a number of clearances between the wing tip and the ground plane, namely
5, 10, 15, 20 and 80mm \((h/c = 0.07, 0.14, 0.21 \text{ and } 0.29)\), 180mm downstream of the trailing edge \((x/c = 2.57)\). These total head plots are shown in Figure 6.15. \(y\) is measured from the wing quarter-chord point, and \(z\) from the ground plane.

![Figure 6.15: Total head contours in wake of vertical wing at \(x/c = 2.57\) at an incidence of \(15^\circ\) and a number of ground clearances.](image)

Again, the vortex appears to lose its structure at low ground clearances, the well defined region of low total head at the vortex centre becoming more diffuse as the ground clearance is reduced. Since the total pressure measurements are time-averaged, this diffuse appearance could either be brought about through wandering of the vortex or through vortex breakdown. In order to identify the true state of the vortex instantaneous velocity data is required. To this end, DPIV measurements were made in the wake of the vertical wings. Figure 6.16 shows velocity vectors and \(x\)-component of vorticity contours which has been averaged over between 15 and 20 data sets for the four lowest ground clearances. The perspective error has also been subtracted from this data. In all four plots there is a small region of high vorticity which has been circled. This was due to spurious vectors in that area resulting from light being reflected from some small part of the vertical wing. These data sets were taken with the TSI 2048 × 2048 pixel camera, from which 128 × 128 vectors were
calculated. To increase clarity every other vector has been omitted. Averaging has been used to fill in missing vectors, which for the averaged plots accounts for less than 1%, to reduce spurious values in the calculated x-vorticity field.

Figure 6.16: Averaged PIV data at x/c=2.0 behind vertical wings at α = 15° at a number of tip ground clearances. Spanwise separation b/c=4.29 (other wing to left, and arranged as mirror image).

At a ground clearance of h/c = 0.07, weak circulatory motion is present, identifying a weak and diffuse form of the vortex. At h/c = 0.14, the vortex appears slightly stronger, but still lacking a concentrated core. However, a concentrated core does appear at h/c=0.21 and 0.29, increasing in strength with increased ground clearances. Of particular interest is the area of positive vorticity just inboard (on the up-going side) of the vortices at the two larger ground clearance. At h/c = 0.21 the positive vorticity appears in a curved arc around the lower inboard quadrant of the
vortex. At $x/c = 0.29$ the positive vorticity is confined to a much smaller region next to the ground.

A computer program was written to help identify the vortex cores. A method was used which defines the vortex core centre based on the rate of change of circulation with radius. Looking back at the definition of a Lamb-Oseen Vortex, the vortex radius is the point where $\Gamma/r$ is a maximum. The program first chooses a number of points in the field based on peak vorticity. It then moves around each point, calculating the circulation around a number of circles of increasing radius, searching for the point from which the peak $\Gamma/r$ is greatest. That point is defined as the vortex centre, and the radius for maximum $\Gamma/r$ is the vortex core radius.

The following figures show instantaneous $x$-vorticity and velocity vector plots at four random instantaneous times for each of the four wing tip ground clearances. Since abrupt changes in velocity can result in large values of the vorticity, again interpolation has been used to fill in gaps in the vector fields. This accounts for approximately 5% of the vectors in these instantaneous plots. For each plot, 10 vortex centres were calculated, and the resulting positions are shown as black circles. These tend to all converge on the same point where there is a single concentrated vortex core.

Figure 6.17 shows the data for a vertical wing tip ground clearance of $h/c = 0.07$ (5mm). The instantaneous plots show a behaviour which is markedly different from the averaged plots-although there is an identifiable net rotation in all of the plots, there is no clear vortical structure, with a number of patches of both positive and negative vorticity spread over a rather large area. The vortex finding routine fails to find a single vortex core. It would therefore appear that the vortex has broken down, as a single coherent core is no longer identifiable. As this data is only available in the cross-stream plane, identifying the structure of the breakdown is not possible. Moseley (1999) showed his Formula One style front wing vortices to breakdown in ground effect in either the spiral or bubble modes, the spiral mode having a high winding angle. If this is the case here then the vortex would be passing through the cross-flow plane at an angle rather than normal to it. The contours of $x$-vorticity plotted here are therefore not as meaningful as for when the vortex axis
is approximately normal to the plane.

Figure 6.17: Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.07$.

Figures 6.18 to 6.20 show similar instantaneous velocity vector and $x$-vorticity contour plots for the vertical wing at tip ground clearances of $h/c = 0.14$, 0.21 and 0.29. As the tip ground clearance is increased the vortex becomes more coherent. At $h/c = 0.14$, the vortex is more clearly identifiable than at $h/c = 0.07$, but still lacks structure, being more coherent at some times than others. Again, there are a number of patches of positive and negative vorticity, and the vortex finding routine does not succeed in converging on a single point for the vortex core.

At $h/c = 0.21$ there is a well defined vortex core, characterised by a single small area of negative vorticity. Positive secondary vorticity can now be clearly seen to be lifted up from the ground plane on the up-going side of the vortex. In the averaged
Figure 6.18: Four instantaneous PIV vorticity and velocity vector fields at \( x/c = 2.0 \) behind the vertical wings at \( \alpha = 15^\circ \) and \( h/c = 0.14 \).

plots, this appeared as an arc of positive vorticity, whilst in the instantaneous plots, discrete patches of positive vorticity can clearly be seen to be lifted up from the ground plane, and orbit around the primary vortex. However, it seems that these patches, or secondary vortices, lose strength as they orbit around the main vortex. Since they are extremely close to the main vortex, it could well be the case that they merge with it, which, since they are much weaker, would involve the positive vorticity being absorbed by the larger vortex. This would degrade the primary vortex strength with distance downstream.

At \( h/c = 0.29 \) there is much weaker interaction between the primary and secondary vorticity, and it appears that the production of secondary vorticity is not strong enough to cause it to separate from the ground plane into discrete vortices.
Figure 6.19: Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.21$.

This results in a more stable position of the primary vortex, and presumably would reduce the weakening of the primary vortex by its merging with the secondary vorticity.

The PIV data presented here suggests a strong interaction between the primary and secondary vortices. This is the most likely reason for the breakdown of the vortices at low tip ground clearances. Assuming the vortex motion to be unsteady, when the vortex is close to the ground, it produces secondary vorticity at the ground plane, whose induced velocity at the primary vortex causes it to rise and move inboard. As it rises, its interaction with the ground plane diminishes, reducing the rate of production of secondary vorticity, thus allowing it to return to its original position closer to the ground. This would tend to excite a spiral motion of the three-
Figure 6.20: Four instantaneous PIV vorticity and velocity vector fields at $x/c = 2.0$ behind the vertical wings at $\alpha = 15^\circ$ and $h/c = 0.29$.

dimensional vortex, which may promote the spiral mode of breakdown. Indeed, Moseley (1999) recorded spiral angles in his spiral mode of vortex breakdown in ground effect greater that those recorded by other researches studying breakdown away from a solid boundary.

Due to the limited time for which the PIV equipment was made available, it was not possible to gather enough data to determine the exact mechanism by which the interaction occurs. The available data however suggests that it is a highly unsteady and complex phenomenon. The ideas put forward above about the interaction between the primary vortex and secondary vorticity suggest that there may be a quasi-periodic motion of the vortex system. In order to study this, a hot-wire probe was placed at the vortex centre as measured through the time-averaged total
head data of Figure 6.15, with a set also taken at $h/c = 1.14$ to give a 'far from the ground' case.

Power spectra for the hot-wire data are shown in Figure 6.21. The hot wire was placed horizontally and normal to the free stream at the time-averaged vortex centres at a distance downstream of the trailing edge of $x/c = 2.57$. The free-stream velocity for this dataset was 20 m/s, requiring a sampling frequency of 40kHz, at which $2^{24} \approx 16.8 \times 10^6$ samples were taken. A $-5/3$ power law slope is also shown on the plot, showing that between the frequencies of $10^3$ and $10^4$Hz, the PSD is approximately following the Kolmogorov hypothesis for the energy cascade, and so the energy in this region is generated by isotropic turbulence.

![Figure 6.21: Power spectral density from hot-wire measurements at $x/c = 2.57$ in vertical wing vortex core at a number of ground clearances.](image)

The forms of the power spectra are all very different, the most interesting being at the ground clearance of $h/c = 0.21$, where there is a definite peak in the spectrum at a frequency of approximately 440 Hz which is not reproduced at the other ground clearances. Visualisation of this peak is aided by plotting the power spectral density multiplied by the frequency, to give a 'normalised power'. This increases the reso-
olution of the power contained in the smaller scales of the motion. The normalised power spectral densities are shown in Figure 6.22.

![Figure 6.22: Normalised power spectral density from the hot-wire measurements at \( x/c = 2.57 \) in vertical wing vortex core at a number of ground clearances.](image)

There are a number of interesting features in Figures 6.21 and 6.22 which have significant implications for the vortex behaviour in ground effect. At large ground clearance \( (h/c = 1.14) \) there is very little energy contained in the signal compared with the other signals. The energy at very low frequencies is most probably generated by the vortex wandering, the strong peak for the ground clearance of \( h/c = 0.21 \) being due to the strong interaction with the secondary vorticity, as seen in the PIV data. At the large ground clearances, this energy is significantly reduced, indicating much less wandering of the vortex. There is less energy contained in the inertial subrange of the high frequency turbulent fluctuations as well. It is known that the large strain rates in the core of a vortex are relaminarising (Devenport et al., 1996), which would explain this. With greater wandering of the vortex, the probe is exposed to more of the turbulent flow outside of the vortex core, and the high-frequency turbulent fluctuations become more apparent in the spectra.
It appears that at \( h/c = 0.14 \) there is a broadening of the peak that exists at \( h/c = 0.21 \), presumably due to the breakdown of the vortex into a range of motion scales. At \( h/c = 0.07 \) it also appears that this peak has broadened, but reduced in magnitude considerably. A weak secondary peak is just visible in the normalised spectrum at \( h/c = 0.21 \) at approximately double the frequency of the original peak. This may be explained by considering the motion of the vortex and its effect on the signal produced by the hot-wire.

Figure 6.23 shows the positions of the vortex relative to its time-averaged location for \( h/c = 0.21 \) from 20 instantaneous PIV data sets, of which 4 were shown in Figure 6.19. Although the vortex trajectory may not be inferred from this data, it gives an indication of the vortex motion about the mean location. For a rough approximation of the effects of the motion on the hot wire data, an elliptical trajectory of the vortex in the cross-flow plane is assumed.

![Figure 6.23: Vortex centre locations at 20 instantaneous points in time as taken from PIV data for the vertical wing vortex at a tip ground clearance of \( h/c = 0.21 \).](image)

Figure 6.24 shows the variation in mean velocity normal to the wire at the vortex core, and RMS velocity fluctuation with wing tip ground clearance. It can be seen
that at the largest ground clearance of \( h/c = 1.14 \) there is a velocity excess in the core, with a core axial velocity above that of the freestream velocity. The power spectra suggest that at this ground clearance vortex wandering is minimal, so it is unlikely that this is due to the probe being exposed to the tangential component of velocity as the vortex moves. Reducing the tip ground clearance reduces the mean velocity sensed by the probe, to the minimum at \( h/c = 0.21 \). This is where the vortex wandering is at a maximum without it breaking down. If the vortex axis is perpendicular to the hot wire, then the velocity measured by the probe would be increased by the added tangential component of velocity. However, this is not the case and the velocity at the probe is reduced considerably. If the vortex is spiralling, as has been suggested, then the axial velocity at the axis of the spiral may be reduced, which in the spiral form of vortex breakdown ultimately results in stagnation of the flow.

Figure 6.24: Variation in mean velocity normal to hot wire and RMS velocity fluctuation with vertical wing tip ground clearances at time-averaged core centre location.

From the PIV data, the vortex has an average circulation of 0.32 \( m^2/s \) and a core radius of 5.8mm. From all of the data presented here, the vortex may be modelled
as having a Gaussian distribution of axial velocity, with the minimum being 9.8 m/s at the centre, and a Lamb-Oseen distribution of tangential velocity in the vortex. An elliptical path is assumed in the cross-stream plane with major and minor axes equivalent to the limits of vertical and horizontal motion from the PIV data shown in Figure 6.23. The peak in the power spectrum is at 440 Hz for \( h/c = 0.21 \). A single cycle of the elliptic motion of the vortex could be expected to produce two cycles of velocity fluctuation, since the probe would pass through two tangential velocity peaks with each cycle. The elliptic motion frequency is therefore set to 220 Hz. Figure 6.25 shows the simulated \( y \) location, axial and tangential velocities, and velocity component normal to the hot wire.

Figure 6.25: Simulated vortex \( y \) location, axial and tangential components of velocity at the hot wire location, and component normal to the wire.

Running the same power spectral density algorithm on the velocity data produced by this simulation results in the spectra shown in Figure 6.26. This demon-
strates that an elliptical motion of the vortex will in fact produce a number of peaks in the power spectrum: the first at twice the frequency of elliptical motion, and the second at double the frequency of the first. This explains the two peaks seen in the power spectra, and would suggest that the spiralling frequency is half that of the first and strongest spike in the spectrum.

