Large eddy simulation for automotive vortical flows in ground effect

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Additional Information:

- A Doctoral Thesis. Submitted in partial fulfilment of the requirements for the award of Doctor of Philosophy of Loughborough University.

Metadata Record: https://dspace.lboro.ac.uk/2134/12555

Publisher: © L. Schembri-Puglisevich

Please cite the published version.
This item was submitted to Loughborough University as a PhD thesis by the author and is made available in the Institutional Repository (https://dspace.lboro.ac.uk/) under the following Creative Commons Licence conditions.

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
Large Eddy Simulation for Automotive Vortical Flows in Ground Effect

Lara Schembri Puglisevich

Department of Aeronautical and Automotive Engineering
Loughborough University

A Doctoral Thesis
Submitted in partial fulfilment of the requirement for the award of
Doctor of Philosophy of Loughborough University
June 2013

©L Schembri Puglisevich, 2013
Large Eddy Simulation for Automotive Vortical Flows in Ground Effect

by

Lara Schembri Puglisevich

Abstract

Large Eddy Simulation (LES) is carried out using the Rolls-Royce *Hydra* CFD code in order to investigate and give further insight into highly turbulent, unsteady flow structures for automotive applications. LES resolves time dependent eddies that are modelled in the steady-state by Reynolds-Averaged Navier-Stokes (RANS) turbulence models. A standard Smagorinsky subgrid scale model is used to model the energy transfer between large and subgrid scales. Since *Hydra* is an unstructured algorithm, a variety of unstructured hexahedral, tetrahedral and hybrid grids are used for the different cases investigated. Due to the computational requirements of LES, the cases in this study replicate and analyse generic flow problems through simplified geometry, rather than modelling accurate race car geometry which would lead to infeasible calculations.

The first case investigates the flow around a diffuser-equipped bluff body at an experimental Reynolds number of $1.01 \times 10^6$ based on model height and inlet velocity. LES is carried out on unstructured hexahedral grids of 10 million and 20 million nodes, with the latter showing improved surface pressure when compared to the experiments. Comparisons of velocity and vorticity between the LES and experiments at the diffuser exit plane show a good level of agreement. Flow visualisation of the vortices in the diffuser region and behind the model from the mean and instantaneous flow attempts to explain the relation or otherwise between the two. The main weakness of the simulation was the late laminar to turbulent transition in the underbody region. The size of the domain and high experimental Reynolds number make this case very challenging.
After the challenges faced by the diffuser-equipped bluff body, the underbody region is isolated so that increased grid refinement can be achieved in this region and the calculation is run at a Reynolds number of 220,000, reducing the computational requirement from the previous case. A vortex generator mounted onto a flat underbody at an onset angle to the flow is modelled to generate vortices that extend along the length of the underbody and its interaction with the ground is analysed. Since the vortex generator resembles a slender wing with an incidence to the flow, a delta wing study is presented as a preliminary step since literature on automotive vortex generators in ground effect is scarce. Results from the delta wing study which is run at an experimental Reynolds number of $1.56 \times 10^6$ are in very good agreement with previous experiments and Detached Eddy Simulation (DES) studies, giving improved detail and understanding. Axial velocity and vorticity contours at several chordwise stations show that the leading edge vortices are predicted very well by a 20 million node tetrahedral grid. Sub-structures that originate from the leading edge of the wing and form around the core of the leading edge vortex are also captured.

Large Eddy Simulation for the flow around an underbody vortex generator over a smooth ground and a rough ground is presented. A hexahedral grid of 40 million nodes is used for the smooth ground case, whilst a 48 million node hybrid grid was generated for the rough ground case so that the detailed geometry near the ground could be captured by tetrahedral cells. The geometry for the rough surface is modelled by scanning a tarmac surface to capture the cavities and protrusions in the ground. This is the first time that a rough surface representing a tarmac road has been computed in a CFD simulation, so that its effect on vortex decay can be studied. Flow visualisation of the instantaneous flow has shown strong interaction with the ground and the results from this study have given an initial understanding in this area.

**Keywords:** Large Eddy Simulation, Automotive Diffuser, Vortex Generator, Delta Wing, Rough Ground
Dedicated to my late grandfather

Austin Puglisevich
Acknowledgement

I would like to express my sincere thanks to Dr. Gary Page for giving me the opportunity to carry out this study under his supervision. I thank him for his dedication, discussion and encouragement throughout the duration of the study.

I would like to extend my gratitude to Dr. Martin Passmore who supported me with the experimental data required for the diffuser case, Dr. Andrew McMullan for several discussions on LES and training with post-processing software, and Mr Rob Flint for IT support. Computing time for the simulations in this project was provided by the Loughborough University HPC facility using the Hydra cluster.

The Department of Aeronautical and Automotive Engineering provided me with financial support through departmental funding.

Special thanks goes to my close relatives and friends for their love and encouragement, namely my partner Mark Vella, my grandmother, aunt and brothers.

Finally, I feel eternally grateful for the unconditional support that my parents, Robert Schembri and Rosanne Puglisevich, have given me throughout my years in education, enabling me to pursue my dreams and accomplish my goals, leading me to where I am today.
# Contents

## Chapter 1: Introduction

1. Development of Aerodynamics in MotorSport ............................................. 4
2. Tools for Race Car Aerodynamics Study ..................................................... 9
   1.2.1 Wind tunnel Testing ................................................................. 9
   1.2.2 Computational Fluid Dynamics (CFD) .......................................... 11
   1.2.3 Large Eddy Simulation (LES) .................................................... 12
3. Background .................................................................................................... 13
   1.3.1 Underbody Diffuser ................................................................. 13
   1.3.2 Trapped Vortices in Ground Effect ............................................. 22
   1.3.3 Rough Ground Modelling ......................................................... 26
   1.3.4 Importance of LES for automotive applications .......................... 27
4. Objectives and Thesis Content ................................................................. 35

## Chapter 2: Methodology

1. Introduction ................................................................................................. 39
2. Governing Equations ..................................................................................... 40
   2.2.1 Conservation Laws ........................................................................ 40
   2.2.2 Reynolds Averaged Navier-Stokes (RANS) ................................... 41
      2.2.2.1 Eddy-Viscosity Hypothesis ............................................... 44
      2.2.2.2 Turbulence Models .......................................................... 44
   2.2.3 Large Eddy Simulation (LES) ...................................................... 47
      2.2.3.1 LES Filtered Equations .................................................. 47
CONTENTS

3.5 Closure ................................................................. 98

Chapter 4: Vortical Flow from a Delta Wing ......................... 100
4.1 Introduction .......................................................... 101
4.2 Background .......................................................... 101
4.3 Experimental Configuration ...................................... 108
4.4 Computational Set-Up .............................................. 110
  4.4.1 Geometry ......................................................... 110
  4.4.2 Grid Generation ................................................ 110
  4.4.3 Boundary Conditions ........................................ 115
  4.4.4 Calculation ..................................................... 115
4.5 Results ................................................................. 116
  4.5.1 Subgrid-scale Viscosity Ratio ............................... 116
  4.5.2 Vorticity Magnitude ......................................... 116
  4.5.3 Surface Pressure .............................................. 120
  4.5.4 Surface Streamlines ........................................ 123
  4.5.5 Mean Axial Velocity ........................................ 124
  4.5.6 Axial Vorticity ............................................... 126
  4.5.7 Turbulent Kinetic Energy .................................... 132
4.6 Closure .................................................................. 136

Chapter 5: Vortex Generator in Ground Effect ....................... 138
5.1 Introduction .......................................................... 139
5.2 Experiments .......................................................... 140
5.3 Computational Set-Up .............................................. 143
  5.3.1 Geometry ......................................................... 143
  5.3.2 Grid Generation ................................................ 149
  5.3.3 Boundary Conditions ........................................ 154
  5.3.4 Calculation ..................................................... 155
## List of Figures