![Simulated power spectral density and normalised power spectral density.](image)

Figure 6.26: Simulated power spectral density and normalised power spectral density.

The dependency of this behaviour on free-stream velocity has been investigated by taking hot wire measurements at free stream velocities from 10 to 35 $m/s$ in steps of 5 $m/s$, all with a wing tip ground clearance of $h/c = 0.21$. The power spectral densities for these results are shown in Figure 6.27.

All of the spectra are of remarkably similar appearance, exhibiting the same double peaks in the spectra discussed above, with the second weaker peak appearing at approximately double the frequency of the first. This would suggest that the phenomenon is Reynolds number independent over the range of Reynolds numbers tested. The frequency of the first peak scales linearly with the velocity, as shown in
Figure 6.27: Power spectral density from hot wire measurements in vortex core for vertical wing tip ground clearance of $h/c = 0.21$ at a range of free stream velocities.

The plot of Figure 6.28, in which the frequency of the first peak is plotted against free stream velocity.

Moseley (1999) calculated the non-dimensional spiralling frequency for his vortex in ground effect, $f^* = fc/U_\infty$, where $f$ is the frequency of spiralling motion, and $c$ is the wing chord, after breakdown, finding values between 2.22 and 3.33. Assuming that the frequencies presented in Figure 6.28 are due to spiralling at a frequency equal to half that of the power spectra peak, this calculation yields an average value of $f^* = 0.79$, which is well below the post-breakdown values found by Moseley.

It is suggested, therefore, that as a vortex is brought into close proximity to the ground, it experiences a growing interaction with the secondary vorticity which it lifts up from the ground plane. As it does so, it produces secondary vortices which induce a motion of the vortex which is upward and towards the up-going side of it. As the vortex rises, the production of secondary vorticity falls, allowing the vortex to descend and move outboard again. This cycle produces a spiralling motion of the vortex. When the wavelength of the spiralling motion rises above a certain point,
vortex breakdown is initiated. This means that the persistence of a vortex of fixed strength is reduced if it is brought below a certain ground clearance.

The smoke visualisations of Figure 6.10 showed that the vortex is more susceptible to burst with the ground moving than with the ground stationary. It has now been established that the vortex breakdown is driven by the interaction between the vortex and secondary vorticity lifted up from the ground plane. It would therefore follow that, since the vortex is more stable with the stationary ground plane, less secondary vorticity is produced. Figures 6.29 to 6.34 show PIV data for the vertical wing vortex at a tip ground clearance of \( h/c = 0.21 \) with the moving and stationary ground planes at positions behind the wing trailing edge of \( x/c = 0.86, 1.71, \) and 2.57. The LaVision digital PIV system was used to collect this data with seeding provided by a smoke generator. Producing and positioning sufficient smoke proved to be a difficult task with this system, and subsequently there is a greater rate of data dropout. For this reason, no interpolation has been used to fill in missing vectors. This means that spurious results may be produced in the calculated \( x \)-vorticity field, especially in the vortex core. This becomes more apparent with
increasing streamwise distance as the smoke particles are flung from the centre of the vortex due to the radial acceleration there.

![Image of PIV data for moving ground plane with axes labeled x/c and y/c.](image)

Figure 6.29: PIV data for moving ground plane at $x/c = 0.86$ and tip ground clearance $h/c = 0.21$.

With the moving floor simulation, secondary vorticity can clearly be seen growing on the ground plane at $x/c = 0.86$, and lifting up from the ground plane at $x/c = 1.71$, rising further at $x/c = 2.57$. However, with the road stationary very little secondary vorticity is produced at the ground plane, and in fact in most of the plots, there is no identifiable secondary vorticity whatsoever.

Investigation of the reasons why secondary vorticity is produced in greater quantities with the moving ground plane than with the stationary ground plane is beyond the scope of this project. However, the results presented here would suggest that the vortex breakdown in close proximity to the ground is driven by the interaction of the primary vortex with the secondary vorticity lifted from the ground.
Figure 6.30: PIV data for moving ground plane at $x/c = 1.71$ and tip ground clearance $h/c = 0.21$. 
Figure 6.31: PIV data for moving ground plane at $x/c = 2.57$ and tip ground clearance $h/c = 0.21$. 
Figure 6.32: PIV data for stationary ground plane at $x/c = 0.86$ and tip ground clearance $h/c = 0.21$. 
Figure 6.33: PIV data for stationary ground plane at $x/c = 1.71$ and tip ground clearance $h/c = 0.21$. 
Figure 6.34: PIV data for stationary ground plane at $x/c = 2.57$ and tip ground clearance $h/c = 0.21$.  

6.5 Key Findings
For some experiments, two sets of vertical wings were used to generate systems of four trailing vortices. Front wings of Formula One cars have a number of appendages such as flaps, endplates and under-wing vortex generators, and so the behaviour of these vortex systems and their influence on the downstream body is of significant relevance. To investigate the behaviour of these systems, a number of sets of PIV measurements were made with the twin sets of vertical wings. The wings were all at the same tip ground clearance, and the vertical wings were arranged such that the sign of the vortices would be the same on each side of the centreline. The inner wings were set to an incidence of 10°, with a lateral spacing of $b_1/c=3.57$ (250mm), whilst the outer wings were at an incidence of 15°, and their spacing was varied. Figures 6.35 and 6.36 show the PIV $\zeta$-vorticity contours and velocity vectors for tip ground clearances of $h/c = 0.21$ and 0.29. The spacing between the inner and outer vertical wings was $\Delta b/c=1.07$ (75mm), 1.43 (100mm), and 1.79 (125mm).

Although not shown here, the data (50 sets were taken at each configuration) showed a considerably greater amount of unsteadiness compared with the single vertical wing pair case. This is because the two vortices on each side strongly interact, and the lower of the two has a stronger interaction with the secondary vorticity, resulting in a rather complex system.

For $h/c = 0.21$, at the closest spacing of $\Delta b/c=1.07$, the interaction causes the outside vortex to break down, since the velocity imposed on it by the inner vortex causes it to descend towards the ground. For spacings greater than this, the interaction between the two vortices is weaker, but in all cases, the weaker of the two vortices (both having the same sign) rises whilst the other descends. There are varying amounts of secondary vorticity produced at the ground plane by the stronger, outer (lower) vortex. When the vortices are closer together, the outer vortex descends more, causing it to create even more secondary vorticity.

### 6.5 Key Findings

The key findings from this chapter regarding the structure and behaviour of the front wing wake may be summarised as follows:
1. The flow in the wake of a horizontal wing in ground effect comprises a low energy wake in which there is a momentum deficit, a pair of trailing vortices, and an upwash generated by the bound vorticity.

2. Due to boundary layer growth on the suction surface growing with increased adverse pressure gradients, the momentum deficit is greater with increasing incidence and reducing ground proximity. Boundary layer growth on the moving ground plane may also leave a layer of low energy flow near the ground plane at low wing ground clearances.

3. The upwash in the wake of the wing increases with increasing incidence, due to increased strength of bound vorticity, and with increasing ground clearance of the wing, due to the decreasing influence of the ground plane.

Figure 6.35: PIV data for two sets of vertical wings, $h/c = 0.21$. 
4. Since the growth of the boundary layer with increasing incidence and reduced ground clearance is on the lower surface of the wing, the relative position of the low energy wake may be reduced by these factors. The overall location of the wake therefore depends on a combination of upwash and initial wake location.

5. As the distance between a vortex and the ground plane is reduced, there is an increasing quasi-periodic interaction between it and the secondary vorticity generated in the cross-flow boundary layer beneath it, resulting in a spiralling motion. It seems that this excites the spiral mode of vortex break down. The rate of production of secondary vorticity is greater with a plane moving at the free-stream velocity than with the plane stationary, which is why the vortex
in this case is more prone to breakdown.

6. The secondary vortices mentioned above have a considerable effect on the vortex trajectory as it convects downstream.
Chapter 7

Interaction with a Trailing Vortex System

7.1 Introduction

Within the previous chapters, a number of factors which may affect the interaction between a trailing vortex system and a downstream negative lifting body were discussed. These may be summarised as follows:

1. As a body is brought into close proximity to the ground the flow approaching it has an increased vertical component of velocity away from the ground. In three-dimensions it also has a lateral component of velocity away from the body centreline upstream of it. This reduces the mass flow rate underneath the body, reducing the flow velocity and increasing the pressure.

2. Vortex stability is reduced by the action of adverse pressure gradients and proximity to a moving ground plane.

3. Experiments by previous researchers on the interaction between trailing vortices and wings have shown that one of the more important effects is that of local changes in incidence of the flow approaching the body due to the vertical components of velocity induced by the vortex.
4. A vortex passing close to a solid boundary may induce low pressures locally on that surface.

5. The interaction between a vortex and a boundary layer may cause boundary layer material to be lifted up on the up-going side of the vortex, resulting in one or more secondary separations. Flow divergence on the down-going side of the vortex causes local boundary layer thinning.

The aim of this chapter is to investigate the mechanisms governing the interactions between single and twin trailing vortex systems and downstream bodies, in particular the diffuser model. The majority of force measurements presented here are downforce measurements since the influence of the vortices on downforce was considered to be one of the more important factors for consideration. The chapter is divided into the following two sections:

7.2 Interaction between a trailing vortex and a wing

Experiments were conducted with a single pair of vertical wings upstream of the aspect ratio one wing, in which surface static and wake total pressure measurements were made with the aim of investigating how the vortices influence the wing forces, and how the wing affects the vortex path and stability.

7.3 Interaction between a trailing vortex and a diffuser-equipped bluff body

Body force and surface pressure measurements were made on the diffuser model for a number of single and twin vertical wing pair configurations. The diffuser configuration for the majority of measurements was at the optimum ground clearance and diffuser angle for the no upstream wing case. Diffuser angles either side of the optimum were also used for a limited range of experiments to examine how the trailing vortices from the vertical wings affect the diffuser off the optimum configuration. The aim of the section is to build an understanding of the flow physics governing the interaction between the vortices and the diffuser model.
CHAPTER 7

7.2 Interaction Between a Trailing Vortex and a Wing

A number of experiments were conducted with the vertical wings upstream of the low aspect ratio wing at $0^\circ$ incidence as described in Section 3.2.2, with the vertical wings at a fixed lateral separation equal to the span of the rear wing (also equal to the chord of the rear wing, which will be used to non-dimensionalise it, i.e. $b/c_{rear} = 1.00$) and a tip ground clearance $h/c=0.71$. It is more useful in this context to non-dimensionalise the front model ground clearance by the rear model span. In this case it was $h/c_{rear}=0.10$. Figures 7.1 to 7.3 show total head contours beneath and in the wake of the low aspect ratio wing at ground clearances of $h/c=0.05$, 0.08, and 0.10 with the vertical wings upstream in the configuration mentioned above. The streamwise location of the survey plane is measured from the wing leading edge, and displayed here as a fraction of the chord.

At the lowest rear wing ground clearance of $h/c = 0.05$, there is no sign of the vertical wing trailing vortex whatsoever beneath the wing. However, at the most downstream station ($x/c = 1.02$) there is a region of low total head on the upper surface of the rear wing between $y/c=0.4$ and 0.5. This is probably the remains of the vortex, which would suggest that it may have broken down at the leading edge due to the adverse pressure in the area of the stagnation region. At $h/c = 0.08$ the vortex is definitely visible in all three plots, moving inboard as it convects through the underbody channel. This is not due to its reflection in the ground plane, since there is also a counteracting reflection in the lower surface of the wing. This inboard motion is in fact due to the inboard component of flow at the rear of the wing, as discussed in Section 4.2.2. The vortex trajectories in the vertical and horizontal directions are plotted in Figure 7.4.

At $h/c = 0.08$ the vortex rises along the entire distance $h/c=0.60$ to 1.02, which would be expected from the upward slope of the suction surface, whilst at the greater rear wing ground clearance, surprisingly, the vortex moves downwards slightly between $x/c=0.60$ and 0.70 before rising again. Closer examination of the total head contour plot of Figure 7.3 reveals evidence of a secondary vortex just outboard of
the primary vortex at $x/c = 0.60$. It can also be seen at $x/c = 0.80$, where it has been swept downwards by the main vortex. Its sense of rotation is anti-clockwise, the sense being clockwise for the main vortex. It would therefore induce a downward velocity at the primary vortex at $x/c = 0.60$ which is why it moves downwards between $x/c=0.60$ and 0.70. Further downstream, it appears that this secondary vortex merges with the primary vortex. The PIV data of Figure 6.20 shows that at a much lower wing tip ground clearance of $h/c_{rear}=0.04$ (compared with 0.10 here) very little secondary vorticity is lifted from the ground plane. It would therefore seem more likely that the secondary vortex has come from the interaction between the main vortex and the boundary layer on the lower wing surface.

At $h/c = 0.08$ the vortex appears to rapidly become more diffuse between $x/c=0.80$ and 1.02, and even at $x/c = 0.60$ appears more diffuse than for all of
Figure 7.2: Total head contours in the wake of the aspect ratio 1 wing at a ground clearance of $h/c=0.08$ with vertical wings upstream at a tip ground clearance of $h/crear=0.10$.

the chordwise planes for $h/c = 0.10$. Since these are time-averaged plots, the more diffuse appearance of the vortex could be brought about through either intermittent breakdown or wandering. At the lower wing ground clearance, the vortex passes closer to the wing surface, its interaction with which could cause unsteady motion by its interaction with secondary vorticity. Meanwhile, at this lower ground clearance, the adverse pressure gradient is stronger beneath the suction surface towards the trailing edge, which may destabilise the vortex. Unfortunately the data are not sufficient to determine exactly what the cause is, but it is probably a combination of both wandering and intermittent breakdown. From the PIV and hot-wire data for the vertical wing vortices, unsteady periodic motion of the vortex was found as a
precursor to its breakdown when close to the ground, and so a similar phenomenon may be occurring due to its interaction with the wing surface.

The change in surface pressures with the vertical wings compared to without them is presented as contour plots for ground clearances of $h/c = 0.08$ and $0.10$ in Figures 7.5 and 7.6. The locations of the pressure tappings are shown as black diamonds on the plots.

The contour levels have been set with limits at $\Delta C_p = \pm 0.10$ to highlight the positive or negative effects. On the suction surface the effect of the vertical wings is primarily an increase in pressure (shown as red) due to the upwash generated
between the vortices reducing the effective (nose-down) incidence of the wing. On the pressure surface, this results in a reduction in pressure near the nose. The net effect of this would be a nose-up pitching moment and a reduction in downforce. Towards the wing tips \((y/c = 0.50)\) this trend reverses, with a reduction in pressure on the lower suction surface, although the change on the suction surface is minimal. Since the pressure on the suction surface is reduced far more than it is increased on the pressure surface in this area, it is likely that the changes here are mainly due to the vortex passing very close to the wing surface, directly inducing a low pressure on it, rather than a local change in incidence. In this region where the vortex is passing close to the wing surface, the tangential velocities have less influence on the local incidence of the wing. This effect is strongest at \(h/c = 0.08\) where the wing is lower, such that the vortex would pass closer to the surface. As shown in Figure 7.4, towards the rear of the wing the vortices are approximately mid-way between the suction surface of the wing and the ground, and being further from the wing surface they are less likely to have as strong an effect on the surface pressures. In fact, there appears to be no identifiable trends towards the rear of the wing, other than
Figure 7.5: Change in surface pressure coefficients, $\Delta C_p$, on the rear wing for an upstream vertical wing ground clearance $h/c = 0.08$.

The strong increase in pressure on the suction surface due to the overall effective reduction in incidence due to the upwash generated by the vertical wing vortices.

### 7.3 Interaction Between a Trailing Vortex and a Diffuser-Equipped Bluff Body

The experiments with the aspect ratio one wing showed that the main effects of the vertical wings were of local changes in effective incidence due to the induced flow field, and low-pressure tracks where the vortices pass very close to the wing surface. A wider range of vertical wing configurations was tested with the diffuser model. For the majority of the results to be presented here, the diffuser model was set up in the optimum configuration for no upstream wings, which, as discussed in Chapter 6, is a diffuser ramp angle of 12° and a ground clearance of $h/c = 0.06$. Some tests
Figure 7.6: Change in surface pressure coefficients, $\Delta C_p$, on the rear wing for an upstream vertical wing ground clearance $h/c = 0.10$.

were also carried out for configurations either side of this in an attempt to quantify if and how the vortices change the diffuser performance.

### 7.3.1 Influence of Vertical Location of Vortices

Diffuser model body forces were measured with the vertical wings at a large number of different tip ground clearances at incidences of 15° (both producing an upwash at the centreline-‘conventional’) and −10° (both producing a downwash at the centreline - ‘unconventional’), with a lateral separation of $b/c_{rear}=0.6$ (300mm). This lower lateral separation was used because previous experiments showed lateral divergence of the flow approaching the model, which may push the vortices to the sides of it, whilst the aim was to keep the vortices beneath the model. This lateral flow divergence also modifies the incidence of the wings, which is why the wing incidence was reduced for the unconventional configuration. This is very difficult to
measure experimentally, and in any case will be altered by the interaction with the vertical wings. The simplest way of determining it is from the CFD data for the diffuser model without the upstream wings. This gave incidences varying between 1.6° at $b/c_{rear} = 0.50$ and 2.5° at $b/c_{rear} = 1.00$ at the location of the vertical wing tip quarter-chord points for $h/c_{rear} = 0.05$, inclined so as to reduce the effective incidence of the vertical wings. This rises to a maximum of 3.6° at the position of the vertical wing tip trailing edge. Due to this effect, the vertical wings were found to stall at an incidence of $-15°$ when placed upstream of the diffuser model, and so an incidence of $-10°$ was used for the unconventional configuration. Unfortunately it has not been possible to quantify the relative strengths of the vortices due to the presence of the model.

The percentage change in diffuser model downforce brought about by the vertical wings is plotted against tip ground clearance in Figure 7.7 showing an extremely strong dependency of the interaction on wing tip ground clearance. For the conventional signed vortices this is only beneficial for ground clearances below $h/c_{rear} = 0.024$, or $h/c = 0.171$.

![Figure 7.7: Effect of vertical wing tip ground clearance, $h/c_{rear}$, on diffuser model downforce coefficient. $\theta = 12°$, $h/c = 0.06$](image-url)
Referring back to Section 6.4.3, the vortex from the wing in this configuration was found to break down for ground clearances below \( h/c = 0.21 \). In this case, the vortex is likely to be weaker, due to the reduction in effective incidence of the wings as they are upstream of the diffuser model, where the flow is diverging. However, it is still likely that the vortices break down at the lower tip ground clearances (down to \( h/c_{rear} = 0.004 \), or \( h/c = 0.057 \)), where the downforce enhancement continues to rise. It would therefore seem that the state of the vortices by the time they reach the rear body does not have a strong impact on their influence on the rear body downforce.

The effect of the unconventional vortex configuration is almost a mirror of that for the conventional configuration, producing a downforce reduction at low tip ground clearances, and an enhancement at large ground clearances. To help explain these phenomena, diffuser model centreline surface pressure distributions are shown in Figure 7.8 for a selected number of vertical wing tip ground clearances for the conventional vortex configuration.

![Figure 7.8](image)

Figure 7.8: Effect of vertical wing tip ground clearance, \( h/c_{rear} \), on diffuser model underbody centreline pressures. \( \theta = 12^\circ, h/c = 0.06 \)

These surface pressure plots suggest that the change in diffuser model downforce
due to the vertical wings comes about through a change in the surface pressures nearest the front of the underbody: the pressures getting lower with reduced vertical wing tip ground clearance. The change in surface pressure at $x/c=0.28$, on the underbody centreline is plotted against vertical wing tip ground clearance, as is the diffuser model percentage change in downforce, in Figure 7.9. The two curves correspond well, with downforce and inlet suction enhancement occurring over almost exactly the same range of tip ground clearances. There are small differences in the shape of the curves, due to other mechanisms which will be discussed further on in this chapter, but it seems that this change in suction at the model underbody inlet is probably the biggest contributor to the interaction mechanism.

![Figure 7.9: Effect of vertical wing tip ground clearance, $h/c_{rear}$, on diffuser model underbody centreline pressure at $x/c = 0.28$. $\theta = 12^\circ$, $h/c = 0.06$](image)

Looking at the pitching moment coefficient behaviour, if the change in downforce is mostly created at the front of the model, then the change in forces should be accompanied by a corresponding change in pitching moment coefficient, with an increase in downforce mirrored by an increase in (nose-down) pitching moment. The pitching moment data corresponding to the downforce coefficient data shown
above is plotted for the conventional and unconventional vortex configurations in Figure 7.10, showing that this is indeed the case.

![Diagram](image)

Figure 7.10: Effect of vertical wing tip ground clearance, $h/c_{rear}$, on diffuser model pitching moment. $\theta = 12^\circ$, $h/c = 0.06$.

As discussed in Chapter 5, one of the biggest factors affecting the performance of the diffuser model is the flow direction through the underbody, with outboard lateral components on the approach to, and under the upstream portion of the model underbody reducing the flow velocity, and increasing the pressure. The conventional vertical wing tip vortex configuration is likely to have two major effects on the flow field approaching the model. The first is to reduce the effective (nose-down) incidence of the model by the upwash induced between them. This will tend to reduce the velocity under the upstream portion of the underbody, increasing the pressure. The second is to induce an inboard component of flow beneath the vortices, pushing more air beneath the model, increasing the velocity and reducing the pressure. As the vortices are brought towards the ground plane, the first effect will be reduced by the presence of the ground plane, and the second enhanced. For the unconventional vortex configuration, these trends are reversed, with a downwash between the vortices increasing the effective incidence of the model, but at the same time increasing
the outboard component of flow approaching the model.

The diffuser model forces were also measured with diffuser ramp angles either side of the 12° at which the results have been presented so far to see how the vertical wings affect the model performance away from the optimum configuration. These results are plotted in Figure 7.11, and show the effect of the diffuser ramp angle to be small. The general trends are certainly the same, with a downforce enhancement at low tip ground clearances, and the downforce enhancement falling with increased tip ground clearance, giving a reduction in downforce at the greatest ground clearances.

![Figure 7.11: Effect of vertical wing tip ground clearance on diffuser model downforce with different diffuser ramp angles. θ = 12°, h/c = 0.06.](image)

There is, however, some difference between the behaviours at different ramp angles in that at the smallest diffuser ramp angle of θ = 10° the reduction in downforce at the largest wing tip ground clearances is greater than for the other two cases. At the larger diffuser angles, there is a greater amount of boundary layer thickening and separation in the diffuser, both of which reduce the diffuser performance, and thus the downforce produced by the model. Figure 7.12 shows the chordwise centreline pressure distributions on the diffuser model with ramp
angles of 10° and 14° at three different vertical wing tip ground clearances.

Figure 7.12: Effect of vertical wing tip ground clearance on diffuser model centreline underbody pressure distributions for diffuser angles of $\theta = 10^\circ$ and $14^\circ$. $h/c = 0.06$.

As can be seen from the centreline underbody pressure distributions, for a diffuser angle of 10° only the pressures at the most upstream pressure tappings are affected, as found previously. However, for the diffuser angle of 14°, the pressure rise through
the diffuser increases with increased wing tip ground clearance, suggesting improved diffuser performance. Looking more carefully at the surface pressure data for \( \theta = 12^\circ \), there is also an improvement in the pressure recovery through the diffuser with the larger vertical wing tip ground clearances, but this improvement is not as great. To provide a quantitative comparison of this effect, the change in suction with vertical wing tip ground clearance at the diffuser entrance \((x/c = 0.80)\) is plotted for the three ramp angles in Figure 7.13. This figure shows that even at \( \theta = 12^\circ \), the pressure at the diffuser entrance is improved, but just not as much as for \( \theta = 14^\circ \).

![Figure 7.13: Effect of vertical wing tip ground clearance on diffuser model underbody centreline pressure at \( x/c = 0.80 \) with different diffuser ramp angles. \( h/c = 0.06 \).](image)

The pressure distributions at \( \theta = 14^\circ \) in Figure 7.12 show a definite flattening towards the rear of the diffuser, indicating flow separation in the diffuser, which is less prominent with increasing vertical wing tip ground clearance. This improvement in diffuser pressure recovery is therefore most probably due to the vortices delaying flow separation in the diffuser. At \( \theta = 12^\circ \) there is only a small area of intermittent flow separation in the diffuser, so the influence of the vortices is only small, but as the diffuser ramp angle is increased and flow separation becomes more prominent,
their benefit becomes stronger. At the lowest wing tip ground clearances, the vortices break down, and so are unable to aid the diffuser flow. At greater ground clearances, they become more stable, and will interact more strongly with the diffuser boundary layer, resulting in greater improvement in pressure recovery. At $\theta = 10^\circ$ the vortices have virtually no influence on the pressure at the diffuser inlet. At this ramp angle, there is no separation in the diffuser, but there is significant boundary layer growth. The benefit of the vortices on the diffuser flow at the greater ground ramp angles must therefore be more due to their delaying of flow separation than thinning of the boundary layer.

For $h/c_{rear} = 0.05$ there was evidence of the vortex inducing a low pressure track over the diffuser model undersurface at the most upstream row of pressure tappings at $x/c = 0.28$. Figure 7.14 shows the spanwise pressure distributions at this location for the vertical wings at $h/c_{rear} = 0.04$, 0.05, and 0.06. At $h/c_{rear} = 0.06$, there is evidence of the peak moving outboard slightly, which is where the vortex would move under the influence of its image in the model lower surface. Downstream of this, the peak disappears, perhaps due to the vortex lifting from the lower surface due to its interaction with secondary vorticity.
7.3.2 Influence of Lateral Separation of Vortices

7.3.2.1 Single Vortex Pair

The change in downwash due to the vertical wing versus lateral separation at tip ground clearances of $h/c_{rear} = 0.04$, and 0.05 are plotted in Figure 7.15.

Figure 7.14: Semi-spanwise pressure distributions at $x/c = 0.28$ with vertical wings at $h/c_{rear} = 0.04$, 0.05, and 0.06. $\theta = 12^\circ$, $h/c = 0.06$. 
7.3.2 Influence of Lateral Separation of Vortices

7.3.2.1 Single Vortex Pair

The changes in downforce due to the vertical wings versus lateral separation, $b$, at tip ground clearances of $h/c_{rear} = 0.01$, and 0.05 are plotted in Figure 7.15.