<p>| Figure 1.1 | Major Aerodynamic features of a Grand Prix car [1] | 3 |
| Figure 1.2 | Effect of ground clearance on $C_L$ (reference area unknown) [2] | 5 |
| Figure 1.3 | Downforce values between 1989 and 1997 (reference frontal area of $1.47 m^2$) [3] | 7 |
| Figure 1.4 | Effect of Aerodynamic configuration on lap time [3] | 8 |
| Figure 1.5 | Automotive vehicle with rear diffuser [2] | 13 |
| Figure 1.6 | 2D and 3D automotive diffusers | 14 |
| Figure 1.7 | Lift and Drag co-efficient curves [4] | 16 |
| Figure 1.8 | Vortices in region (b) [4] | 17 |
| Figure 1.9 | Lift curve for several diffuser angles at freestream $30 m/s$ [5] | 19 |
| Figure 1.10 | Diffuser Pressure-Recovery Coefficient Maps for fixed and moving ground [6] | 20 |
| Figure 1.11 | Contours of Lift Coefficient maps for fixed and moving ground [6] | 20 |
| Figure 1.12 | Wind-tunnel centre-line profiles of $u/U_\infty$ at $x/c = 0.5$ while varying the upstream diffuser angle for diffuser ride height $hb/d = 0.3$ [7] | 21 |
| Figure 1.13 | Flat-plate model with the vortex generators (dimensions in inches). Vortex roll up only for larger ride-heights [8] | 24 |
| Figure 1.14 | Variation of lift coefficient for different VG side-slip angle set-ups | 25 |
| Figure 1.15 | The isosurface of instantaneous second invariant of the velocity gradient $Q=3000$ on the rear slant of the body. Flow from left to right [9] | 30 |
| Figure 1.16 | Time-averaged velocity profiles at three downstream locations at midplane. LES (solid line; experiment (symbols) [10] | 31 |
| Figure 1.17 | The isosurface of instantaneous second invariant of the velocity gradient $Q=1100$ [11] | 32 |
| Figure 1.18 | Comparison of pressure distribution for RANS, LES and Volvo experiments on the ASMO body [12] | 33 |
| Figure 1.19 | Comparisons of RANS and LES solutions in the wake of semi-complex underbody diffuser at $X/L = 1.05$, $Y = 0$ without moving ground and wheels moving) [12] | 34 |
| Figure 1.20 | LES flow visualisation of the formula car [13] | 35 |
| Figure 2.1 | Construction of Median Dual Control Volume around internal node. | 52 |
| Figure 2.2 | Block meshing in ICEM CFD. | 58 |
| Figure 2.3 | Unstructured grid in CENTAUR [14]. | 59 |
| Figure 2.4 | Frustum shaped source showing grid refinement. | 60 |
| Figure 2.5 | Plots measuring the benefit of parallel processing on the Hydra cluster. | 61 |
| Figure 3.1 | Schematic diagram of wind tunnel model showing pressure tapping positions [15]. | 66 |
| Figure 3.2 | 3D wind tunnel model [15]. | 67 |
| Figure 3.3 | Cross-section of the diffuser-equipped bluff body at the symmetry plane showing size parameters. | 68 |
| Figure 3.4 | 3D views of diffuser. | 69 |
| Figure 3.5 | Domain with diffuser-equipped bluff body. | 70 |
| Figure 3.6 | Diffuser-equipped bluff body geometry showing blocking. | 73 |
| Figure 3.7 | 2D wedge shaped blocking in diffuser section | 74 |
| Figure 3.8 | 2D diamond shaped blocking in diffuser section | 74 |
| Figure 3.9 | Block-structured 20 million node hexahedral grid. | 75 |</p>
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.10</td>
<td>Subgrid scale viscosity to molecular viscosity ratio contours.</td>
<td>79</td>
</tr>
<tr>
<td>3.11</td>
<td>Time-averaged $C_p$ distribution along the underbody and top surface of bluff body.</td>
<td>81</td>
</tr>
<tr>
<td>3.12</td>
<td>Laminar test showing separation immediately downstream of front nose.</td>
<td>82</td>
</tr>
<tr>
<td>3.13</td>
<td>Bluff body in inverted position showing time-averaged $C_p$ on the surface and path taken by the 3-dimensional streamlines.</td>
<td>84</td>
</tr>
<tr>
<td>3.14</td>
<td>Contours of non-dimensional, instantaneous, streamwise velocity $u/U$ at symmetry plane ($z/W = 0.5$).</td>
<td>86</td>
</tr>
<tr>
<td>3.15</td>
<td>Time-averaged streamlines constrained to $y - z$ plane at different axial positions along the diffuser. View from front of the lower left edge of the body.</td>
<td>87</td>
</tr>
<tr>
<td>3.16</td>
<td>Instantaneous ‘streamlines’ constrained to $y - z$ plane at $x/L = 0.813$. Time difference between three images is $tU/H = 1.5$. View from front of the lower left edge of the bluff body.</td>
<td>88</td>
</tr>
<tr>
<td>3.17</td>
<td>Time-averaged contours of velocity magnitude $V_{yz}$, vertical component $v$, spanwise component $w$ and streamwise vorticity $\Omega_x$ for LES (left) and PIV (right) at the diffuser exit plane.</td>
<td>91</td>
</tr>
<tr>
<td>3.18</td>
<td>Iso-surface of pressure coefficient $C_p = -0.35$ and $C_p = -1$ coloured by non-dimensional freestream velocity comparing time averaged (left) and instantaneous (right) structures.</td>
<td>92</td>
</tr>
<tr>
<td>4.1</td>
<td>Vortex formation and vortex breakdown [16]</td>
<td>103</td>
</tr>
<tr>
<td>4.2</td>
<td>Turbulence intensity distributions and power spectral densities for the leading-edge vortex breakdown flowfield [16]</td>
<td>105</td>
</tr>
<tr>
<td>4.3</td>
<td>Total pressure coefficient contours for the delta wing $\alpha = 40^\circ$, $Re= 0.9 \times 10^6$ [17]</td>
<td>106</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>4.4</td>
<td>Evolution of breakdown location as a function of angle-of-attack ($U_\infty = 24 \text{m/s}$) [18]</td>
<td>107</td>
</tr>
<tr>
<td>4.5</td>
<td>Delta wing geometry [19]</td>
<td>108</td>
</tr>
<tr>
<td>4.6</td>
<td>Delta wing model in the wind tunnel.</td>
<td>109</td>
</tr>
<tr>
<td>4.7</td>
<td>Delta wing model for computation</td>
<td>111</td>
</tr>
<tr>
<td>4.8</td>
<td>Domain for delta wing.</td>
<td>111</td>
</tr>
<tr>
<td>4.9</td>
<td>Delta wing showing frustum shaped geometric sources for refinement.</td>
<td>112</td>
</tr>
<tr>
<td>4.10</td>
<td>15 million node grid showing frustum shaped source with node clustering.</td>
<td>113</td>
</tr>
<tr>
<td>4.11</td>
<td>Comparison of 15 million and 8 million node grids at $x/c = 0.48$ showing different levels of refinement in the vortex region and top section of the domain.</td>
<td>114</td>
</tr>
<tr>
<td>4.12</td>
<td>Mean subgrid-scale (SGS) viscosity ratio ($\mu_{SGS}/\mu$) for LES on a perpendicular plane to the wing at four chordwise stations.</td>
<td>117</td>
</tr>
<tr>
<td>4.13</td>
<td>LES instantaneous iso-surfaces of vorticity magnitude coloured by axial vorticity for three views of 70° delta wing half model.</td>
<td>118</td>
</tr>
<tr>
<td>4.14</td>
<td>DES instantaneous iso-surfaces of vorticity magnitude coloured by axial vorticity for three views of 70° delta wing G9A4 in freestream (no wind tunnel walls, no sting) [20].</td>
<td>119</td>
</tr>
<tr>
<td>4.15</td>
<td>Constant $C_P$ lines on the leeward surface of the delta wing</td>
<td>120</td>
</tr>
<tr>
<td>4.16</td>
<td>Pressure coefficient along the wing span at four chordwise stations on the leeward surface of the delta wing.</td>
<td>121</td>
</tr>
<tr>
<td>4.17</td>
<td>Comparison of streamlines on the surface of the wing.</td>
<td>123</td>
</tr>
<tr>
<td>4.18</td>
<td>Mean non dimensional axial velocity component ($u/u_\infty$) for LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations.</td>
<td>125</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 4.19  Mean non-dimensional freestream velocity ($u/U_\infty$) for the LES and experiments in a horizontal plane passing through the vortex core .......................................................... 126

Figure 4.20  View with a vertical sheet of light perpendicular to the surface of the wing [21]. ................................................................. 128

Figure 4.21  Mean non-dimensional axial vorticity component ($\Omega_xc/U_\infty$) for LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations. ............................ 130

Figure 4.22  Instantaneous non-dimensional axial vorticity component ($\Omega_xc/U_\infty$) for LES (left) and DES [22] (right) on a perpendicular plane to the wing at four chordwise stations. ............................ 131

Figure 4.23  Mean non-dimensional axial vorticity component ($\Omega_xc/U_\infty$) for LES on a perpendicular plane to the wing at four chordwise stations. ............................ 132

Figure 4.24  Mean non-dimensional total turbulent kinetic energy ($k/U_\infty^2$) for the LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations. ............................ 134

Figure 4.25  Resolved turbulent kinetic energy ($k/U_\infty^2$) along the core for the LES and Morton’s DES [20] ................................. 135

Figure 4.26  Mean non-dimensional turbulent kinetic energy ($k/U_\infty^2$) for the LES and experiments in a horizontal plane passing through the vortex core. ................................. 135

Figure 5.1  Formula 1 car showing vortices extending along the underbody [23]. 139

Figure 5.2  Red Bull Renault rb6 F1 with ‘L’ shaped vortex generators under front nose. ................................................................. 140

Figure 5.3  Typical application of large scale vortex generators on the lower surface of an open wheel race car [24]. ................................. 141

Figure 5.4  Experimental set-up showing flat plate and vortex generators. 142
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.5</td>
<td>Gothic shaped vortex generator (unrotated).</td>
<td>146</td>
</tr>
<tr>
<td>5.6</td>
<td>Domain representing underbody region of a vehicle with a vortex generator attached to underbody surface.</td>
<td>147</td>
</tr>
<tr>
<td>5.7</td>
<td>Generation of rough ground geometry to replicate a tarmac surface.</td>
<td>148</td>
</tr>
<tr>
<td>5.8</td>
<td>Domain for vortex generator over a rough ground.</td>
<td>149</td>
</tr>
<tr>
<td>5.9</td>
<td>Blocking of vortex generator domain.</td>
<td>151</td>
</tr>
<tr>
<td>5.10</td>
<td>Cross section of mesh for vortex generator.</td>
<td>152</td>
</tr>
<tr>
<td>5.11</td>
<td>Cross-section of tetrahedral mesh near rough ground surface.</td>
<td>153</td>
</tr>
<tr>
<td>5.12</td>
<td>Plan view of 0.01m section of rough ground surface.</td>
<td>153</td>
</tr>
<tr>
<td>5.13</td>
<td>Subgrid-scale viscosity ratio at $z/L = 0.17$ for stationary ground.</td>
<td>157</td>
</tr>
<tr>
<td>5.14</td>
<td>Subgrid-scale (SGS) viscosity ratio at streamwise planes through the vortex generator for different boundary conditions.</td>
<td>158</td>
</tr>
<tr>
<td>5.15</td>
<td>Side view of vortex generator showing iso-surface of mean vorticity magnitude, coloured by non-dimensional streamwise velocity $u/U_{\infty}$ for the three different ground boundary conditions.</td>
<td>161</td>
</tr>
<tr>
<td>5.16</td>
<td>View from bottom showing iso-surface of mean vorticity magnitude, coloured by non-dimensional streamwise velocity $u/U_{\infty}$ for the three different ground boundary conditions.</td>
<td>162</td>
</tr>
<tr>
<td>5.17</td>
<td>Side view showing iso-surface of instantaneous vorticity magnitude coloured by non-dimensional streamwise velocity $u/U_{\infty}$, for different ground boundary conditions.</td>
<td>163</td>
</tr>
<tr>
<td>5.18</td>
<td>Contour of instantaneous vorticity magnitude $\Omega L/U_{\infty}$ at lateral plane ($z/L = -0.17$) passing through the streamwise vortex, for different ground boundary conditions.</td>
<td>164</td>
</tr>
<tr>
<td>5.19</td>
<td>Contours of instantaneous streamwise velocity and instantaneous vorticity magnitude near the rough ground surface showing velocity vectors and grid.</td>
<td>166</td>
</tr>
<tr>
<td>5.20</td>
<td>Contours of mean streamwise velocity ($u/U_{\infty}$) at $x = 0.5l - 0.8l$.</td>
<td>167</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.20</td>
<td>Contours of mean streamwise velocity ( (u/U_\infty) ) at ( x = 0.9l - 1.2l ).</td>
<td>168</td>
</tr>
<tr>
<td>5.20</td>
<td>Contours of mean streamwise velocity ( (u/U_\infty) ) at ( x = 1.3l - 1.6l ).</td>
<td>169</td>
</tr>
<tr>
<td>5.20</td>
<td>Contours of mean streamwise velocity ( (u/U_\infty) ) at ( x = 2l - 5l ).</td>
<td>170</td>
</tr>
<tr>
<td>5.21</td>
<td>Contours of mean streamwise vorticity ( (\Omega_xL/U_\infty) ) at ( x = 0.5l - 0.8l ).</td>
<td>172</td>
</tr>
<tr>
<td>5.21</td>
<td>Contours of mean streamwise vorticity ( (\Omega_xL/U_\infty) ) at ( x = 0.9l - 1.2l ).</td>
<td>173</td>
</tr>
<tr>
<td>5.21</td>
<td>Contours of mean streamwise vorticity ( (\Omega_xL/U_\infty) ) at ( x = 1.3l - 1.6l ).</td>
<td>174</td>
</tr>
<tr>
<td>5.21</td>
<td>Contours of mean streamwise vorticity ( (\Omega_xL/U_\infty) ) at ( x = 2l - 5l ).</td>
<td>175</td>
</tr>
<tr>
<td>5.22</td>
<td>Planes showing vorticity magnitude used to calculate circulation</td>
<td>177</td>
</tr>
<tr>
<td></td>
<td>for the three different ground boundary conditions.</td>
<td></td>
</tr>
<tr>
<td>5.23</td>
<td>Plots of circulation calculated at each non-dimensional position ( x/l )</td>
<td>179</td>
</tr>
<tr>
<td></td>
<td>on planes of different radii for the different boundary conditions.</td>
<td></td>
</tr>
<tr>
<td>5.24</td>
<td>Circulation at each non-dimensional position ( x/l ) for the optimum</td>
<td>180</td>
</tr>
<tr>
<td></td>
<td>radius for each boundary condition.</td>
<td></td>
</tr>
<tr>
<td>5.25</td>
<td>Maximum vorticity magnitude in the vortex core at each streamwise plane.</td>
<td>182</td>
</tr>
</tbody>
</table>
List of Tables

Table 3.1  Grid parameters for hexahedral grids of the diffuser-equipped bluff body. ........................................ 72

Table 4.1  Grid parameters for tetrahedral grids of the delta wing. ...... 113

Table 5.1  Grid Parameters for hexahedral grid of vortex generator over smooth ground. ................................. 150

Table 5.2  Grid Parameters of unstructured tetrahedral grid near rough ground. 150
Notation

$x$, $y$, $z$ are the three Cartesian co-ordinates in the streamwise, vertical and lateral directions respectively.

$u$, $v$, $w$ follow accordingly.

Nomenclature

$C_P$ Pressure coefficient
$C_S$ Smagorinsky Constant
$e$ specific internal energy
$E$ Energy
$i$ Tensor notation ($i$-direction)
$j$ Tensor notation ($j$-direction)
$K$ Kelvin
$l_{bl}$ Baldwin-Lomax mixing length
$l_{smag}$ Smagorinsky turbulent length scale
$m$ Metres
$mm$ Millimetres
$M$ Mach number
$R$ Specific gas constant
LIST OF TABLES

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$s$</td>
<td>Seconds</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$u$</td>
<td>Axial velocity</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>Freestream velocity</td>
</tr>
<tr>
<td>$v$</td>
<td>Vertical velocity</td>
</tr>
<tr>
<td>$w$</td>
<td>Spanwise Velocity</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Non-dimensional normal distance to wall</td>
</tr>
</tbody>
</table>

Acronyms

- **AMR** Adaptive Mesh Refinement
- **AR** Aspect Ratio
- **CAD** Computer Aided Drawing
- **CFD** Computational Fluid Dynamics
- **DES** Detached Eddy Simulation
- **DNS** Direct Numerical Simulation
- **HWA** Hot Wire Anemometry
- **KERS** Kinetic Energy Recovery System
- **LDA** Laser Doppler Anemometry
- **LDV** Laser Doppler Velocimetry
- **LES** Large Eddy Simulation
- **PIV** Particle Image Velocimetry
- **RANS** Reynolds-Averaged Navier-Stokes
- **SGS** Subgrid-scale
- **TKE** Turbulent Kinetic Energy
LIST OF TABLES

Greek Characters

α    Angle of attack
β    Side-slip angle
Δ    Grid size
ε    Dissipation Rate
γ    Ratio of specific heat
Γ    Circulation
k    Turbulent kinetic energy
ρ    Density
τ    Stress tensor
μ    Dynamic viscosity
μₜ   Turbulent viscosity
ν    Kinematic viscosity
Ω    Vorticity
Chapter 1

Introduction
Aerodynamic development is a key area in the automotive industry from which significant performance improvements can be achieved. In motor sport, the focus is mainly on increasing downforce (negative lift) as this has a higher impact to reduce lap times than the effect of reducing drag. In the case of commercial vehicles the focus is on drag reduction to reduce fuel consumption and consequently carbon dioxide emissions.

In Formula 1 there are many factors that lead to reduction in lap times, namely tyre grip, mass of the car, centre of gravity, engine power, KERS (Kinetic Energy Recovery System), aerodynamics and driver fitness. Based on similar percentages of improvement, aerodynamics does not give the greatest reduction in lap times, however, it is very important since most other factors are strictly controlled by the sport’s governing body (FIA).

Tyre performance is crucial since it is the only point of contact between the car and the ground, however, tyres are supplied to the different teams by the same manufacturer and are therefore very similar. All teams aim to keep the car as light as possible, with high downforce and a low centre of gravity to enable faster cornering. Although engines are supplied by different manufacturers, they have to meet specific regulations and currently carry a ban on further development. Electronic aids are very helpful, with KERS providing a fraction of a percent improvement in lap time. This leaves aerodynamics, which although controlled quite heavily, is still an area from which competitive advantage can be gained.

Figure 1.1 outlines the aerodynamic features of an open-wheeled, open-cockpit, single-seater race car. Achieving the optimal aerodynamic balance for the car is important as too much downforce on the rear or front may lead to oversteer or understeer respectively. The underbody diffuser has been the subject of research, especially in recent years, as the underbody is responsible for 40% – 50% of the downforce gen-
erated by a Formula 1 car. Despite this, the underbody is not the greatest source of drag, with values of downforce in this region increasing at much higher rates than that of drag over the past few years. This makes it a very efficient aerodynamic component. Finding a compromise between aerodynamic efficiency and stability is often the challenge in aerodynamic design [3]. An understanding of the vortical structures that dominate the flow field in the underbody region of a race car is essential for flow management and control.

The aim of this thesis is to give insight into the highly unsteady, 3D vortical flow features present in the underbody region by means of unsteady LES calculations capable of predicting the time varying flow. Different cases representing the flow in this region were set up, namely a diffuser-equipped bluff body in ground effect to represent a race car with a rear diffuser and a vortex generator in ground effect to represent the small aerodynamic components found at the front of a race car (under the nose or on the front wing) that generate vortices that extend along the underbody region of the car. Due to limited literature available on the latter case, LES on a delta wing was carried out as a preliminary step as the streamwise vortices coming off a vortex generator are analogous to the leading edge vortices of a delta wing [8].
1.1 Development of Aerodynamics in MotorSport

In a review by Katz [2], the author considers downforce generation and stability as the main issues in the aerodynamic design of a race car. The front wing, rear wing and underbody are responsible for generating most of the downforce on an open wheel race car [3], which should ideally act through a constant centre of pressure to avoid change in pitching moment, improving stability when cornering [2]. Higher levels of downforce allows the car to carry higher speeds around the corners and achieve better corner exit speeds. However, this carries a drag penalty on the straights resulting from lift induced drag. An open-wheel single-seater is capable of generating lateral acceleration above 3g [2].

Aerodynamic efficiency is achieved by generating maximum downforce (negative lift) at the least possible drag [1]. The compromise between the two will establish the aerodynamic set-up for a particular track. A tight track such as Monaco allows cars to have bulkier aerodynamic wings which generate more downforce, since drag penalties are not as significant as for higher speed tracks. On high speed tracks such as Monza, drag penalties are higher due to the higher achievable velocities on the long straights. This balance needs to be integrated with the vehicle dynamics (i.e. inertia, suspension and tyre characteristics) of the car in order to achieve the optimal set-up.

The importance of aerodynamics was only realised during the 1960’s, and is today the main factor responsible for giving the edge in performance between race cars [2]. Prior to that, design was focused on drag reduction as maximum speeds were achievable throughout a significant distance of the tracks used for racing. This kind of development could easily be done by measuring top speeds achieved and did not require the instruments used in aerodynamic development today.

In the early 1960’s Chaparral Cars experimented with body shape, wings and
fan-induced suction which resulted in increased tyre grip through induced downforce. Although the wind tunnel had been around for a long time in aircraft development, it was only introduced to automotive design at this time. Designers started to experiment with movable wings which they mounted onto the cars and acted like the inverted wings of an aircraft, generating downforce instead of lift. They also realised that a streamlined underbody lowered the pressure under the car.

In the mid 1970s Lotus introduced the ground effect phenomena discovered by Chaparral to their F1 car [25]. They also had sliding side skirts all the way to the ground, sealing off the gap between car underbody and the ground [2] and the flow beneath the vehicle was referred to as a Venturi flow which carried very low pressure. This was quite dangerous because once the seal was damaged, the car would experience a great loss of downforce and have a significant affect on stability [26]. Figure 1.2 shows the effect of side-skirt ground clearance on downforce, with downforce increasing significantly as the gap between the ground and the skirt is reduced. These were later banned due to the fact that they made the car so unsafe and the body aerofoil concept was reintroduced.

Regulating bodies continuously impose limits on the magnitude of downforce generated as it greatly effects the braking, accelerating and cornering performance. Fig-

Figure 1.2: Effect of ground clearance on $C_L$ (reference area unknown) [2]
1.1. DEVELOPMENT OF AERODYNAMICS IN MOTORSPORT

Figure 1.3 shows how downforce generation progressed between 1989 and 1997, with a steep increase in downforce values up until 1992 due to increased understanding and research of aerodynamic performance. Regulation changes were the cause of large drops in downforce, especially after the fatal accident of Ayrton Senna in 1994. The front wing was raised higher and the rear wing was moved forward in order to reduce downforce. The extended diffuser was cut back and its influence on performance was reduced from about 70\% to 40\% [27].

Since 1983 the underbody of a Formula 1 car has been flat in the region between the front and rear wheel axles with an angled surface on the rear of the vehicle referred to as the diffuser [25]. The diffuser section is used to further lower the pressure beneath the vehicle and generate higher values of downforce. During recent years a lot of importance has been given to the design of the diffuser region and it has subsequently been a very controversial topic. During the Formula 1 season of 2011, the FIA announced that it would ban off-throttle blown diffusers, which ‘blew’ exhaust into the diffuser even when the driver was not on the throttle, so enhancing diffuser performance.

Other regulations changes in the past few years which relate to downforce were the change in geometry of the rear wing in 2009, and the ban of the ‘F-duct’ system in 2011. Formula 1 cars had to have narrower rear wings in order to produce less downforce, in an attempt to create more overtaking opportunities. The ‘F-duct’ system required driver movement (hand or knee) to operate an air duct in the cockpit that altered the airflow to the rear wing, stalling it on the straights, reducing downforce and hence drag, to achieve higher top speeds. This was condemned unsafe by teams who were not using it and was consequently banned by the FIA. In 2011 a drag reduction system (DRS) was introduced which opens a flap on the rear wing to reduce drag on the straights. In its closed position the rear wing acts as an inverted wing generating downforce to get round the corners quicker, whereas on the straight the
1.1. DEVELOPMENT OF AERODYNAMICS IN MOTORSPORT

![Graph showing downforce values between 1989 and 1997.](image)

Figure 1.3: Downforce values between 1989 and 1997 (reference frontal area of 1.47m²) [3]

flap is opened to let air through, losing the additional downforce and hence achieving higher top speeds.