![Figure 7.15: Effect of vertical wing spanwise separation, $b/c_{rear}$, on diffuser model downforce for $h/c_{rear} = 0.01$ and 0.05, with conventional and unconventional vortex configurations. $\theta = 12^\circ$, $h/c = 0.06$.](image)

In all four cases, for $b/c_{rear} < 0.7$ the diffuser model downforce is relatively insensitive to lateral separation of the vertical wings, and the downforce is increased or reduced as per the findings of the previous section for the variation in downforce with wing tip ground clearance. However, for lateral wing separations above this, the downforce decreases for the conventional configuration down to a minimum at $b/c_{rear} \approx 0.85-0.90$, above which the downforce rises again. For the unconventional vortex configuration, the trend again is almost a mirror image, with downforce rising for increasing lateral separations between 0.70 and 0.85, above which the downforce falls again. However, the downforce peaks are not as prominent as the troughs for the conventional configuration, particularly at $h/c_{rear} = 0.05$. 

...
The general behaviour may be explained using the same arguments as in the previous section, that the conventional vortices produce an upwash between them that increases with increased ground clearance, and an inboard component of flow beneath them that is enhanced with reduced ground clearance. As the spanwise separation is increased, for fixed ground clearance, so long as the vortices pass beneath the diffuser model, the variation in upwash and inboard flow component, affecting the mass flow beneath the model, will be small and gradual. At small spanwise separations, the action of upwash in reducing the effective incidence of the model acts over a smaller portion of the model width, whilst the action of the inboard component of flow in increasing the flow beneath the model would also be reduced. However, as soon as the vortices pass to the sides of the model, the inboard component of flow generated beneath the conventional signed vortices will rapidly become less effective in pushing more flow beneath the model, whilst the upwash will continue to behave as before. The net result is a rapid decrease in downforce. As the spanwise separation is increased further, the level of upwash generated by the vortices continues to diminish, allowing the downforce to rise again.

As shown in the previous section, for a wing tip ground clearance of $h/c = 0.05$ a low pressure track was found to appear on the diffuser model undersurface. Figure 7.16 shows semi-spanwise pressure distributions on the underbody at $x/c = 0.28$ for a number of spanwise separations of the vertical wings at a tip ground clearance of $h/c_{rear} = 0.05$.

The low pressure peak is clearly visible for all of the spanwise pressure distributions for $b/c_{rear} < 0.75$, moving outboard with increasing spanwise separation of the vertical wings, as would be expected. At $b/c_{rear} = 0.75$, there is evidence of the vortex at the most outboard pressure tapping, suggesting the vortex to still be beneath the model, albeit right at its edge. At $b/c_{rear} = 0.80$ the peak disappears, suggesting that the vortex now passes to the side of the model, coinciding with a sudden rise in pressure along the entire span.

Figure 7.17 shows the variation in centreline pressures with lateral separation of the wings at $x/c = 0.28$ and $x/c = 0.80$, for the conventional and unconventional vortex configurations with a wing-tip ground clearance of $h/c_{rear} = 0.05$. For the
conventional vortex configuration, the pressure at $x/c = 0.28$ is only significantly increased for $b/c_{rear} = 0.80-0.85$, where the minimum downforce occurs. Meanwhile the pressure at $x/c = 0.80$, the diffuser inlet, is reduced over the entire range, signifying an improvement in diffuser performance. For $\alpha = -10^\circ$, however, the pressure at $x/c = 0.28$ is reduced over the entire range of lateral separations, whilst that at the diffuser inlet is only slightly increased for most cases, if at all.

Referring back to the plot of downforce change in Figure 7.15, this pressure data does not, by itself, sufficiently explain the trends seen in the forces. For example, the slight pressure increase at $b/c_{rear}=0.90$, for $\alpha = 15^\circ$ at $x/c = 0.28$ is much less that the pressure drop for the same configuration at $x/c = 0.80$. The underbody centreline pressure distributions for this configuration, and for the case with no upstream wings is shown in Figure 7.18. This would suggest an increase in downforce as overall the average pressure is reduced between $x/c = 0.28$ and 0.95, whilst in fact this configuration reduces the downforce generated by the diffuser model.

Looking at the trends in the pressure distributions with varying vertical wing
Figure 7.17: Centreline underbody pressure variation at $x/c = 0.28$ and $0.80$ versus lateral separation of vertical wings at $\alpha = -10^\circ$ and $15^\circ$ at a tip ground clearance $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$.

Figure 7.18: Centreline underbody pressure distributions with and without vertical wings at $\alpha = 15^\circ$, $h/c_{rear} = 0.05$, and $b/c_{rear} = 0.90$. $\theta = 12^\circ$, $h/c = 0.06$. 

---

**Solid Lines**: $x/c=0.28$

**Dotted Lines**: $x/c=0.80$
separation, it would suggest that the pressure further upstream becomes progressively greater than that with no vertical wings as the nose is approached. This means that the stagnation point must be lower with the vertical wings, as would be expected from the upwash generated by the trailing vortices, which would not only increase the pressure beneath the nose, but reduce the pressure above it. This would reduce the (nose-down) pitching moment, which as can be seen from Figure 7.19 is in fact the case.

![Figure 7.19: Pitching moment coefficient versus lateral wing separation at $\alpha = -10^\circ$ and $15^\circ$, $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$.](image)

7.3.3 Interaction with Multiple Vortex Pairs

The arrangement of a Formula One front wing is such that a number of vortices of different strengths and signs may be shed from either side of the centreline, as illustrated in Figure 2.11. To consider the effects of this, a number of configurations involving two sets of vertical wings were tested with the diffuser model, again in the optimum no-wing configuration. The key aim of this exercise was to see if the knowledge gained during the course of the project could be applied successfully to this problem. Adding an extra set of wings not only significantly increases the
number of possible configurations which could be tested, but also adds significant complexity to the problem, which is still not fully understood for the case of a single vortex pair.

The configurations tested involved the two sets of vertical wings, with the outer wings at a fixed incidence of $15^\circ$, and the inner wings at an incidence of either $10^\circ$ or $-10^\circ$. Tests were made with the outer set of wings at a fixed lateral separation of $b/c_{rear} = 1.0$, whilst the inner set of wings were moved between $0.5 < b/c_{rear} < 0.85$. Further tests were conducted with the inner set of wings at a fixed lateral separation of $b/c_{rear} = 0.50$, whilst the outer set were moved between $0.65 < b/c_{rear} < 1.00$. This was only carried out at vertical wing tip ground clearances of $h/c_{rear} = 0.05$.

7.3.3.1 Same signed vortices

It has been established that for the conventional vortex configuration, it is most beneficial for the trailing vortices to be inboard of $b/c_{rear} \approx 0.70$ and as close to the ground as possible. With two sets of vortices, there is not only the action of both of the vortices to consider, but also the interaction between them which has significant consequences for their trajectories. For vortices of the same sign, both producing an upwash at the model centreline (positive incidence) the inner vortices will move upwards and outwards, both of which are detrimental to the diffuser model downforce. The outer vortices will move downwards and inboard, which is likely to have a positive effect. However, with both sets inducing an upwash at the centreline, reducing the effective incidence of the diffuser model, the overall effect is likely to be a reduction in downforce. Figure 7.20 shows the change in downforce due to the same conventional signed vortices with wing tip ground clearance of $h/c_{rear} = 0.05$. The data for the single set at an incidence of $15^\circ$ are also shown on the plot for comparison.

With the inner vertical wing fixed, there is very little difference between the twin wing results and the single wing pair, with the same basic trends, but modified in some places. There is only one position where the downforce is greater than that for the single wing pair, at $b/c_{rear} = 0.80$. Meanwhile, with the outer vortices fixed, the reduction in downforce is considerably greater than for the single wing pair, and the
The position of minimum downforce has been shifted to a lower spanwise separation. It is helpful when considering this data to refer back to Figures 6.35 and 6.36, showing the vortex positions at the location of the rear wing for different separations between the wing pairs. The cases $\Delta y/c = 1.07, 1.43, \text{ and } 1.79$ correspond to the cases here for the fixed outer wings with the inner wings at $b/c_{rear} = 0.7, 0.6, \text{ and } 0.5$ respectively, and for the fixed inner wings with the outer wings at $b/c_{rear} = 0.8, 0.9, \text{ and } 1.0$ respectively.

With the inner wing at $b/c_{rear} = 0.50$, and the outer wings at $b/c_{rear} = 1.0$, the vortex positions are not strongly affected by the interaction between them. The resulting reduction in downforce is almost exactly the sum of the reductions for the single pair at $b/c_{rear} = 0.5 \text{ and } 1.0$ (although the inner vertical wing is at a lower incidence, thereby producing a weaker vortex). As the outer vertical wing is moved inwards, the inner vortex rises moving up and over the main vortex. Since the main vortex is stronger, they will orbit about a point closer to it, such that its path is less affected than that of the weaker vortex. However, as it is pushed towards
the ground by the weaker vortex, its interaction with the ground causes it to be swept further inboard, which is probably why the position for minimum downforce is at a slightly greater outer vertical wing position. Moving the outer wings further inboard results in a less detrimental effect since both vortices are closer to the model centreline. It is extremely difficult to know what the exact mechanisms are as it was not possible to determine the precise trajectories of the vortices: their interaction with each other, the ground plane, and the diffuser model will have a considerable effect on the interaction, which is extremely complicated and difficult to predict.

As the inner vertical wings are moved outboard, the fixed outer vortices lift them upwards, where it is known they are more detrimental to the downforce produced by the diffuser model. It is also known that they are also more detrimental the further outboard they are, and so the downforce is further decreased. The location for minimum downforce occurs at a smaller spanwise separation of the inner vortices, because they move the outer vortices inboard, possibly resulting in a situation where both vortices are closer to the location just outboard of the model sides where it is known that they have the worst effect.

In conclusion, as may be expected, if two vortices of the same sign are to be shed from an object upstream of the model, they should both be as far inboard as possible for minimum loss of downforce.

7.3.3.2 Opposite signed vortices

Figure 7.21 shows the change in downforce with the two sets of wings as before but with the inner wings at an incidence of $-10^\circ$. The results for the single wing pair at incidences of $15^\circ$ and $-10^\circ$ are also included on the plot for comparison. It should be noted that due to the outboard component of flow upstream of the model, which as mentioned previously has been estimated from CFD data to reduce the wing incidences by between 1.6 and 2.5°, the effective incidence of the outer and inner (positive for one and negative for the other) vertical wings may be similar in magnitude.

Assuming that the vortices are of similar magnitude, they will both tend to rise under their mutual induction. This will become stronger as they are brought closer
CHAPTER 7

7.4 Key Findings

The interaction of the individual wing vortices strongly influences the lift distribution of the trailing edge. This is seen in Figure 7.21, which shows the effect of vertical wing spanwise separation, $b/c_{rear}$, on diffuser model downforce for two wing pairs, with $\alpha_1 = -10^\circ$ and $\alpha_2 = 15^\circ$, and for a single wing pair at $\alpha = 15^\circ$ and $\alpha = -10^\circ$, all at $h/c_{rear} = 0.05$. $\theta = 12^\circ$, $h/c = 0.06$.

together, not only because of their increased proximity, but also due to the reduced influence of the counterpart vortex pair on the other side of the centreline: the negative incidence wing, being closer, inducing a downward velocity on them. It is known that for the unconventional signed vortices, the benefit is greater the higher the vortex is, the reverse being true for the conventional signed vortex. Again, the basic behaviour is not dissimilar to a superposition of the behaviours of the two sets of results for the single wing pairs. However, for the case where the inner set of wings is moved outboard, the enhancement in downforce is above that which may be expected from a superposition of the single pair results alone. This is because as the inner wings are brought towards the outer wings they rise more, and it is known that the unconventional vortices are more beneficial the higher they are from the ground. Meanwhile, since the outer wing vortices are passing to the sides of the model, the fact that they also rise is less important than if they were further inboard. The net result is therefore a gain in downforce beyond that which may be expected from the results for the single wing pairs.
7.4 Key Findings

The investigation of the interaction between trailing vortex systems from an upstream body and a negative lifting wing or diffuser-equipped bluff body has resulted in the following key findings, listed in order of their apparent significance, greatest first.

1. Probably the biggest single influence of the trailing vortices is that of their induced velocity field. This generates local changes in incidence along the span which diminish with reduced ground clearance due to their images in the ground plane. An upwash between the vortices reduces the effective incidence of the rear body, reducing the downforce. This effect does not appear to be strongly influenced by the state of the vortices, and the general trends with reducing ground clearance continue even though the vortices may break down.

2. The lateral component of flow induced by the vortices is greater beneath than above them due to the influence of the ground, and increases with reduced ground clearance. If this component is directed towards the model centreline, then they direct more flow beneath the model, increasing the downforce. This influence is most effective when the vortices are arranged so that they pass underneath the rear model, rather than to the sides of it.