The table in figure 1.4 translates different aerodynamic configurations into improvements or otherwise of lap times for a Formula 1 car around a typical circuit. Although the improvements listed may seem small, they result in a significant amount of advantage over the period of a complete race. According to the figures in Table 1, an increase of 10% downforce gives a similar lap time improvement to a reduction of 10% drag, even though increased downforce normally results in increases drag. In motor racing such as Formula 1, downforce generation is normally given more importance than drag reduction as higher cornering speeds and stability will allow the driver to get round the track quicker. In the case of road performance cars and commercial vehicles drag reduction is more important as it allows higher top speeds in the former case and reduces carbon emissions in the latter.

The flow in the underbody region of an open wheel single-seater is highly turbulent and separated and accounts for 50% of the total downforce generated [25]. The present study focuses on the aerodynamics in this region by carrying out LES calculations for different underbody cases. The first case investigates the flow through an underbody
1.1. DEVELOPMENT OF AERODYNAMICS IN MOTORSPORT

![Table showing the effect of aerodynamic configuration on lap time and maximum speed.](image)

Figure 1.4: Effect of Aerodynamic configuration on lap time [3]

and single-channel diffuser. The second case, a prerequisite for the third case, looks into the vortices over a delta wing as it produces similar vortices to those produced by an automotive vortex generator (fin) in ground effect, which makes up the third case. The third case looks at a vortex generator in close proximity to a smooth ground and a rough ground, in an attempt to find the decay factor caused by the cavities in the tarmac.

Vortex generators are commonly placed in the underbody region in competitive motor sport racing to sustain higher levels of downforce for reasons which will be discussed in this study. Although to the author’s knowledge, the ground plane has always been modelled as a smooth surface both in wind tunnel and CFD tests, the cavities in the tarmac must have a significant effect on the vortex structures present in close proximity to the ground. This difference in ground modelling contributes to discrepancies between wind tunnel, CFD and track testing results. LES is performed on these cases of isolated components due to the computational requirements of this method. The aim is to gain a better understanding of the flow structures present which will allow better flow management and control.
Aerodynamic studies can be split into two main categories: 1) Performance Analysis and 2) Flow development and understanding. The first involves testing different components or configurations for a certain model in order to obtain lift, drag and moment values which helps the designer in choosing the correct angle or geometry for a component and is common practice in industry. The second category normally involves off surface measurements and is more about understanding the flow mechanisms present to enable better control of the flow. Wind tunnel testing and CFD (Computational Fluid Dynamics) are the two most commonly used tools for the above mentioned aerodynamic studies, alongside track testing. In an attempt to make racing teams more equal by reducing costs, the FOTA (Formula One Team’s Association) have imposed a restriction on a balance of CFD usage and wind tunnel testing.

### 1.2.1 Wind tunnel Testing

For many years flows have been studied experimentally by using wind tunnels and are still the strongest aerodynamic design tool being used today. In the late 1960s and 1970’s quarter-scale model wind tunnels were being used by Formula 1 teams and by the late 1970s engineers realised the importance of moving ground facilities in order to reproduce the correct track boundary conditions. Today the small scale wind tunnels are not enough and all teams have access wind tunnels where they can test 40-50% scale models at 40-70m/s [3], which allows variation of body shapes and different set-ups to be tested at feasible costs as full scale models would cost too much.

Wind tunnels provide a means of taking aerodynamic measurements on a stationary object in a controlled environment. Air is made to move past the object so that force and pressure generated by the flow can be measured easily. They are equipped
1.2. TOOLS FOR RACE CAR AERODYNAMICS STUDY

with instrumentation such as a multiple component force balance or a strain gauge to measure forces and moments in all directions, and high speed pressure scanners for surface pressure measurements, all of which are capable of giving accurate performance measurements. Although many successful tests have been carried out in wind tunnels, the greatest difficulty lies in visualising turbulent flow characteristics, which is where techniques such as Hot Wire Anemometry (HWA), Laser Doppler Anemometry (LDA) and Particle Image Velocimetry (PIV) come into practice. These techniques are capable of giving velocity, energy vorticity and turbulence levels in the flow.

The wind tunnel at Loughborough University is currently equipped with PIV instrumentation which was used to take off-surface measurements in a recent study [15] on a bluff body diffuser. PIV is a technique used to give quantitative flow information with high spatial resolution. It does this by mapping velocity vectors of a particular flow field taken from a number of instantaneous flow images. The result is a vector field of the flow. The flow is seeded with an amount of particles which are illuminated by a pulsed laser sheet. Images are then recorded by means of a high speed video camera. A cross correlation is done to track each particle and determine the direction of motion. This image capturing is repeated to produce a series of vector fields resulting in a video of the motion of the flow. Particle Image Velocimetry is capable of resolving the velocity components, giving information on turbulence in the flow field and captures the unsteadiness of the flow. However, the technique may require large observation areas which are costly to run. Although it has better spatial resolution than HWA and LDA since it does not require interpolation of point measurements, it cannot have the temporal resolution which is comparable to the small timesteps used in LES.

There are other problems which arise in wind tunnel tests such as the difficulty to run tests at the correct Reynolds number since scaled models are used. Correct
1.2. TOOLS FOR RACE CAR AERODYNAMICS STUDY

Reynolds number modelling is important in order to reproduce the same flow characteristics present on a full size vehicle as scaling effects can greatly affect the accuracy of the results. Larger models give more accurate results not only due to correct Reynolds number modelling, but it also enables detailed modelling of very small components. The inability to test true cornering in a wind tunnel is another limitation. Struts, mounts or levers supporting the model may affect the results of a wind tunnel simulation and it is not always straightforward to take measurements in the flow field. Measurement probes can disturb the flow under study or optical access might not be possible. Hence, to optimise designs, CFD is a useful tool which can be used to complement wind tunnel tests.

1.2.2 Computational Fluid Dynamics (CFD)

CFD is a very useful tool which is capable of both calculating performance parameters and visualising the flow. Geometry changes can be carried out relatively quickly since unlike wind tunnel testing, it does not require the models to be built for the tests to be carried out, which can be quite costly and time consuming. Numerical methods have been used for several years now to model the flow around a Formula 1 car. They are used to solve modelled versions of the Navier-Stokes equations [28], which represent the flow. CFD solves these equations numerically by obtaining numerical approximations to the solution of the governing fluid flow equations, allowing a more detailed study of the interaction of complex flow structures present. Generally, velocity profiles from the wind tunnel are imposed at the boundaries of the CFD domain so that the conditions are comparable.

CFD has become increasingly important in the development of Formula 1 car aerodynamics with more affordable high-performance computational power, however, it still has its limitations. Very fine meshes are required for the numerous small
1.2. TOOLS FOR RACE CAR AERODYNAMICS STUDY

components found on racing cars, which often makes it impossible to model the car as a whole. CFD is dependent on the turbulence model being used and hence correlation with experiment is still required. Rather than looking at CFD as a replacement to wind tunnels, it is an effective tool in identifying flow problems in particular regions which can then be further investigated in the wind tunnel.

This thesis centres around a CFD method called Large Eddy Simulation (LES) which although not used in Formula 1 and most industrial applications due to its large computational requirements, has become feasible for research purposes as a result of more affordable computational power.

1.2.3 Large Eddy Simulation (LES)

The need for LES arises from the failure of the RANS turbulence models [29] to model highly turbulent, 3d separation accurately. LES focuses on the larger eddies of a turbulent flow which are highly dependent on the geometry of the model. The small eddies are nearly isotropic and are believed to have universal behaviour. The larger energy-carrying eddies are computed, while the smaller eddies are represented by a subgrid-scale (SGS) model. Although LES requires modelling of the SGS, it is assumed that in highly separated flows the influence of the small scales are minimal and can easily be modelled since it is the large scale eddies which dominate the flow.

LES is very costly since it requires very fine grids and computational resources, however, it is effective in solving complicated unsteady flow problems providing a solution that changes in time [29] [30]. LES is especially effective in flows of large 3D separation such as the automotive underbody diffuser case, automotive vortex generator case and delta wing case chosen for this study.
1.3 Background

1.3.1 Underbody Diffuser

The underbody diffuser refers to the swept section of the underbody as shown in figure 1.5 that releases the accelerated air beneath the car [27]. In Formula 1, diffusers had a much greater role when the diffuser extended beyond the rear axle of the car. Today, the region between the front and rear wheel axles is occupied by a flat, rigid, impervious undertray, as imposed by the FIA regulations. Immediately downstream of this is a diffuser which is used to lower pressures beneath the car, resulting in higher downforce. The diffuser alters the entire pressure distribution beneath the car as it provides higher suction in certain areas.

As mentioned previously, about 40% of the downforce is generated by the undertray and bodywork of the car [3]. Improving the design of the diffuser and being able to control the flow in this region can be very beneficial as it does not necessarily lead to a significant increase in drag. Aerodynamic stability is just as important as trying to maximise downforce and reduce drag as it directly affects the car driveability.

The undertray has a low aspect ratio and is a region of highly turbulent, vortical,
1.3. BACKGROUND

(a) 2D simplified diffuser geometry [6]

(b) 3d diffuser with trailing vortices [2]

Figure 1.6: 2D and 3D automotive diffusers

three-dimensional flow, so its limiting factors should be well predicted. If the aerodynamicist can predict stall conditions then the undertray can be designed to operate at an optimal set-up. The rear wing can be used to lower the base pressure of the diffuser so that it can generate lower pressures in the underbody region. Higher angle diffusers can sustain lower magnitudes of pressure but tend to stall at higher ride-heights [26].

When considering the underbody in a two-dimensional sense (figure 1.6a), flow accelerates between the ground and the body, just like it would in a narrow pipe. Flow is delivered to a fixed exit pressure. For continuity to be obeyed, the flow accelerates at the diffuser inlet and pressure drops. In reality the diffuser is a three
1.3. BACKGROUND

dimensional aerodynamic component. As observed in figure 1.6b, the diffuser has a three-dimensional vortical effect. Flow enters from the sides of the diffuser and rolls up into two rotating structures.

Several experiments have been conducted to observe how parameters such as ride height, diffuser length and diffuser angle influence pressure, downforce and drag. Cooper et al. [31] examined the physics of a single-plane underbody diffuser at a range of angles and ride heights to identify its effect on downforce and drag. The authors identified three downforce mechanisms: i) ground interaction: downforce is generated as the model is placed in close proximity to the ground, ii) upsweep: the cambered shape that the diffuser gives the model (inverted wing), iii) diffuser pumping: a downforce mechanism driven by pressure recovery along the diffuser.

Several other studies [31, 5, 4] have commented about ‘ground effect’ and confirm that as the model is brought close to the ground, downforce increases as flow accelerates beneath the body and pressure decreases. Downforce levels continue to rise with reduction in ride height until a critical value is reached, below which there is a loss of downforce as the diffuser stalls.