3. The vortices, if they survive their passage to the diffuser, can help maintain attached flow in the diffuser, enhancing its effectiveness. This results in a lower pressure at the diffuser entrance.

4. If they pass close enough to the underbody, then the vortices may induce a low pressure directly where they pass over it. This does not seem to have a strong effect on the body forces, however, and so is a secondary consideration. The low pressure track is most prominent furthest upstream, as the vortex tends to lift away from the surface further downstream.

These findings also apply to the case of multiple trailing vortices, but the prediction of the resulting forces is complicated by the complex interaction between the vortices, the ground and the rear body which alters their path.
Chapter 8

Interaction with an Upstream Horizontal Wing

8.1 Introduction

The understanding of the interaction between the diffuser model and an upstream horizontal wing is the main objective of this study, and the purpose of the previous chapters has been primarily to build a basic understanding of the various mechanisms involved in the problem. The principal findings from the previous chapters relating to this goal may be summarised as follows:

1. The wake of a horizontal wing in ground effect consists of one or more pairs of trailing vortices, a low energy (momentum-deficit) turbulent wake, including a layer of low energy flow near the ground remaining from the boundary layer beneath the wing on the ground plane, and the upwash generated by the bound vorticity.

2. Depending on its height above it, a trailing vortex may have a complicated interaction with the ground plane involving secondary vorticity lifted up from the cross flow boundary layer beneath the vortex on its up going side. This results in a quasi-periodic motion of the vortex, which can ultimately result in its breakdown.
3. The flow approaching a body in ground effect has a lateral velocity away from the body centreline, and vertical component away from the ground. These components increase in magnitude with increasing proximity to the ground. This reduces the volume flow rate of air passing beneath the body, reducing its velocity, increasing the pressure, and lowering the stagnation point on the body nose, reducing the effective (nose-down) incidence.

4. Upwash generated by an upstream wing further reduces the effective (nose-down) incidence of the downstream body, resulting in a loss in downforce and pitching moment. The upwash in the wake of such a wing reduces with increasing ground proximity.

5. A streamwise vortex in ground effect generates a tangential component of flow that is greater between it and the ground than above it. A pair of trailing vortices from an upstream wing modify the lateral component of flow approaching the downstream model, either enhancing or reducing the downforce produced by it depending on whether the tangential velocity is towards or away from the centreline. For a negative lifting wing, the trailing vortices produce an upwash between them and an inboard component of lateral flow towards the rear body centreline, forcing more flow beneath it and increasing its downforce.

6. When passing very close to the model lower surface, the vortices may induce a low pressure track on it. This low pressure track tends to fade and disappear with distance downstream as the vortex lifts from the model surface through its interaction with secondary vorticity.

7. For large diffuser ramp angles, where the flow is otherwise partially or fully separated, the trailing vortices, if still coherent by the time they reach it, can delay flow separation from the ramp. This enhances the pressure recovery through the diffuser, increasing the level of downforce produced.

Since the upstream vertical wing leaves behind it an upwash and a low energy wake, it is most likely to reduce the downforce produced by the diffuser model. However, it is known that the trailing vortices may have a beneficial effect, and
optimal use of this component of the wake should allow the downforce loss to be
minimised. This interaction will be discussed in the present chapter, divided into
the following sections:

8.2 Influence of low energy turbulent wake

A cylindrical rod is used to simulate the momentum deficit component of the
wake. Force and surface pressure measurements on the diffuser model with
different diameter rods at different ground clearances are used to identify the
influence of the low energy wake on the diffuser model.

8.3 Influence of full-span horizontal wing

Measurements were made on the diffuser model with the full-span horizontal
wing upstream of it. Having a span equal to that of the diffuser model, the
flow divergence ahead of the rear model causes the trailing vortices to pass to
the sides, rather than beneath it. This allows the influence of the front wing
to be analysed without the direct interaction between the trailing vortices and
the model underbody.

8.4 Influence of partial-span horizontal wing

Finally, a partial span front wing is placed upstream of the diffuser model,
having a span equal to 60% of the rear model span. At this lower span the
trailing vortices pass beneath the diffuser model, allowing the direct interaction
between them and the model underbody and diffuser to be studied. Using the
results of the previous two sections, a description is given of how the various
components of the front wing wake affect the aerodynamics of the diffuser
model.

8.2 Influence of Low Energy Turbulent Wake

As described in Section 6.3, cylindrical rods of varying diameter were used to sim-
ulate the momentum deficit in the wake of the wing. The benefit of this is that
it separates the momentum deficit component of the wake from the upwash and
trailing vortices associated with the wing. However, it has two significant disadvantages. Firstly the turbulence intensities and structure will be different in the two cases. Secondly, only centreline traverses were used to compare the wake momentum deficits. In reality both wakes are three-dimensional, in particular in the case of the wing whose trailing vortices will tend to lift the centre of the wake, giving it a curved shape, as shown in the schematic diagram of Figure 8.1.

![Figure 8.1: Schematic diagram showing possible distortion of front wing wake.](image)

Cylindrical rods of varying diameter were placed upstream of the diffuser model, at its optimum configuration for the no upstream body case, at a number of different ground clearances. The variation in downforce with cylindrical rod ground clearance for the different rod diameters, \( \phi \), is plotted in Figure 8.2.

These results are surprising since it was assumed prior to these experiments that the momentum deficit in the wake would result in a lowering of the velocity underneath the diffuser model, and a reduction in the level of downforce produced. In fact, on average the downforce is enhanced by the cylindrical rods, if only slightly. Furthermore, there is very little effect of rod diameter, again a surprising result since, as shown in Figure 6.9, the momentum deficit in the wake of the rod increases almost linearly with increasing diameter. This would therefore suggest that the momentum deficit is not as important a consideration as was first thought. Figure 8.3 shows
Figure 8.2: Variation in diffuser model downforce with cylindrical rod ground clearance for a number of rod diameters. $\theta = 12^\circ$, $h/c = 0.06$.

the change in pressure at the centreline underbody tappings at $x/c = 0.28$ and 0.80 plotted against ground clearance of the rod due to the upstream cylindrical rod of diameter $\phi/c_{rear} = 0.016$.

For all ground clearances, the pressure at $x/c = 0.28$ is increased by the presence of the upstream rod. This pressure increase, on average, slowly falls with increasing ground clearance of the rod at a rate which increases significantly between $h/c_{rear} = 0.60$ and 0.10 until at the largest ground clearance the pressure is actually reduced slightly by the presence of the rod. As the ground clearance is increased, the wake rises such that some of it will begin to pass over as opposed to under the diffuser model. The average momentum deficit passing beneath the diffuser model therefore falls, allowing the pressure at $x/c_{rear} = 0.28$ to drop.

Meanwhile the pressure at the diffuser entrance, at $x/c = 0.80$, is reduced for all ground clearances suggesting an improved diffuser performance, a surprising result since, as discussed in Section 2.2.6 diffuser effectiveness tends to be degraded by inlet blockage, although the blockage being referred to was in the form of the boundary
Figure 8.3: Variation in centreline underbody pressures at $x/c = 0.28$ and $0.80$ with ground clearance of $\phi/c_{rear} = 0.016$ cylindrical rod. $\theta = 12^\circ$, $h/c = 0.06$.

layers rather than a momentum deficit through the centre. The greatest increases in diffuser entrance suction are at the lowest and highest ground clearances of the cylindrical rod, and there is a large reduction (almost to zero) in the suction increase at $x/c_{rear} = 0.04$. The CFD data for the diffuser model without an upstream body suggests that with the cylindrical rod at this ground clearance, the centre of the wake would pass almost exactly mid-way between the ground and lower surface of the diffuser model.

It is difficult to speculate exactly what is happening since the wake of the cylindrical rod is likely to be distorted significantly as it passes beneath the diffuser model. However, the most logical conclusion would be that the turbulence in the wake of the cylinder is helping to maintain attached flow in the diffuser. The vertical location of the cylindrical rod determines what part of the wake affects the diffuser flow. Whilst the turbulence in the wake would help maintain attached flow, if the momentum deficit in the wake merges with the underbody boundary layer, reducing its energy and thickening it. This could certainly be expected to reduce the diffus-
er performance by the increased boundary layer thickness and may even promote separation. Furthermore, the turbulence in the wake may only help to reenergise the boundary layer if there is higher energy flow present with which to mix it. If the wake passes midway between the lower surface and ground, then all of the flow there may be momentum deficit wake flow, which is the worst-case scenario with regards to the diffuser performance.

Figure 8.4: Variation in downforce with ground clearance of $\phi/h_{\text{rear}} = 0.016$ cylindrical rod for $\theta = 10^{\circ}$, $12^{\circ}$, and $14^{\circ}$. $h/c = 0.06$.

Figure 8.4 shows the variation in diffuser model downforce with the $\phi/c_{\text{rear}} = 0.016$ rod ground clearance for diffuser angles of $\theta = 10^{\circ}$, $12^{\circ}$, and $14^{\circ}$. This shows that the improvement in diffuser model downforce increases with increasing diffuser angle, again supporting the hypothesis that the turbulence in the wake of the cylindrical rod delays separation in the diffuser. At the lowest diffuser angle of $10^{\circ}$ the flow in the diffuser without the upstream rod is fully attached, and so the effect of the momentum deficit in the wake of the rod reducing underbody velocities, thereby reducing the downforce, is the dominant one. At the largest diffuser ramp angle, where there is the greatest degree of separation in the diffuser, the action of the rods in delaying separation is the most important effect. The centreline underbody pres-
Pressure at $x/c = 0.28$ and $0.80$ are plotted against rod ground clearance in Figure 8.5 for the diffuser ramp angles of $\theta = 10^\circ$ and $14^\circ$.

![Figure 8.5: Variation in centreline underbody pressures at $x/c = 0.28$ and $0.80$ with ground clearance of $\phi/c_{rear} = 0.016$ cylindrical rod with $\theta = 10^\circ$ and $\theta = 14^\circ$. $h/c = 0.06$.](image)

There is a trade off between the two effects: one due to the momentum deficit; and the other due to the separation-delaying turbulence in the wake of the cylindrical rod. Referring to Figures 8.5 and 8.3, in all cases the pressure at $x/c_{rear} = 0.28$ rises gradually with increasing ground clearance of the rod but by an amount which is reduced with increasing diffuser ramp angle. Meanwhile, the pressure at $x/c_{rear} = 0.80$ is increased for $\theta = 10^\circ$, and reduced for the other two diffuser angles by an amount which varies with the ground clearance of the rod. At the larger diffuser angles, the improvement in diffuser flow caused by the turbulent wake of the rod delaying separation helps to lower the pressure at $x/c_{rear} = 0.28$. Hence the increase in pressure at this point due to the momentum deficit falls with increasing diffuser ramp angle.

Overall it would seem that the momentum deficit in the wake of the rod increases the pressure on the underbody just downstream of the nose by an amount which
falls with increasing ground clearance of the rod, as more of the wake is diverted over the model, and less passes beneath it. Meanwhile the turbulence in the wake delays separation in the diffuser, lowering the pressure at its entrance. The amount by which it does so is highly dependant on the vertical position of the cylindrical rod, and the optimum location is a function of the diffuser ramp angle. However, the effect may only be beneficial where there is otherwise some separation of the flow from the diffuser ramp, and so has virtually no benefit at low diffuser ramp angles. The result is that for values of $\theta$ where there is no separation in the diffuser without an upstream body, the cylindrical rod reduces the downforce produced by the diffuser model, but for higher values of $\theta$ the presence of the rod can actually enhance the downforce by a considerable amount (almost as much as 10% here).

### 8.3 Influence of Full-Span Horizontal Wing

Measurements were made on the diffuser model with an upstream wing having the same span, $b/c_{rear} = 1.0$. Since, with this wing, the trailing vortices pass to the sides of the diffuser model, the results show the effects of the momentum deficit and associated upwash in the wake of the front wing with little direct interaction between the trailing vortices and the model underbody. The trailing vortices may still, however, be expected to have some indirect interaction in the velocity they induce upstream of it, and along the model sides.

Figure 8.6 shows the variation in downforce for the diffuser in its optimum configuration with upstream wing ground clearance for a number of incidences. In all cases the downforce is reduced by the presence of the front wing, and the downforce generally falls with both increasing wing ground clearance and incidence, except at the larger ground clearances where a further increase in ground clearance of the front wings results in a small increase in diffuser model downforce.

There are now four considerations to be made: the effect of the momentum deficit in the wake; the turbulence in the wake; the upwash; and the influence of the trailing vortices passing to the side of the model. As discussed in Section 6.3, the momentum deficit in the wake generally increases with increasing incidence and
Figure 8.6: Variation in diffuser model downforce with upstream full-span \((b/c_{rear} = 1.0)\) wing ground clearance for a number of wing incidences. \(\theta = 12^\circ, h/c = 0.06.\)

ground proximity. Meanwhile, the upwash in the wake increases with increasing incidence and ground clearance, reducing the effective (nose-down) incidence of the diffuser model, reducing the downforce. However, the upwash also lifts the wake, and as shown in the previous section, the behaviour of the downforce depends very much on where the wake passes relative to the diffuser model. The trailing vortices at this spanwise separation are known to reduce the downforce by generating an upwash at the centreline by an amount which increases with increasing ground clearance.