In studies by Senior and Zhang [32], Ruhrmann and Zhang [5] and Zhang et al. [4], the authors identified four stages of downforce generation. As the model was moved from a height where it began to be influenced by the ground to a height where downforce was lost, it went through the following stages as observed in Fig. 1.7. Firstly (a) the force enhancement as the body is lowered into the second stage (b), which consists of a plateau region in which forces are constant over a range of ride heights. This is followed by a linear increase in downforce and drag until a point where maximum force is reached. Below this point, in the third stage (c), force reduction is observed until a further reduction in ride height leads to the final stage (d), characterised by a complete loss of downforce. In previous studies, such as that by
1.3. BACKGROUND

Cooper et al. [31], surface pressures and forces had already shown similar downforce and drag behaviour.

Senior and Zhang [32] and Zhang et al., [4], link these stages to the presence of 3D counter rotating vortices which were observed using Laser Doppler Anemometry (LDA). Figure 1.8a shows a 3D image of the vortices on the upswept surface. These form as flow enters beneath the side plates of the diffuser and wind up into counter-rotating vortices. In the force enhancement region downforce and drag increase as the model is lowered. The flow is symmetric, vortices are concentrated, having a high axial speed core and turbulence levels are low. The vortices grow as the model is lowered.
1.3. BACKGROUND

(a) Oil flow visualisation of surface streak lines on the upswept surface, flow from left to right

(b) Mean velocity vectors showing vortices behind diffuser

Figure 1.8: Vortices in region (b) [4]

In the plateau region the vortices have increased in size and axial speed is low. Vortex turbulence levels are high. In the force reduction region the flow becomes asymmetric as one of the vortices breaks down due to 3D flow separation at the diffuser inlet. Flow reversal is present, whilst a weak vortex still exists. In the loss of downforce region, there is not enough mass flow through the system. The differences behind the force enhancement processes in the first two stages may be due to a change in vortex structure.

It is often believed that the suction at the diffuser inlet and the suction on the
1.3. BACKGROUND

sides of the diffuser ramp (due to the attached vortices on the sides) generate most of the downforce in the diffuser. Previous diffuser studies fail to mention the mechanism by which the change of angular momentum as the flow changes direction around the sharp diffuser inlet and at the exit of the diffuser generates downforce. The vortices play a role in keeping the flow attached to the ramp, allowing the diffuser to operate at higher angles without stalling, rather than being the source of downforce generation.

Ruhrmann and Zhang [5] found that low angle diffusers show a steady increase in downforce up until very low ride heights, while higher angles generate higher downforce values and experience a sudden loss of downforce at larger ride heights (Fig. 1.9). Cooper et al. [31] attributed loss of downforce at very low ride heights to the fact that at this point the boundary layer thicknesses of the ground and the model occupy most of the region below the model.

A study on a 17 degree angle diffuser by Senior and Zhang [32] suggests that flow separation at the diffuser inlet leads to loss of suction. In a later study by Ruhrmann and Zhang [5], tests done on 5, 10, 15, 17 and 20 degree angles, state that for the higher angles it is the separation at the inlet together with vortex break down process that causes loss of pressure recovery and hence loss of downforce. For the 5 degree angle, separation does not occur and it was suggested that vortex breakdown may be the cause. However, at very low ride heights, it would be sensible to say that interaction of the boundary layers restricts the amount of flow passing through the underbody region resulting in momentum change being minimal (since momentum is dependent on mass flow) and hence downforce generation is greatly reduced.

In a study on different diffuser angles and ride heights, Jowsey and Passmore[15] found that separation close to the inlet gave rise to higher drag values. They observed that at a given ride height, the 16 degree diffuser gave higher drag than the 13 degree diffuser, which was probably due to the separation rather than increased vortex
1.3. BACKGROUND

Figure 1.9: Lift curve for several diffuser angles at freestream 30m/s [5]

strength as downforce values were similar for both.

The largest difference between [31] and [32] is the effect of the ground simulations. Cooper et al. [31] find that both ground simulations, fixed and moving, produced similar downforce values with the largest differences being at very low ride heights. Drag differed mostly for angles below 9.64 degrees. However, in a study on the 17 degree diffuser [32] the authors outline the importance of the moving ground with great differences in maximum downforce and position of maximum downforce even at this high angle. The discrepancy increases at lower ride heights. This emphasise the importance of moving ground simulation for race car applications, since race cars normally ride very low to achieve higher levels of downforce.

In a later study, Cooper et al. [6] present pressure recovery maps (Fig. 1.10 and Fig. 1.11), as an aid to narrow design decisions on diffuser length and area ratio for race cars. The results apply to diffuser lengths of 0.25L, but are applicable to a range of diffuser lengths for which the diffuser flow is isolated from the entry flow, i.e. the entry flow is fully recovered and does not interact with diffuser flow. Diffusers on Formula 1 cars are limited to the region between the rear axle and back of the car, making these maps applicable. The main difference between ground simulations is that the maximum pressure/lift coefficient for the moving ground tests fall at a
1.3. BACKGROUND

Figure 1.10: Diffuser Pressure-Recovery Coefficient Maps for fixed and moving ground [6]

Figure 1.11: Contours of Lift Coefficient maps for fixed and moving ground [6]

lower area ratio parameter than for the fixed ground. With a moving ground, the boundary layer is smaller than for a fixed ground. This means that for a given area ratio, the moving ground simulation will have a greater effective area ratio than the fixed ground. Hence, the same pressure recovery can be achieved at a lower area ratio.

Studies by Soso and Wilson [33] [7], investigate the effect of an upstream diffuser on a downstream front wing. This is representative of a two Formula 1 cars following each other. Although the studies mainly focus on the effects on the front wing of a trailing car, they give some insight into the wake of the diffuser. It was found that the greatest flow deficit in the streamwise direction was greater at locations closer to the
1.3. BACKGROUND

centre of the wind tunnel. This was also the region of highest upwash, produced by the counter-rotating vortices. The greatest turbulence was in the centre of the wake. Varying the diffuser angle resulted in the highest freestream deficit at the low angles (5 and 10 degrees) and the 16.7 degree diffuser inducing the least deficit (see figure 1.12).

Figure 1.12: Wind-tunnel centre-line profiles of $u/U_\infty$ at $x/c = 0.5$ while varying the upstream diffuser angle for diffuser ride height $hb/d = 0.3$ [7]
1.3.2 Trapped Vortices in Ground Effect

Section 1.3.1 explains how the underbody diffuser reduces the pressure beneath the car resulting in increased downforce. This is done most efficiently if ‘clean air’ is supplied to the underbody region. Aerodynamic devices ahead of the underbody condition the flow entering the underbody region. The front wing, winglets and other vortex generating devices such as the ‘turning vanes’ under the front nose, generate vortices which must be managed correctly.

The large ‘turning vanes’ under the nose are used to generate large vortices from the lower edge that interact with the smaller vortices from the front wing and direct them outwards away from the underbody. They also generate vortices from the upper edge which enter and run along the length of the underbody region. If positioned at the right angle, these vortices will keep the flow attached and provide the diffuser at the back with high energy flow capable of producing high levels of downforce.

To date, little work has been done on the study of vortices in ground effect in relation to ground vehicles. Studies on aircraft wing in ground effect include that of Katz and Levin [34], wherein the authors recorded increased values of lift on a delta wing with the presence of a ground plane. Rossow [35][36][37] carried out some experiments on an aircraft wing in ground effect with the aim of generating very high values of lift particularly for V/STOL conditions. Rossow [35] proposed a set up with flaps which were used to generate vortices parallel to the leading edge, and suction orifices were meant to organise and hold the vortex at a fixed location. The two dimensional analysis indicated lift coefficients up to 10, however, in the experiments, an externally trapped steady state vortex was difficult to generate and then hold at a particular position.

Another configuration proposed by Rossow [36, 37] included a second fence at
1.3. BACKGROUND

some distance behind the first in order to trap the vortex in between the two fences, preventing mass removal from the vortex core and no drag associated with the vortex trapping. In practice, application of these theories proved difficult, especially across the span of a full sized wing. Improvements in $C_L$ were limited to a small range of low angle-of-attack highly swept wings. On the other hand, ground vehicles operate very close to the ground and similarly trapped vortices between the vehicle underbody and ground may have quite a pronounced effect.

A few attempts have been made to model vortices in the underbody region of a race car [38, 8, 24]. Experiments were carried out on a flat plate representing the underbody, with a pair of vortex generators mounted on each side to generate streamwise vortices (see figure 1.13). Initially, the idea of having two vortex generators on each side was to trap the vortex in between just as Rossow [36, 37] did in his experiments. A ground plane was placed parallel and above the plate. This inverted set-up enabled the flat-plate to rest on the force balance and direct force measurements could be taken. In [8], the authors investigate rectangular vortex generators at varying ride height and orientation at a Reynolds number of $2.7 \times 10^6$ (based on flat plate length). Figure 1.14a shows that downforce enhancement is achieved with reduction in ride height and larger incidence angles to the flow. The authors make an analogy between the downforce enhancement at larger side slip angles and the lift augmentation of delta wings at high angles of attack. In both cases, at a critical angle this force enhancement is lost due to vortex breakdown, at which point this phenomena has more influence than at the lower angles.

Flow visualisation revealed that at large ride heights, the two vortices coming off the two VGs (on each side) were interacting with each other, whereas at lower ride heights, the VGs seemed to untwist themselves and move closer to the plate, resulting in increased downforce. This was also observed in [38], were it is also mentioned that the vortices moved closer to the vehicle’s surface as ride height was reduced. It also
1.3. BACKGROUND

emerged that having the inner VG at a low angle and the outer VG at a higher angle generated more downforce and gave a better L/D ratio than having the two parallel to each other. This is probably due to less interaction of the two vortices (on each side), however, further investigations must be carried out to gain a better understanding of the flow. The authors found that the role of the second vortex generator on each side, is for increased downforce rather than to trap the vortex as previously used by Rossow [36, 37], as a stable vortex was generated successfully even with one VG on each side.

Investigations in [24] looked at different VG shapes and sizes, where the authors concluded that lift was related to VG surface area. The longer VGs resulted in more downforce as did the rectangular and gothic shapes in comparison to ogive and triangular shapes (see figure 1.14b). The triangular VGs are more unlikely to lose downforce at very low ride-heights. References [38], [8] and [24] present mainly force data on the underbody vortex flow, which is not enough to derive certain conclusions about the flow. The investigations provide good grounds for further study, where CFD analysis, especially LES, of trapped vortex flow could provide more information on the structure and breakdown of the tip vortices, with the intention of giving a better understanding of the above mentioned observations. The triangular vortex...
1.3. BACKGROUND

(a) Yaw angle variation for parallel VGs on each side, \( d = 25 \text{mm} \) [8]

(b) Several VG shapes (\( \beta = 20/30 \) stands for inner VGs at 20 deg, and outer VGs at \( \beta = 30 \text{ deg} \)) [24]

Figure 1.14: Variation of lift coefficient for different VG side-slip angle set-ups

generator can be compared to a delta wing which has been studied extensively and may provide us with more information on the formation of trailing vortices which form off the leading edge.