The change in diffuser model underbody centreline surface pressures at \(x/c = 0.28\) and 0.80 are plotted against ground clearance of the full span wing at incidences of 0\(^\circ\), 4\(^\circ\), and 8\(^\circ\) in Figure 8.7.

Comparing this figure to the plots of downforce coefficient in Figure 8.6, the behaviour of the downforce appears to be linked to the changes in underbody centreline pressure at \(x/c = 0.28\). In all cases there is a slight reduction in pressure at \(x/c = 0.80\), which increases with increasing ground clearance, but this is a secondary feature compared with the change in pressure at \(x/c = 0.28\). For the wing
at $8^\circ$ incidence the maximum increase in pressure at $x/c = 0.28$ is $\Delta C_p \approx 0.14$. The maximum pressure increase at the same point due to the $\phi/c_{rear} = 0.016$ cylindrical rod, which produces a greater momentum deficit than the horizontal wing is $\Delta C_p = 0.03$. The most important factor must therefore be the effective reduction in incidence generated by the trailing and bound vortices. This explains why the changes in downforce with the full-span wing are much greater than for the cylindrical rod.

The reduction in pressure at $x/c = 0.80$ due to the full-span horizontal wing is of the same order of magnitude as for the $\phi/c_{rear} = 0.016$ cylindrical rod, indicating that the mechanism increasing the diffuser performance in both cases may be similar. Since the vortices for the full-span wing pass to the side of the diffuser model, they are likely to have any effect on the diffuser performance, and so this change must be due to the interaction between the turbulent low energy wake of the wing and the diffuser flow. Again, as for the cylindrical rod, the turbulence possibly delays separation in the diffuser, improving its effectiveness, and this outweighs the
reduction in performance due to the diffuser ingesting the momentum deficit of the wing wake. Unfortunately there is no data available to compare the turbulence fields in the wakes of the cylindrical rods and wings, since a meaningful analysis would require all of the Reynolds stress components to be evaluated, which is beyond the scope of this project.

The variation in diffuser model downforce with horizontal wing ground clearance for different incidences at diffuser ramp angles of $\theta = 10^\circ$ and $14^\circ$ is shown in Figures 8.8 and 8.9. The general trends are the same as for the diffuser ramp angle of $12^\circ$, but with two small but significant differences. Firstly, the maximum loss in downforce due to the full-span front wing decreases with increasing diffuser ramp angle. Secondly, there are indications of some reversal in the reduction in downforce with increasing wing angle of attack for the greater diffuser ramp angles. In particular, at $h/c_{\text{rear}} = 0.03$, for $\theta = 10^\circ$, the downforce falls monotonically with increasing wing incidence. However, at $\theta = 12^\circ$ the rate at which the downforce falls with increasing wing incidence diminishes rapidly for $\alpha > 4^\circ$, whilst at $\theta = 14^\circ$ the trend reverses completely for $\alpha > 4^\circ$, with a further increase in incidence resulting in a downforce increase for the diffuser model.

The fact that the loss in downforce is reduced with increasing diffuser ramp angle adds further evidence that the interaction between the turbulent wake of the wing and the diffuser flow delays stall in the diffuser, increasing its effectiveness. Therefore for diffuser configurations where the flow in the diffuser is otherwise separated, the downforce loss due to the upstream wings is reduced. The reasons behind the behaviour in the trends for the wing ground clearance of $h/c_{\text{rear}} = 0.03$ are not fully understood, but as the trend reversal mentioned above increases with increasing diffuser ramp angle, it is highly probable that it is closely related to the effect of the front wing wake in delaying diffuser flow separation. As shown in the previous section, this process is highly dependent on the path of the upstream body wake, which is extremely difficult to predict in this case.
Figure 8.8: Variation in diffuser model downforce with upstream full-span \((b/c_{rear} = 1.0)\) wing ground clearance for a number of wing incidences. \(\theta = 10^\circ, h/c = 0.06\).

Figure 8.9: Variation in diffuser model downforce with upstream full-span \((b/c_{rear} = 1.0)\) wing ground clearance for a number of wing incidences. \(\theta = 14^\circ, h/c = 0.06\).
8.4 Influence of Partial-Span Horizontal Wing

Finally, experiments were conducted with the horizontal front wing span of $h/c_{rear} = 0.60$, which is low enough to ensure that the trailing vortices pass beneath the diffuser model. Since it is known that the trailing vortices can have a beneficial effect, it was thought that this case would be of particular interest. The variation in downforce with partial-span horizontal wing ground clearances for a number of incidences is plotted for diffuser ramp angles of $\theta = 10^\circ$, $12^\circ$, and $14^\circ$ in Figures 8.10, 8.11, and 8.12.

![Figure 8.10: Variation in diffuser model downforce with upstream partial-span ($b/c_{rear} = 0.60$) wing ground clearance for a number of wing incidences. $\theta = 10^\circ$, $h/c = 0.06$.](image)

The general trends are similar to those for the greater span front wing, but with some quite major differences. Firstly, for all diffuser ramp angles, as before, the downforce level generally decreases with increasing ground clearance and incidence of the wing. However, the levels of change in downforce are much smaller than those for the greater span wing, with a maximum downforce reduction here of 13%, compared with almost 30% for the greater span wing. More interestingly, the loss in downforce for any front wing configuration decreases with increasing diffuser ramp
Figure 8.11: Variation in diffuser model downforce with upstream partial-span \( (b/c_{rear} = 0.60) \) wing ground clearance for a number of wing incidences. \( \theta = 12^\circ, h/c = 0.06 \).

Figure 8.12: Variation in diffuser model downforce with upstream partial-span \( (b/c_{rear} = 0.60) \) wing ground clearance for a number of wing incidences. \( \theta = 14^\circ, h/c = 0.06 \).
angle dramatically, such that at the greatest ramp angle, $\theta = 14^\circ$, for the majority of cases the downforce is actually enhanced by the presence of the partial-span wing by a significant amount. As with the larger span wing, the influence of increasing the wing incidence becomes less significant with increasing diffuser ramp angle.

Relating this back to the context of a Formula 1 car, these results imply that not only would adding a front wing to the car increase its downforce directly, it would also improve the downforce generated by the car underbody indirectly through its interaction with the wake. It is also beneficial to have a configuration in which changes to the front wing angle of attack do not reduce the downforce produced by the chassis, since this is often a problem with setting up a car: increasing the downforce produced by the front wing can mostly be expected to be detrimental to the rear of the car.

Centreline pressure distributions on the diffuser body for the upstream wings at an incidence of $14^\circ$ and at three different ground clearances for the diffuser ramp angles of $10^\circ$, $12^\circ$, and $14^\circ$ are shown in Figure 8.13, Figure 8.14, and 8.15.

![Graph showing centreline pressure distributions](image)

Figure 8.13: Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01, 0.03$ and $0.05$ at $\alpha = 4^\circ$. $\theta = 10^\circ$, $h/c = 0.06$
Figure 8.14: Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01$, 0.03 and 0.05 at $\alpha = 4^\circ$. $\theta = 12^\circ$, $h/c = 0.06$

Figure 8.15: Diffuser model underbody centreline pressure distributions with and without $b/c_{rear} = 0.60$ upstream wing at $h/c_{rear} = 0.01$, 0.03 and 0.05 at $\alpha = 4^\circ$. $\theta = 14^\circ$, $h/c = 0.06$
For all three diffuser ramp angles, the behaviour of the pressure on the underbody at $x/c = 0.28$ with front wing ground clearance is very similar, generally with a small increase in pressure with the wings, which falls with increasing diffuser ramp angle, and reducing ground clearance. The reason for this is most probably due to the conflicting effects of the trailing vortices and the momentum deficit in the wake. It has been established that the pressure at this point is increased by the passage of the momentum deficit in the wake beneath the model. Referring back to the results for the cylindrical rod in Figure 8.3, as the momentum deficit rises from the ground (i.e. as the rod ground clearance increases), the pressure at this station drops. Meanwhile, from the previous chapter on the effect of the trailing vortices, it is known that the vortices for this configuration, both producing an upwash at the centreline, are more beneficial to the diffuser model pressures upstream on the underbody the closer they are to the ground. Therefore, at the lowest wing ground clearances, the momentum deficit increases the pressure at $x/c = 0.28$, whilst the trailing vortices tend to reduce it. As the ground clearance of the wing is increased, the reduction in suction from the momentum deficit falls.

For the lowest diffuser ramp angle of $\theta = 10^\circ$, the pressures through the diffuser are virtually unchanged by the upstream wing. As the diffuser ramp angle is increased, the pressure recovery through it rises, such that for $\theta = 14^\circ$ there is a considerable reduction in pressure at $x/c = 0.80$ for all three configurations of the front wing. The change in surface pressure at $x/c = 0.80$ is plotted against horizontal wing ground clearance for wing incidences of $\alpha = 0^\circ$, $4^\circ$, and $8^\circ$ for diffuser ramp angles of $10^\circ$ and $14^\circ$ in Figure 8.16.

For the lower diffuser ramp angle, the pressures at $x/c = 0.80$ are virtually unaffected by the partial span wing. For $\theta = 14^\circ$, however, there is a substantial reduction in pressure at the diffuser entrance, which grows with increasing incidence at low ground clearances, and falls with increasing incidence at larger ground clearances. This is a surprising result, since it was found that for the vertical wing vortices at the same spanwise separation as the span of the partial span wing that increasing the ground clearance increases the diffuser effectiveness (see Figure 7.13). It is not known whether the vortices for this case break down at low ground clear-
Figure 8.16: Difference in diffuser model underbody centreline pressure at $x/c = 0.80$ for different upstream horizontal incidences versus wing ground clearance for $\theta = 10^\circ$ and $\theta = 14^\circ$. $h/c = 0.06$

ances, but from the total head data shown in Figure 6.1 it seems likely that they do. Even if they don't breakdown, however, this would not explain the difference in trends, since even at the greater vertical wing ground clearances, where the vortex is stable, the pressure at $x/c = 0.80$ still falls with increasing ground clearance, whilst it rises here.

The influence of the front wing wake and the trailing vortices on the boundary layer in the diffuser is likely to be a complicated function of the turbulence stress components and intensities in the wake, the state of the vortices, the distribution of axial and tangential velocity within them and the trajectories of both of these. These factors have not been fully evaluated within this study and more work is needed to fully understand the phenomena observed. However, since the change in pressure at $x/c = 0.80$ follows the same trends as for the cylindrical rod and not the vertical wings, it is probably the influence of the turbulent wake of the wing rather than the trailing vortices which dominates the phenomenon.

Compared to those found with the vertical wings, as shown in Figure 7.14, the
low pressure tracks formed by the front vertical wing trailing vortices on the diffuser model underbody are much weaker. From the total head surveys, the horizontal wing vortices are less concentrated than those from the vertical wings, which probably explains the difference. Only a weak peak in the surface pressures at $x/c = 0.29$ was found for the largest wing incidence of $\alpha = 8^\circ$. Semi-spanwise plots of underbody surface pressures at $x/c = 0.28$ with the upstream partial-span at an incidence of $8^\circ$ and ground clearances of $h/c_{rear} = 0.02, 0.04, \text{ and } 0.06$ are plotted in Figure 8.17.

![Figure 8.17: Semi-spanwise pressure distributions on diffuser model under-body at $x/c = 0.28$ with the partial-span horizontal wing at $\alpha = 8^\circ$ and $h/c_{rear} = 0.02, 0.04, \text{ and } 0.06$. $\theta = 12^\circ$, $h/c = 0.06$](image)

8.5 Key Findings

The main findings from the investigation of the influence of the partial-span horizontal wing on the diffuser model may be summarised as follows:

1. The turbulent low energy wake of the horizontal wing has two conflicting effects on the downstream diffuser model. The first is that of the momentum deficit in the wake which reduces the flow velocity over the upstream portion
of the flat underbody, raising the pressure. As the wing is lifted, less of the low momentum flow passes beneath the model and the upstream underbody pressure falls. The second is due to the turbulence in the wake, which helps to maintain attached flow in the diffuser, increasing its effectiveness and reducing the pressure at the diffuser inlet. This effect increases in importance with increasing diffuser ramp angle.

2. The upwash in the wake of the wing due to the bound and trailing vortices reduces the effective incidence of the diffuser model, increasing the pressure on the upstream portion of the model lower surface. This upwash increases in magnitude with increasing wing incidence and ground clearance.

3. A low pressure track may form on the undersurface over the vortex path, but in these experiments it was rather weak, and so unlikely to have a marked effect on the results. However more concentrated vortices, such as those from the vertical wings will have a stronger effect and induce greater suction peaks.

4. The partial-span wing is much less detrimental to the model downforce than the full-span wing due to the beneficial effect of the trailing vortices. As for the vertical wings, the lower the vortices are to the ground they are, the more beneficial so long as they pass beneath the model and not to its sides. The pressure recovery through the diffuser is also increased the most with the partial span front wing.

5. Although for the majority of cases the interaction reduced the downforce generated by the diffuser model, the interaction for certain configurations was found to significantly enhance the downforce. The performance of the diffuser model is therefore highly sensitive to the configuration of the front wing.