In his RANS CFD study on vortices in ground effect, Chambers [39] shows relatively good correlation with Katz’ experiments for a vortex generator at a 35° side-slip angle, with the highest discrepancies occurring at the lowest ride height. This is probably due to the fact that at this ride height a steady vortex is not able to form and an unsteady calculation such as LES may be more suitable to resolve the unsteady,
1.3. BACKGROUND

three dimensional structures.

1.3.3 Rough Ground Modelling

To the author’s knowledge, this is the first study that attempts to resolve the flow near a physically rough ground surface for automotive applications. Existing wind tunnel facilities consist of smooth floor surfaces, some of which are equipped with moving belts which is necessary for accurate modelling of automotive flows. Numerical studies are also normally carried out on smooth grounds, since most CFD in industry uses the Reynolds-Averaged Navier Stokes, and LES would require very fine grids to be able to resolve the small structures in the cavities of a rough surface.

Rough surface modelling is a topic that comes up in atmospheric modelling which is used for weather forecasts. This roughness is of a much larger scale as it is meant to represent the landscape over the ground, ranging from trees and shrubs to mountains and skyscrapers. These simulations usually make use of specially developed wall models or subgrid scale models that will account for the smaller scales close to the ground. The design of the subgrid-scale model depends highly on the scale of the simulation, with large scales requiring very specialised models [40] that are based on statistical observations and data from finer scale simulations.

Dong et al. [41] carried out wind tunnel tests to determine the aerodynamic roughness length of gravel surfaces of different size and coverage at different free-stream velocities. The authors describe the aerodynamic roughness as the ‘no slip’ layer between the airflow and the boundary. They found that near-surface flow behaved differently over different gravel surfaces, and is a function of gravel size, gravel coverage and freestream velocity. Aerodynamic roughness length decreased with freestream velocity and increased with more coverage, with its maximum being reached at 40%
An earlier study by Raupach et al. [42] also involved wind tunnel tests on a regularly arrayed rough surface. The authors identify two surface influences on the mean flow field: wake diffusion and horizontal inhomogeneity. The wake diffusion in the roughness sublayer (the height of which was established in the experiment) caused the vertical velocity gradients to differ from that of the semi-logarithmic region which implies the importance of rough surface modelling. Lateral velocity profiles showed well defined spatial periodicities in both streamwise and vertical velocity profiles showing that horizontal inhomogeneity is non-negligible near the ground. They also found that the scatter introduced by the horizontal inhomogeneity was not enough to mask the wake diffusion effect completely.

1.3.4 Importance of LES for automotive applications

Transport equation turbulence models in RANS calculations are capable of providing accurate, converged results for attached flows. They do not require very fine grids and can be easily applied to complex geometry, with grid independent solutions achieved on finer grids. However, convergence becomes an issue when dealing with unsteady, large scale separation, 3D flows. The turbulence models find difficulty in modelling correctly the large-eddies in highly separated flows and the solution can only ever be as good as the turbulence model.

Large Eddy Simulation is capable of giving a time-dependent solution and resolve the eddies in the flow but usually requires very fine grids, especially with increasing Reynolds number. This makes it costly and infeasible for complex geometries. Inadequate grid spacing has often lead to the suppression of turbulent flow features close to the geometry. The biggest challenge for RANS is modelling the regions of massive
separation such as wheels and flap edges, where the momentum carrying turbulent eddies are assumed to be around the same scale as the geometry (e.g. wheel or flap chord) [43].

Turbulent scales around a ground vehicle range from those the size of the vehicle to microscopic ones in the boundary layer. The high Reynolds number of such flows leads to very fine scales that have to be resolved, which requires higher resolution and higher computing power. This makes LES for ground vehicles at operating velocities infeasible and for this reason simplified models are normally used to isolate and understand generic flow problems rather than solve for a flow around specific geometries.

One approach to reduce resolution requirements has been to use detached eddy simulation (DES), which uses RANS in the boundary layer, where LES would otherwise require very fine grid spacing to resolve the small scale eddies, and LES in the separated regions. This sounds like an ideal compromise where a time-dependent solution can be achieved on a reasonable grid. However, the solution is dependent on the grid, the turbulence model selected, placement of the matching line between RANS and LES regions, correct boundary conditions between the two methods may be difficult to achieve, and there is also the added complication of solving two sets of governing equations simultaneously [44].

Some studies [44, 45, 46, 47] have suggested that if the large scale separation is determined by the sharp geometry rather than by viscosity and upstream conditions, then LES simulations can be run at lower Reynolds numbers. In a study by Rodi et al. [48], a cube mounted on one wall of a channel flow was run at Reynolds numbers of 3000 and 40000 (based on the cube height). It emerged that the flows were qualitatively very similar to each other and produced very similar length scales. Similar work was later carried out on the same flow by Krajnović and Davidson [46]
[47] at a Reynolds number of 40000. Accurate results, which matched up closely to
the experiments, were achieved at a feasible cost of 60 CPU hours.

Similar success was achieved at higher Reynolds numbers in a study of the flow
around an Ahmed body [44] with a 25% slant, where the authors ran the LES cal-
culations at a Reynolds number of $2 \times 10^5$, four times smaller than the experimental
Reynolds number [49]. Velocity profiles and turbulent kinetic energy on the slanted
rear surface and in the wake compared very well with the experiments. This does not
necessarily mean that reducing the Reynolds number will always give the same level
of success.

In a related study [45], the authors investigate the vortical flow features and com-
pare the instantaneous and mean flow. They conclude that some vortex structures on
the surface identify regions affected by large pressure forces, time averaged stream-
lines give a good indication of where water and dirt settles over a long period of
time, however certain structures such as those in the wake, have no relation with the
instantaneous vortex structures in the same region. What appears as a strong horse-
shoe vortex in the mean flow, actually consists of numerous irregular structures that
change over time, which makes the relevance of the time-averaged flow insignificant
in such regions.

Some of the structures resolved in a similar study [9] are shown in Figure 1.15,
which depicts the vortices on the slanting back by means of the Q criterion in the
instantaneous flow. Several unsteady vortex structures can be observed which steady-
state RANS cannot predict. The LES was able to capture three pairs of streamwise
vortices separating off the sides of the slant and separation was also observed on the
edge just before the slant, with vortices parallel to the edge. The reduced Reynolds
number made it possible to resolve the near-wall energy-carrying eddies. Although
the eddies were of different size to those for higher Reynolds numbers when compared
1.3. BACKGROUND

Figure 1.15: The isosurface of instantaneous second invariant of the velocity gradient $Q=3000$ on the rear slant of the body. Flow from left to right [9].

to experiments [50], such studies proved that the flow reproduced by the LES at the lower Reynolds numbers was qualitatively similar.

In another study by Krajnović and Davidson [10], the authors investigated the flow around a ground vehicle body represented by a simple bluff body with a rounded nose at a Reynolds number of $2.1 \times 10^5$. The time-averaged velocity profiles in the wake shown in figure 1.16 agreed reasonably well to the experiments, whereas the integrated base pressure ($C_{P_{base}}$) was overpredicted by 24% but follows the same trend as the experiments. The pumping frequency in the wake was also in good agreement with the experiments.

Similar levels of accuracy were achieved in an LES study [11] of a bus shaped body at the same Reynolds number. Small structures that had not been observed in the experiments [51, 52] due to their proximity to the wall and that were too unsteady to be captured by RANS were captured for the first time in this LES study. The instantaneous flow showed evidence of periodic structures such as those that form on the rear edges, and structures that appear randomly in time and space such as those in the far wake. Figure 1.17 shows the separation and hairpin vortices around the side and top of the bus, which may be more sensitive to Reynolds number than the
1.3. BACKGROUND

Figure 1.16: Time-averaged velocity profiles at three downstream locations at mid-plane. LES (solid line; experiment (symbols) [10]

flow downstream. The low frequency change in pressure on the rear surface requires long time averaging which comes at the cost of computational time. The authors suggest that the qualitative analysis from this study is transferable to higher Reynolds number flows.

As observed in Fig. 1.18a, higher base pressure from LES calculations was also predicted in a validation study on the ASMO (Aeroynamishes Studien Modell) [53], which predicted the base pressure exceptionally well. As reported in a study on turbulence models [54], the $k-\varepsilon$ model was unable to predict accurately the base pressure,
1.3. BACKGROUND

Figure 1.17: The isosurface of instantaneous second invariant of the velocity gradient $Q=1100$ [11]

which may be due to better prediction of the boundary layer on the surface of the car by the LES solution. The pressure distribution along the top and bottom surfaces of the ASMO vehicle (figure 1.18b) also show the LES to be in closer agreement with the experiments. Investigation of vorticity in the wake of the car showed differences between structures from the RANS solution, time-averaged LES and instantaneous LES. The instantaneous flow field showed strong positive and negative vortical structures which interact with each other resulting in complicated turbulence motion. In a more recent study by Kitoh et al., the authors performed RANS and LES calculations on the flow around a vehicle with a semi-complex underbody.

Several LES studies have been carried out on the Earth Simulator [13], the worlds fastest supercomputer between 2002 and 2004. Kitoh et al. [12] studied a semi-complex underbody, comparing velocities and pressure distribution for LES and RANS at a Reynolds number of $2.85 \times 10^6$. The ASMO study was used as validation for the LES method, so no experimental data is available for this model. Similar to the observations on the ASMO body, Fig. 1.18 shows that the LES calculations gave higher values of base pressure distribution than the RANS calculations, leading to lower drag predictions. The velocity distribution in the wake shown in Fig. 1.19b shows different trends for the two methods. Experimental investigation is required in
1.3. BACKGROUND

(a) Base pressure distribution  (b) Pressure distribution on the upper body and underbody, at the centre plane

Figure 1.18: Comparison of pressure distribution for RANS, LES and Volvo experiments on the ASMO body \[12\]

order to understand the reasons behind this.

Large Eddy Simulation on a complete Formula Car was performed by Tsubokura et al. \[13\] on the Earth Simulator. A grid of nearly 120 million cells was required due to the very small aerodynamic parts. Very fine grid resolution is needed in these areas for accurate flow prediction. Figures 1.20a and 1.20b are evidence of the unsteady characteristics of the flow, especially in the wake of the car. Time dependent flow structures are also present around the front tyres, which interact with the rest of the car. These images outline the importance of time dependent solutions. Lift coefficient ($C_L$) was within 1\% of the wind tunnel data, while drag coefficient ($C_D$) was 10\% higher than the experiments. RANS is normally used in the automotive industry due to the short time-scales they work in. In most cases this is sufficient for predicting forces, however, this only gives averaged flow information and unsteady simulation is required to gain a better understanding of the detailed structures in the flow for flow management and control.
1.3. BACKGROUND

Figure 1.19: Comparisons of RANS and LES solutions in the wake of semi-complex underbody diffuser at \(X/L = 1.05, Y = 0\) without moving ground and wheels moving) \[12\]

In a review by Moin [55], the author discusses the immersed boundary method investigated by Verzicco et al. [56] on a square back vehicle. A grid was generated around the baseline geometry, and drag reduction devices were added to the model without having to re-generate the grid. LES was carried out at Reynolds numbers of 20,000 and 100,000, with results being in relatively good agreement with the experiments [57], especially at the higher Reynolds number. The immersed boundary method proved to be effective for small geometrical changes, however, further tests should be carried out at more representative Reynolds numbers before any conclusions can be made.

Large Eddy Simulation has become a very powerful tool for aerodynamic assessment of the flow around vehicles, especially with increased computational power. Several studies have used it to measure unsteady and transient aerodynamic forces acting on the vehicle in transient pitching and yawing angle change [58, 59]. This has been typically carried out using the moving boundary and sliding grid techniques. Although these studies have been carried out at relatively low Reynolds numbers (\(Re = 230,000\),
1.4 Objectives and Thesis Content

After reviewing work by previous authors including experimental, RANS and LES, it is apparent that LES would be a useful tool for further investigation into the flow field of the underbody diffuser and trapped vortices in ground effect. The aim of this study is to investigate the limitations of RANS for the cases which are being studied and to explore the benefits or otherwise of LES. As the literature suggests, LES is suitable for highly turbulent flows where separation is defined by the geometry rather than being highly Reynolds number dependent, as the former requires less computational effort. The following cases will be considered.
1.4. OBJECTIVES AND THESIS CONTENT

**Underbody Diffuser**

LES will be carried out on a diffuser-equipped bluff body based on an experimental study [15] carried out at Loughborough University since results are readily available and will be used for validation and comparison. Calculations on a 13 degree diffuser at 28mm ride height will be carried out as very reliable results are available from the experiments for this particular set-up. The aim of this case is to give insight into the unsteady flow structures and assess the feasibility of LES on large domains.

**Delta Wing**

As a preliminary step for the trapped vortex case, LES on a 70° swept delta wing at an angle-of-attack of 27° based on experiments by Morton [20] will be investigated. The delta wing has been extensively researched and produces a similar flow field to that of the vortex generator. A comprehensive understanding of the vortices generated by the delta wing is necessary since literature on the vortex generator is limited.

**Trapped Vortices in Ground Effect**

The success of the delta wing calculations will lead to the study on the decay of a trapped vortex in ground effect. The focus will be solely on the flat underbody region of the vehicle with a vortex generator (fin) attached to it. The VG will be positioned at an angle of 20° to the oncoming flow, at which vortex breakdown should not occur on the surface. Detailed analysis of the mean and instantaneous flow will be presented to gain a deeper understanding of such devices. The culmination of the study will be the modelling of a rough ground representing a tarmac surface so that the interaction of the vortex with the ground can be investigated and vortex decay for the smooth
1.4. OBJECTIVES AND THESIS CONTENT

and rough boundary conditions can be compared. This will be done by using a laser
scanner to obtain the CAD of a piece of tarmac and using it as a boundary in the
LES.
Chapter 2

Methodology
2.1 Introduction

The numerical discretisation of the governing equations as applied in the Hydra code will be described in this chapter. The averaged equations for steady-state flow (RANS) as well as the filtered equations for unsteady motion (LES) will be presented. Although this work is based primarily on Large Eddy Simulation, the Reynolds Averaged Navier-Stokes method was also used to identify its limitations in the application of this study and appreciate the benefits of LES in flows which are dominated by highly unsteady turbulent flow structures.

Hydra [60] is a Rolls Royce code capable of solving the Euler, RANS, LES and DES equations for fluid flow. It has been validated on various flow problems and has proven to perform well even with complicated geometries [61]. Since Hydra is a predeveloped algorithm, it was not necessary to perform any testing or validation and could be readily applied to this study. It is an unstructured algorithm which formulates a solution methodology for compressible flow equations. The discretisation uses a finite volume method and the solution procedure is based on a density-based method. Steady state convergence can be accelerated by using a multigrid method [62]. The fact that it uses unstructured grids makes it easily applicable to various types of complex geometries. The unstructured algorithm uses a cell vertex scheme.

The chapter begins with the governing equations and their form for the RANS and LES methods. This is followed by the spatial discretisation and the domain decomposition strategy in Hydra. The control volume definition for the cell-vertex scheme is explained and the discretisation of inviscid and viscous fluxes is provided. Details of the sub-grid scale model used in Hydra as well as limiting factors in the near-wall region are explained. The grid generators, ICEM CFD and CENTAUR, which were used for this study are presented and the advantages of each one for the different cases is discussed. Finally, parallel processing on the HPC and efficient use
2.2 Governing Equations

2.2.1 Conservation Laws

The governing equations of fluid flow are derived from the following conservation laws: 1) Conservation of Mass, 2) Conservation of Momentum and 3) Conservation of Energy. They are applied to a fluid continuum and collectively referred to as the Navier-Stokes equations. They are presented below in tensor notation in differential form.

The Continuity equation is based on the principle that mass cannot be created nor destroyed, and is represented by the following equation:

\[
\frac{\partial \rho}{\partial t} + \sum_{i} \left( \frac{\partial (\rho v_i)}{\partial x_i} \right) = 0 \tag{2.1}
\]

where \( \rho \) is the density, the variable being conserved.

The Conservation of Momentum equation is defined as:

\[
\frac{\partial}{\partial t} (\rho v_i) + \sum_{j} \left( \frac{\partial (\rho v_j v_i)}{\partial x_j} \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2.2}
\]

where \( p \) is the pressure and \( \tau_{ij} \) is the viscous stress tensor defined as:

\[
\tau_{ij} = 2\mu S_{ij} + \lambda \frac{\partial v_k}{\partial x_k} \delta_{ij} = 2\mu S_{ij} - \left( \frac{2\mu}{3} \right) \frac{\partial v_k}{\partial x_k} \delta_{ij} \tag{2.3}
\]
where $\mu$ is the dynamic molecular viscosity, $\lambda = -\frac{2}{3}\mu$ is based on Stokes’ hypothesis and $S_{ij}$ and the strain rate tensor is given by:

$$S_{ij} = \frac{1}{2} \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right)$$ (2.4)

The Conservation of Energy equation is defined as:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j}(\rho v_j H) = \frac{\partial}{\partial x_j}(v_i \tau_{ij}) + \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right)$$ (2.5)

where the total energy per unit mass is represented by $E = e + \frac{1}{2}v_i v_i$, the total enthalpy per unit mass is represented by $H = h + \frac{1}{2}v_i v_i$, $k$ is the thermal conductivity coefficient and $e = c_v T$.

The fluid is usually assumed to behave like a calorically perfect gas as this provides a simple closure to the Navier-Stokes. The ideal gas equation is defined as:

$$p = \rho RT = (\gamma - 1) \rho \left( E - \frac{1}{2}(u^2 + v^2 + w^2) \right),$$ (2.6)

where $T$ is the temperature, $R$ is the specific gas constant and is defined by $R = c_p - c_v$, $c_v$ and $c_p$ are the specific heat at constant volume and pressure respectively and are related by $\gamma = \frac{c_p}{c_v}$.

### 2.2.2 Reynolds Averaged Navier-Stokes (RANS)

When dealing with highly turbulent flows, Direct Numerical Simulation (DNS) of the Navier-Stokes equations is inherently expensive and infeasible with current computational resources for flows with Reynolds numbers typically found in engineering.
2.2. GOVERNING EQUATIONS

The Navier-Stokes equations can be approximated using Reynolds averaging which involves the decomposition of flow variables into a mean and fluctuating part and can then be solved for the mean values. \textit{Hydra} uses a mass-weighted, time-averaging on all flow variables except for density and pressure to which conventional time-averaging is applied. The mass-weighted averaging known as Favre averaging is used on compressible flows to avoid correlations of density fluctuations which introduces additional complexity. Favre averaging defines a density weighted average of the flow variables in the following manner:

\[
\tilde{\phi}(x) = \frac{1}{\rho} \Delta t \lim_{\Delta t \to \infty} \frac{1}{\Delta t} \int_{t_0}^{t_0 + \Delta t} \rho(x,t)\phi(x,t)dt, \Delta t \gg t_{\text{turb}} \tag{2.7}
\]

Hence a Favre averaged variable can be expressed as:

\[
\tilde{\phi} = \frac{\bar{\rho}\bar{\phi}}{\bar{\rho}} \tag{2.8}
\]

The instantaneous variable \(\phi\) can be decomposed into a Favre-averaged quantity \(\tilde{\phi}\) and a redefined fluctuating component \(\phi''\) to give:

\[
\phi(x,t) = \tilde{\phi}(x) + \phi''(x,t) \tag{2.9}
\]

As a result, the compressible Reynolds-Favre averaged Navier-Stokes equations can be written as:
2.2. GOVERNING EQUATIONS

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \tilde{v}_i) = 0, \quad (2.10)
\]

\[
\frac{\partial}{\partial t} (\rho \tilde{v}_i) + \frac{\partial}{\partial x_j} (\rho \tilde{v}_j \tilde{v}_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \tilde{\tau}_{ij} - \rho \tilde{v}_i'' \tilde{v}_j'' \right), \quad (2.11)
\]

\[
\frac{\partial}{\partial t} (\rho \tilde{E}) + \frac{\partial}{\partial x_j} (\rho \tilde{v}_j \tilde{H}) = \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} - \rho \tilde{v}_j'' \tilde{h}'' + \tilde{\tau}_{ij} \tilde{v}_i'' - \rho \tilde{v}_i'' K \right) + \frac{\partial}{\partial x_j} \left[ \tilde{u}_i \left( \tilde{\tau}_{ij} - \rho \tilde{v}_i'' \tilde{v}_j'' \right) \right] \quad (2.12)
\]

where the overline denotes a Reynolds averaged variable and tilde represents a Favre-averaged variable.

These equations include an additional term which represents the transfer of momentum due to the turbulent fluctuations. This is referred to as the Reynolds-stress tensor and defined by:

\[
\tau_{ij}^F = -\rho \tilde{v}_i'' \tilde{v}_j'' \quad (2.13)
\]

The laminar viscous stresses are evaluated by:

\[
\tilde{\tau}_{ij} = 2\mu \tilde{S}_{ij} = \mu \left( \frac{\partial \tilde{v}_i}{\partial x_j} + \frac{\partial \tilde{v}_j}{\partial x_i} \right) \quad (2.14)
\]

The Reynolds stress tensor results in six additional relations [28] which need to be modelled to close the RANS equations.
2.2. GOVERNING EQUATIONS

2.2.2.1 Eddy-Viscosity Hypothesis

The Boussinesq hypothesis [63, 64] assumes that the turbulent shear stress is related linearly to mean rate of strain as in a laminar flow, with eddy viscosity being the proportionality factor as defined by the equation below:

\[
\tau_{ij}^F = -\rho \bar{\nu}''_i \bar{v}''_j = 2\mu_T \tilde{S}_{ij} - \left( \frac{2\mu_T}{3} \right) \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \rho \tilde{k} \delta_{ij}\]  

(2.15)

where \( \tilde{S}_{ij} \) and \( \tilde{k} \) are the Favre-averaged strain rate and turbulent kinetic energy respectively.