6. The key mechanisms involved in the interaction have been identified in this study. However, the way in which these mechanisms interact with one another is highly complex and still not well understood, so the understanding of the interaction would benefit greatly from further research.
Chapter 9

Closing Remarks, Conclusions and Suggestions for Further Work

9.1 Closing Remarks

Before drawing conclusions from the present work, the following points which have not been mentioned in the preceding chapters should be made.

9.1.1 Applicability of Results

It is emphasized that by necessity the Reynolds number at which the experimental studies were performed is significantly lower than that for a full scale Formula One car at a typical speed. The Reynolds number for the majority of the experimental results for the diffuser model, based on its length, was $6.8 \times 10^5$, whilst that for a Formula One car body (assumed length of flat underbody and diffuser $\approx 2m$) at a typical velocity ($\approx 80 \text{ m/s}$) will be of the order of $1 \times 10^7$. A number of phenomenon involved in the interaction process being studied have been introduced, and these may be influenced by varying amounts by the change in Reynolds number. For instance, whilst the effect of the upwash of the front wing on the rear body is unlikely to be sensitive to Reynolds number, the influence of the turbulent wake of the wing on the diffuser flow may be. The flow physics of the various mechanisms involved in the interaction which have been identified during the course of the research will
still be highly relevant, but some will be modified in the strength of their influence due to the greater Reynolds number. This will therefore be greatly beneficial to the understanding of flow at full scale Reynolds numbers.

### 9.1.2 Relevance to Formula One

The simplification of the car geometry is likely to have some significant implications for how the results of the project may be applied to a Formula One car. For instance, the result of using the front wing trailing vortices to induce more flow under the car to increase downforce may be significantly affected if the induced flow is brought in from the wake of the front wheels. There are in reality many flow features involved in the interaction on a Formula One car which have been neglected for the sake of simplicity, but so long as these are brought into consideration, it should be possible to use the findings of this project to the advantage of the car's performance.

Meanwhile, whilst the emphasis of the project has been on the interaction between the front wing and underbody, many of the findings are also relevant to other components of a Formula One car, such as the barge boards which generate large vortices that pass directly under the car body.

### 9.2 Conclusions

During the course of the present investigation four features of the flow in the wake of a horizontal front wing have been identified, each of which affects the diffuser model in a different way. These components and the parts they play in the interaction process are as follows.

1. **Trailing vortices.**

   The trailing vortices were found to have three main effects on the diffuser model:

   (a) **Induced flow upstream of model.**

      This was found to be one of the biggest influences of the interaction. Upwash generated between a pair of trailing vortices reduces the effective
(nose-down) incidence of a body downstream of it, reducing the downforce it generates. If they generate a downwash between them then the reverse is true, and the downforce of the rear body is enhanced. The magnitude of this vertical component of flow rises with increasing height of the vortices above the ground plane.

Meanwhile, they generate a lateral component of flow at the ground plane which is enhanced with reduced ground clearance. For the case where the vortices generate an upwash between them, this lateral component is directed towards the centreline, pushing more flow beneath the model, increasing the flow velocity, reducing the static pressure and enhancing the downforce. Again, the reverse is true for the downwash case, with an outboard component generated which reduces the downforce. For this influence to be effective, the vortices have to pass underneath and not to the sides of the downstream body.

The vortices for the upwash case (which would be generated by a negative lifting wing) should be as close as possible to the ground and remain inboard of the underbody sides for the optimum configuration.

(b) Delayed flow separation in diffuser.

It was found that the trailing vortices may delay flow separation in the diffuser, enhancing its effectiveness and increasing the pressure drop across it, resulting in an increase in downforce. When the trailing vortices pass close to the ground plane they may break down, in which case the effect is not nearly so strong. This influence is only significant when there is otherwise separation in the diffuser, and so its significance increases with increasing diffuser angle beyond that at which flow separation first occurs.

(c) Low pressure track.

When close to the lower surface of the downstream body, the trailing vortices can directly induce a low pressure track similar to that produced by the leading edge vortex of a delta wing. This effect is the least signif-
icant, however, and its influence on the lifting behaviour is secondary to
the mechanisms mentioned above.

Overall, the first effect is more significant than the other two, and the best
performance of the diffuser model was found with the vortices as close as
possible to the ground plane.

2. Upwash generated by the bound vorticity.

The upwash generated by the horizontal wing bound vorticity reduces the
effective incidence of the diffuser model, reducing the downforce produced.
This upwash grows with increasing ground clearance due to the diminishing
influence of the ground.

3. Turbulent flow.

The turbulent flow in the wake of the front wing was found to delay separation
of flow in the diffuser under certain circumstances. Again, this would only
increase the pressure rise through the diffuser if the flow there is otherwise
separated to some extent. The strength of this influence therefore grows with
increasing diffuser angle.

The mechanism governing this effect is not well understood, and the optimum
positioning of the turbulent wake has not been ascertained. It seems to depend
on a number of factors, including the diffuser angle, wing incidence and ground
clearance of both the upstream wing and diffuser model.

4. Momentum deficit in the wake.

The momentum deficit in the wake of the wing slows the flow beneath the
rear body, increasing the pressure and reducing the downforce generated. The
increase in pressure is most significant near the front of the lower surface, and
falls with increasing ground clearance of the front wing as more of the low
energy wake flow passes over, and less under the rear body.

In addition to these conclusions, a number of significant findings were also found
that do not relate directly to the initial objectives of the research. These may be
CHAPTER 9
summarised as follows.

1. The stagnation point on the rounded nose of a body lowers as the body is brought into close ground proximity, resulting in an increasing vertical velocity away from the ground of the flow approaching it. This reduces the effective incidence of the body, reducing the downforce.

2. The flow also takes an increasing lateral velocity away from the body centreline on its approach to the leading edge, reducing the underbody flow velocity and increasing the pressure. Further back, this reverses with an inboard component of flow towards the rear of the underbody. This increases the mass flow rate with distance downstream, reducing the pressure by increasing the flow velocity, but also reducing the effectiveness of the diffuser due to the non-constant mass-flow, reducing the pressure drop across it, and increasing the pressure at the diffuser inlet.

3. Reynolds number and transition become more important for bodies in close proximity to the ground since the effective ground clearance of the lower surface is reduced by the boundary layer displacement thickness. Transition from laminar to turbulent flow results in a thicker boundary layer which can have large effects on the aerodynamic behaviour of the body. In particular, a thicker boundary layer at the inlet of a diffuser can reduce its effectiveness, and separates earlier.

4. When a vortex passes close to a boundary it generates secondary vorticity on its up-going side (away from the boundary) which may separate into discrete secondary vortices. The velocity imposed on the primary vortex by the secondary is significant due to its close proximity, and so has a strong effect on its trajectory. At low ground clearances, where this production is strong, the quasi-periodic interaction produces a spiralling motion of the main vortex, which may excite the spiral mode of breakdown. Production of secondary vorticity is greater with the boundary moving at the freestream velocity than when it is stationary. For this reason, vortices in close proximity to a moving
ground plane are more prone to breakdown than those which are close to a stationary plane.

In summary, it has been found that the performance of the diffuser model is highly dependant on the positioning and configuration of an upstream horizontal wing, and can actually be improved for certain arrangements. The mechanisms involved in the interaction are, however, extremely complicated and still not fully understood, but it has been shown that correct design of the upstream wing can significantly improve performance of the flat underbody and diffuser.

9.3 Suggestions for Further Work

Little research exists in the public domain on the type of interactions which have been studied during this course of research. Although it is believed that the major mechanisms governing the interaction have been unravelled, there are still a number of important questions which remain unanswered. Further research on the topic would therefore be extremely beneficial. Suggestions for where this research may be concentrated are as follows.

1. A quasi-periodic interaction between a vortex and secondary vorticity lifted from the ground plane has been identified which seems to excite vortex breakdown. However, the mechanism governing the interaction and subsequent breakdown is not well understood. Further research is required on this area. More extensive PIV data would be of particular interest both in the vertical planes perpendicular and parallel to the freestream. Fast DPIV systems are now becoming available, and time-dependant DPIV data sets would be highly beneficial to this field.

2. It has been found that turbulence in the wake of a wing can delay separation in the diffuser, whilst blockage from the momentum deficit in the wake can be detrimental to diffuser effectiveness. The balance between these opposing factors has not been quantified, and further investigation would certainly benefit the level of understanding greatly. This should include evaluation of the
distribution of Reynolds stresses within the front wing wake, and surveys of the velocity distribution through the diffuser with and without the front wing. The influence of a turbulence grid placed upstream of the test section may also be used to investigate the influence of the turbulence without the momentum deficit associated with the wing or cylindrical rod.

3. Further wind tunnel investigations of the interaction over a wider range of Reynolds number would help to understand how sensitive the various mechanisms involved are to this parameter. The performance of the diffuser has been shown to be highly Reynolds number dependant, so it is possible that certain aspects of the interaction will be also.

4. The behaviour of the drag of the diffuser model has not been considered because changes in drag due to the upstream bodies were generally smaller than the accuracy of the internal balance. It is believed that sufficient improvement in the balance accuracy to resolve these differences would be brought about by upgrading of the associated electronics, and this data would be of great assistance in improving the understanding of the problem.

5. The influence of the diffuser model on the front wing has not been investigated during the present study, and this may be a significant consideration. It is known that the flow approaching a body near the ground takes an increasing vertical velocity away from the ground and lateral velocity away from the centreline. This will certainly have some influence on the aerodynamics of the front wing, and it would be beneficial to the subject to investigate this.
Appendices
Appendix A

Model Drawings
Appendix B

Aerofoil Co-ordinates
## B.1 Tandem wing model

<table>
<thead>
<tr>
<th>$x/c$</th>
<th>$y/c$</th>
<th>$x/c$</th>
<th>$y/c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.000</td>
<td>0.000</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>0.960</td>
<td>0.011</td>
<td>0.960</td>
<td>-0.009</td>
</tr>
<tr>
<td>0.880</td>
<td>0.027</td>
<td>0.880</td>
<td>-0.022</td>
</tr>
<tr>
<td>0.800</td>
<td>0.041</td>
<td>0.800</td>
<td>-0.035</td>
</tr>
<tr>
<td>0.720</td>
<td>0.052</td>
<td>0.720</td>
<td>-0.048</td>
</tr>
<tr>
<td>0.640</td>
<td>0.061</td>
<td>0.640</td>
<td>-0.061</td>
</tr>
<tr>
<td>0.560</td>
<td>0.070</td>
<td>0.560</td>
<td>-0.072</td>
</tr>
<tr>
<td>0.480</td>
<td>0.075</td>
<td>0.480</td>
<td>-0.081</td>
</tr>
<tr>
<td>0.400</td>
<td>0.078</td>
<td>0.400</td>
<td>-0.087</td>
</tr>
<tr>
<td>0.320</td>
<td>0.078</td>
<td>0.320</td>
<td>-0.089</td>
</tr>
<tr>
<td>0.280</td>
<td>0.078</td>
<td>0.280</td>
<td>-0.088</td>
</tr>
<tr>
<td>0.240</td>
<td>0.076</td>
<td>0.240</td>
<td>-0.087</td>
</tr>
<tr>
<td>0.200</td>
<td>0.074</td>
<td>0.200</td>
<td>-0.083</td>
</tr>
<tr>
<td>0.160</td>
<td>0.070</td>
<td>0.160</td>
<td>-0.078</td>
</tr>
<tr>
<td>0.120</td>
<td>0.064</td>
<td>0.120</td>
<td>-0.072</td>
</tr>
<tr>
<td>0.100</td>
<td>0.060</td>
<td>0.100</td>
<td>-0.067</td>
</tr>
<tr>
<td>0.080</td>
<td>0.055</td>
<td>0.080</td>
<td>-0.061</td>
</tr>
<tr>
<td>0.060</td>
<td>0.049</td>
<td>0.060</td>
<td>-0.053</td>
</tr>
<tr>
<td>0.040</td>
<td>0.041</td>
<td>0.040</td>
<td>-0.042</td>
</tr>
<tr>
<td>0.020</td>
<td>0.030</td>
<td>0.020</td>
<td>-0.027</td>
</tr>
<tr>
<td>0.016</td>
<td>0.027</td>
<td>0.016</td>
<td>-0.023</td>
</tr>
<tr>
<td>0.012</td>
<td>0.024</td>
<td>0.012</td>
<td>-0.018</td>
</tr>
<tr>
<td>0.008</td>
<td>0.020</td>
<td>0.008</td>
<td>-0.013</td>
</tr>
<tr>
<td>0.004</td>
<td>0.015</td>
<td>0.004</td>
<td>-0.007</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

Table B.1: Tandem wing model section co-ordinates.
B.2  NASA Langley GA(W)-2 LS(1)-0416

The Table B.2 gives the coordinates of the NASA Langley LS(1)-0413 wing section modified to 16% thickness. This will be referred to as the NASA LS(1)-0416 wing section. This was used for the large wing model, horizontal front wing model, and vertical wings.
<table>
<thead>
<tr>
<th>x/c</th>
<th>y/c</th>
<th>x/c</th>
<th>y/c</th>
<th>x/c</th>
<th>y/c</th>
<th>x/c</th>
<th>y/c</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.000</td>
<td>0.002</td>
<td>0.425</td>
<td>-0.104</td>
<td>1.000</td>
<td>0.0087</td>
<td>0.425</td>
<td>0.054</td>
</tr>
<tr>
<td>0.975</td>
<td>-0.006</td>
<td>0.400</td>
<td>-0.104</td>
<td>0.975</td>
<td>0.0042</td>
<td>0.400</td>
<td>0.055</td>
</tr>
<tr>
<td>0.950</td>
<td>-0.014</td>
<td>0.375</td>
<td>-0.104</td>
<td>0.950</td>
<td>0.0016</td>
<td>0.375</td>
<td>0.055</td>
</tr>
<tr>
<td>0.925</td>
<td>-0.021</td>
<td>0.350</td>
<td>-0.103</td>
<td>0.925</td>
<td>0.0007</td>
<td>0.350</td>
<td>0.056</td>
</tr>
<tr>
<td>0.900</td>
<td>-0.028</td>
<td>0.325</td>
<td>-0.102</td>
<td>0.900</td>
<td>0.0010</td>
<td>0.325</td>
<td>0.056</td>
</tr>
<tr>
<td>0.875</td>
<td>-0.035</td>
<td>0.300</td>
<td>-0.101</td>
<td>0.875</td>
<td>0.0022</td>
<td>0.300</td>
<td>0.055</td>
</tr>
<tr>
<td>0.850</td>
<td>-0.042</td>
<td>0.275</td>
<td>-0.099</td>
<td>0.850</td>
<td>0.0043</td>
<td>0.275</td>
<td>0.055</td>
</tr>
<tr>
<td>0.825</td>
<td>-0.048</td>
<td>0.250</td>
<td>-0.097</td>
<td>0.825</td>
<td>0.0070</td>
<td>0.250</td>
<td>0.054</td>
</tr>
<tr>
<td>0.800</td>
<td>-0.055</td>
<td>0.225</td>
<td>-0.094</td>
<td>0.800</td>
<td>0.0102</td>
<td>0.225</td>
<td>0.053</td>
</tr>
<tr>
<td>0.775</td>
<td>-0.061</td>
<td>0.200</td>
<td>-0.091</td>
<td>0.775</td>
<td>0.0139</td>
<td>0.200</td>
<td>0.051</td>
</tr>
<tr>
<td>0.750</td>
<td>-0.067</td>
<td>0.175</td>
<td>-0.087</td>
<td>0.750</td>
<td>0.0177</td>
<td>0.175</td>
<td>0.049</td>
</tr>
<tr>
<td>0.725</td>
<td>-0.072</td>
<td>0.150</td>
<td>-0.083</td>
<td>0.725</td>
<td>0.0218</td>
<td>0.150</td>
<td>0.047</td>
</tr>
<tr>
<td>0.700</td>
<td>-0.077</td>
<td>0.125</td>
<td>-0.078</td>
<td>0.700</td>
<td>0.0258</td>
<td>0.125</td>
<td>0.044</td>
</tr>
<tr>
<td>0.675</td>
<td>-0.082</td>
<td>0.100</td>
<td>-0.072</td>
<td>0.675</td>
<td>0.0298</td>
<td>0.100</td>
<td>0.040</td>
</tr>
<tr>
<td>0.650</td>
<td>-0.086</td>
<td>0.075</td>
<td>-0.065</td>
<td>0.650</td>
<td>0.0337</td>
<td>0.075</td>
<td>0.036</td>
</tr>
<tr>
<td>0.625</td>
<td>-0.090</td>
<td>0.050</td>
<td>-0.055</td>
<td>0.625</td>
<td>0.0375</td>
<td>0.050</td>
<td>0.031</td>
</tr>
<tr>
<td>0.600</td>
<td>-0.094</td>
<td>0.038</td>
<td>-0.049</td>
<td>0.600</td>
<td>0.0410</td>
<td>0.038</td>
<td>0.027</td>
</tr>
<tr>
<td>0.575</td>
<td>-0.096</td>
<td>0.025</td>
<td>-0.041</td>
<td>0.575</td>
<td>0.0442</td>
<td>0.025</td>
<td>0.024</td>
</tr>
<tr>
<td>0.550</td>
<td>-0.099</td>
<td>0.013</td>
<td>-0.030</td>
<td>0.550</td>
<td>0.0469</td>
<td>0.013</td>
<td>0.018</td>
</tr>
<tr>
<td>0.525</td>
<td>-0.101</td>
<td>0.005</td>
<td>-0.020</td>
<td>0.525</td>
<td>0.0491</td>
<td>0.005</td>
<td>0.012</td>
</tr>
<tr>
<td>0.500</td>
<td>-0.102</td>
<td>0.002</td>
<td>-0.013</td>
<td>0.500</td>
<td>0.0510</td>
<td>0.002</td>
<td>0.006</td>
</tr>
<tr>
<td>0.475</td>
<td>-0.103</td>
<td>0.000</td>
<td>0.000</td>
<td>0.475</td>
<td>0.0524</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>0.450</td>
<td>-0.104</td>
<td>0.450</td>
<td>0.0535</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table B.2: NASA LS(1)-0416 wing section co-ordinates.
Appendix C

Internal Wind Tunnel Balance

C.1 design

An internal 5-component balance capable of being mounted inside the diffuser model and measure lift, drag, pitching moment, yawing moment and rolling moment was designed and built for the experiments described in this thesis. The balance is based on a technology developed in the Department of Aeronautics at Imperial College. It contains six load cells, four orientated to measure lift, and the other two in drag. These are divided into two single blocks positioned either side of the balance, each containing two lift and one drag cell. An photograph of the balance is shown in Figure C.1.

The balance structure results in shear in the lift cell webs when a lift force is applied, and shear in the drag cells when a drag force is applied. The aim of the design is to give as close as possible a constant shear strain in the webs, to which shear strain gauges are attached either side to form a full bridge, thereby removing the influence of direct stress from the shear cell output. The design is such that the shear in the drag cells is not influenced by a lift force and vice versa. Finite element analysis was performed in Pro-Mechanica to confirm this during the design process. Contours of shear strain throughout the model are plotted in Figure C.2, for a pure lift and a pure downforce.
Figure C.1: Photograph of internal wind tunnel balance showing lift and drag cells.
Figure C.2: Shear strain (in the same plane as the view) in one of the two balance units when a pure lift and pure drag force is applied.
<table>
<thead>
<tr>
<th>PROJECTION</th>
<th>DRAWN: BJG</th>
<th>ALL DIMENSIONS IN MM</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SCALE 1:2</td>
<td>INTERNAL WIND TUNNEL</td>
</tr>
<tr>
<td></td>
<td></td>
<td>BALANCE</td>
</tr>
</tbody>
</table>

| 150        | 42        | 100                  |

---

**Notes:**
- Dimensions are in millimeters.
- The diagram represents an internal wind tunnel balance.
- The scale of the drawing is 1:2.
C.2 Calibration

In theory the balance measures lift as the sum of the lift cells, drag as the sum of the drag cells. Pitching moment is calculated from the forward-aft lift differential, rolling moment from the left-right lift differential, and yawing moment from the drag differential. Although efforts were made in the design of the balance to reduce the cross-coupling between these forces to a minimum, some cross coupling exists, particularly in yaw. In order to account for this, together with the different calibration factors for each load cell, and the redundancy (6 cells for 5 components) a calibration matrix is determined to relate the 6 cell voltages to the 5 force components, assuming the behaviour of all 6 cells to be linear. The calibration and data reduction is performed as follows:

\[
\begin{bmatrix}
\text{Lift} \\
\text{Drag} \\
\text{Pitching Moment} \\
\text{Rolling Moment} \\
\text{Yaw Moment}
\end{bmatrix}
\]

\[
\begin{bmatrix}
V_1 \\
\vdots \\
V_6
\end{bmatrix} = 6 \times 1 \text{ matrix of cell voltage outputs.}
\]

If the cell behaviour is linear then we can assume:

\[
v = A \cdot f
\]

Calibration is performed by loading the balance in all 5 components individually, and reading the voltages for all 6 cells for a number of loads for each component. The constants of proportionality for the voltage-load relationship are then calculated for each cell for each component. This results in a set of equations that can be written in matrix form as follows, where \( v_{ij} \) represents the volt per unit load for the \( i^{th} \) cell and \( j^{th} \) component.

\[
V = \begin{bmatrix}
v_{11} & \cdots & v_{15} \\
\vdots & \cdots & \vdots \\
v_{61} & \cdots & v_{65}
\end{bmatrix}, \quad F = \begin{bmatrix}
1 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 \\
\vdots & \vdots & \vdots & \vdots & \vdots \\
0 & 0 & 0 & 0 & 1
\end{bmatrix}
\]

It then follows that \( A = V \cdot F^{-1} \).
In order to calculate the balance voltages back to forces, the following equation is then applied:

\[ f = C \cdot v \]

where \( C \) is the calibration matrix, calculated by:

\[ C = [A^T \cdot A]^{-1} \cdot A^T \]
Consider inviscid flow through a one-dimensional duct. The pressure coefficient at any point is related to the volume flow rate per unit span by the following equation.

\[ Q = hV = hU_\infty \sqrt{1 - C_p} \]

where \( U_\infty \) is the 'free-stream' velocity, i.e. the velocity at the point where \( C_p = 0 \).

For a constant mass flow through the underbody, the pressure at any point \( C_p \) with local ground clearance \( h \) may be related to the pressure at some reference point, \( C_{p_r} \), with local ground clearance \( h_r \) by assuming inviscid flow as follows

\[ C_p = 1 - (1 - C_{p_r}) \left( \frac{h_r}{h} \right)^2 \]

Now consider a variable volume flow rate due to the distributed leakage of flow into or out of the underbody:

\[ Q = Q_0 + \int_{x_0}^{x} \frac{\partial Q}{\partial x} \, dx \]

For illustrative purposes we consider \( \frac{\partial Q}{\partial x} \) as shown in Figure 5.14

\[
\frac{\partial Q}{\partial x} = \begin{cases} 
  ax/c + b & 0 < x/c < x_d/c \\
  x_d/a + b & x_d/c < x/c < 1.0 \\
  0 & x/c = x_c/c 
\end{cases}
\]
This is an arbitrary but plausible distribution for the diffuser model considered in the study.

$x_c$ and $x_d$ are the chordwise locations of the point where there is zero leakage, and the point downstream of which the leakage is constant respectively.

Take two points as boundary conditions, $x_0$ and $x_1$ with pressures $C_{p0}$ and $C_{p1}$ respectively. For this model they are taken as the furthest upstream and downstream centreline pressure tappings, $x_0/c = 0.28$, and $x_1 = 0.95$. $Q_0$ is the volume flow-rate per unit span at $x = x_0$.

The condition at $x_c$ gives $b = -ax_c$. The equation for $Q_1$ becomes

$$Q_1 = h_1 U_\infty \sqrt{1 - C_{p1}} = h_0 U_\infty \sqrt{1 - C_{p0}} + \int_{x_0}^{x_1} \frac{\partial Q}{\partial x} \, dx$$

Evaluating this integral gives the following equation for the constant $a$.

$$a = \frac{h_1 \sqrt{1 - C_{p1}} - h_0 \sqrt{1 - C_{p0}}}{x_d x_1 + x_0 x_c - x_1 x_c - 0.5(x_0^2 + x_1^2)}$$

The volume flow rate per unit span is then calculated as follows.

For $0 \leq x/c \leq x_d/c$:

$$\frac{Q(x)}{U_\infty} = h_0 \sqrt{1 - C_{p0}} + \frac{a}{U_\infty} \left( \frac{x^2}{2} - x_c x - \frac{x_0^2}{2} + x_c x_0 \right)$$

and for $x_d/c < x/c \leq 1.0$:
\[ \frac{Q(x)}{U_\infty} = h_0 \sqrt{1 - C_p 0} + \frac{a}{U_\infty} \left( \frac{x_d^2}{2} - x_c x_d - \frac{x_0^2}{2} + x_c x_0 + (x_d - x_c)(x - x_d) \right) \]

Finally the pressure coefficient is recovered from the following equation.

\[ C_p(x) = 1 - \left( \frac{Q(x)}{U_\infty h(x)} \right) \]
Bibliography


Bagley, J. A. (1960), 'The Pressure Distribution on Two-Dimensional Wings near the Ground', *ARC R&M 3238*.


Bearman, P. W., Harvey, J. K. and Gardner, C. M. (1976), 'Preliminary Calibration of the 4.5' x 4.0' (1.37m x 1.21m) Low Speed Wind Tunnel', *Aeronautics Department, Imperial College London I.C. Aero Report 76-01.*


Elliott, M. S. (2000), An Investigation into the Wing-Rock of an 80 Degree Delta Wing, PhD thesis, Department of Aeronautics, Imperial College of Science, Technology and Medicine, London.


Hoerner, S. F. (1958), *Fluid-Dynamic Drag*, Published by the Author.

Hoffmann, P. H. (1981), ‘The WOMBAT Eight Channel Anemometer’, *Department of Aeronautics, Imperial College London I.C. Aero TN76-01*.


Wieselsberger, C. (1922), ‘Wing resistance near the ground’, *NACA TM 77*.