If \( \mu_T \) is determined, this can be added to the laminar viscosity and the RANS equations can be closed to model turbulent flow.

2.2.2.2 Turbulence Models

Time averaging the equations of motion results in additional terms which require modelling to represent the effect of the turbulent field on the mean flow. The Reynolds stress tensor arising from the RANS equations introduces six independent unknowns which means that suitable closure approximations in the form of turbulence models are required.

First-order closures offer the the simplest method to approximate the Reynolds stresses in the Reynolds/Favré-averaged Navier-Stokes equations. Their role is to determine the eddy viscosity \( \mu_T \). The Spalart-Allmaras one-equation model was easily applied to all the cases in this study, whilst the \( k-\varepsilon \) and \( k-\omega \) SST were only applied to the delta wing case.

The Spalart Allmaras [65] model is represented by a scalar convection-diffusion
2.2. GOVERNING EQUATIONS

equation with source terms:

\[
\frac{\partial \tilde{\nu}}{\partial t} + u \frac{\partial \tilde{\nu}}{\partial x} + w \frac{\partial \tilde{\nu}}{\partial z} = \frac{1}{\sigma} \left( \nabla \cdot [(\nu + \tilde{\nu}) \nabla \tilde{\nu}] + c_{b2}(\nabla \tilde{\nu})^2 \right) + S
\]  

(2.16)

where \( \nu \) is the molecular kinetic viscosity and \( \tilde{\nu} \) is the turbulent working variable.

The source term is expressed as:

\[
S = c_{b1} \tilde{S} - \left( c_{w1} f_w - \frac{c_{b1}}{k_2} f_{t2} \right) \left( \frac{\tilde{\nu}}{d} \right)^2 + f_{t1} \Delta u^2,
\]  

(2.17)

which may be divided into production, destruction and trip terms:

\[
S \equiv P(\tilde{\nu}) - D(\tilde{\nu}) + T
\]  

(2.18)

where

\[
P(\tilde{\nu}) = c_{b1} \tilde{S} \tilde{\nu},
\]  

(2.19)

\[
D(\tilde{\nu}) = \left( c_{w1} f_w - \frac{c_{b1}}{k_2} f_{t2} \right) \left( \frac{\tilde{\nu}}{d} \right)^2,
\]  

(2.20)

\[
T = f_{t1} \Delta u^2.
\]  

(2.21)

The trip term provides a mechanism for triggering transition to turbulence at a specified location on the geometry. Modelling of transition was not studied in this project and the trip term was omitted, meaning the flow was turbulent throughout the entire domain.

The turbulent eddy viscosity is defined by

\[
\mu_t = \bar{\rho} \tilde{\nu} f_{v1},
\]  

(2.22)
2.2. GOVERNING EQUATIONS

\( f_{v1}, f_w, f_{t1} \) are derived from auxiliary relations and \( c_{b1}, c_{b2} \) and \( c_w \) are constants [66].

The \( k - \varepsilon \) turbulence model is a two-equation model which solves transport equations for turbulent kinetic energy \( k \) and the turbulent dissipation rate \( \varepsilon \). Important developments were made by Jones and Launder [67, 68], Launder and Sharma [69] and Launder and Spalding [70] in the 1970’s and it is the most commonly used two-equation eddy viscosity model in engineering applications. It requires damping functions in the viscous sublayer to ensure reasonable values of \( k \) and \( \varepsilon \) in the near wall region. It is known to perform quite well for boundary type flows but not as well for flows with adverse pressure gradients. The turbulent viscosity is expressed as:

\[
\mu_t = \rho C_{\mu} \frac{\tilde{k}^2}{\tilde{\varepsilon}}
\]  \hspace{1cm} (2.23)

The widely used \( k - \omega \) Shear Stress Transport (SST) turbulence model [71], combines the \( k - \omega \) model of Wilcox [72] and [73] and the high Reynolds number \( k - \varepsilon \) model. The \( k - \omega \) model is employed in the sublayer and logarithmic part of the boundary layer as it does not require damping functions and performs better in adverse pressure gradients and compressible flows. The \( k - \varepsilon \) model is used in wakes and shear layers providing a compromise between accuracy and numerical stability. The \( k - \omega \) SST model has a modified turbulent eddy-viscosity function that accounts of the transport of turbulent shear stress to improve the accuracy of strong pressure gradients.

The energy equation is given similar treatment to the models discussed above in order to close the RANS equations, however, since the cases considered here are incompressible (Mach < 0.3), the modelling of this equation is unimportant and will not be described in more detail.
2.2 GOVERNING EQUATIONS

2.2.3 Large Eddy Simulation (LES)

*Hydra* has the capability of running Large Eddy Simulation (LES). LES provides a three-dimensional, time-dependent solution of the Navier-Stokes governing equations, hence capturing the unsteady behaviour of most engineering flows. The idea is to model the small isotropic structures that are believed to have universal behaviour and resolve the large, energy-carrying structures. *Hydra* uses an implicit approach which utilises the local grid size as a spatial filter. Turbulent structures that are smaller than the characteristic filter width are filtered from the solution and modelled by a sub-grid scale model, whilst the larger structures are fully resolved by the filtered Navier-Stokes equations. This means that LES requires much higher grid refinement than RANS but is still computationally cheaper than DNS due to its filtering approach.

Although some LES methods make use of wall functions in their calculations, this was not the case in this study. A ‘wall-resolving’ approach was adopted by generating a good near wall mesh and modifying the standard Smagorinsky subgrid scale (SGS) model with a Baldwin-Lomax correction to improve its near wall behaviour as will be discussed in this chapter.

In LES mode *Hydra* uses the second-order spatial discretisation scheme of Moinier [66] and the explicit 3 stage third-order accurate Runge-Kutta time stepping scheme. The standard Smagorinsky SGS model is used to model the small scale structures.

2.2.3.1 LES Filtered Equations

Spatial filtering is required to separate the large scales from the small scales. Any flow variable $\phi$ is decomposed into a filtered, large-scale, resolved part $\overline{\phi}(x_i, t)$, and a
2.2. GOVERNING EQUATIONS

sub filter, small-scale, unresolved part \( \phi'(x_i, t) \):

\[
\phi(x_i, t) = \overline{\phi}(x_i, t) + \phi'(x_i, t) \tag{2.24}
\]

The difference between time averaging in equation 2.9 and spatial filtering in equation 2.24 should be noted. Here the variables remain a function of both space and time. The filtered variable at location \( \vec{r}_0 \) is defined as:

\[
\overline{\phi}(\vec{r}_0, t) = \int_D \phi(\vec{r}, t) G(\vec{r}_0, \vec{r}, \Delta) d\vec{r} \tag{2.25}
\]

where \( D \) is the entire domain, \( G \) represents the filter function and \( \vec{r} \) is the position vector.

When LES is applied to compressible flows, Favre-Averaging is employed together with spatial filtering to the governing equations in order to avoid products between density and other variables. The LES Favre-filtered equations as defined by [28] read:

\[
\frac{\partial \overline{\rho}}{\partial t} + \frac{\partial}{\partial x_j} (\overline{\rho} \overline{\nu}_j v_i) = 0, \tag{2.26}
\]

\[
\frac{\partial}{\partial t} (\overline{\rho} \overline{v}_i) + \frac{\partial}{\partial x_j} (\overline{\rho} \overline{\nu}_j \nu_i) + \frac{\partial \overline{\rho}}{\partial x_i} \frac{\partial \hat{\sigma}_{ij}}{\partial x_j} = -\frac{\partial \tau_{ij}^{SF}}{\partial x_j} + \frac{\partial}{\partial x_j} (\overline{\sigma}_{ij} - \hat{\sigma}_{ij}) \tag{2.27}
\]

\[
\frac{\partial}{\partial t} (\overline{\rho} \overline{e}) + \frac{\partial}{\partial x_j} (\overline{\rho} \overline{v}_j \overline{e}) + \frac{\partial \hat{q}}{\partial x_j} + P \hat{S}_{kk} - \hat{\sigma}_{ij} \hat{S}_{ij} = -A - B - C - D \tag{2.28}
\]

where the overline represents an unweighted filter, tilde represents a density-weighted filter, \( e \) denotes the internal energy per unit mass, \( \hat{S}_{ij} \) is the Favré-filtered strain-rate tensor, and \( \tau_{ij}^{SF} = \overline{\rho}(v_i v_j - \overline{v}_i \overline{v}_j) \) represents the Favre-averaged subgrid scale stress. The right hand side of equation 2.27 \( \tau_{ij} \) is approximated but \( (\overline{\sigma}_{ij} - \hat{\sigma}_{ij}) \)
2.3 Sub Grid Scale Model

A sub grid scale (SGS) model is required to model the subgrid scale stresses. It is responsible for modelling the energy transfer between large and subgrid scales. *Hydra* uses a standard Smagorinsky SGS model [74], an eddy viscosity model. It is the earliest and most commonly used model, where the Favre-averaged subgrid scale tensor as given by Blazek [28] is:

\[
\tau_{ij}^{SF} = -2\mu_{sgs} \tilde{S}_{ij} + \left( \frac{2\mu_{sgs}}{3} \right) \frac{\partial \tilde{v}_k}{\partial x_k} \delta_{ij} \tag{2.29}
\]

where the components of the strain rate tensor are given by:

\[
\tilde{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_j} \right) \tag{2.30}
\]

The subgrid scale viscosity \( \mu_{sgs} \) is defined as:

\[
\mu_{sgs} = \rho_{smag}^2 \sqrt{2\tilde{S}_{ij} \tilde{S}_{ij}}, \tag{2.31}
\]

where the Smagorinsky length scale is,

\[
l_{smag} = C_s \Delta, \tag{2.32}
\]

and \( S_{ij} \) is the strain rate. The filter width \( \Delta \) is the cube root of the control volume and \( C_s \) is a model constant. Approaching the wall the sub grid scale viscosity should tend to zero. In typical grids, the spacing is reduced normal to the wall in order
2.3. SUB GRID SCALE MODEL

to resolve predominant gradients in the normal direction. The high aspect ratios of the grid near the wall results in unrealistically large length scales when based upon volume and consequently excessive sub grid scale viscosity. Here a limit is imposed on length scale based upon traditional RANS mixing length models.

The Baldwin-Lomax [75] or Cebeci-Smith [76] mixing length RANS model computes turbulent viscosity in the inner region from,

\[(\mu_t)_i = \rho l_{bl}^2 |\omega| \]

where the mixing length is,

\[l_{bl} = \kappa y [1 - \exp(y^+) \]] \quad (2.34)\]

The Smagorinsky subgrid scale model stress should always be smaller than the Reynolds stress from the mixing length model, so the length scale in the Smagorinsky model is restricted by

\[l_{smag} = \min(C_S \Delta, \kappa y) \]

This restriction is similar to a Detached Eddy Simulation (DES) approach, but uses a RANS type model only in the inner region of the boundary layer, where the Smagorinsky model would otherwise create a spuriously high value of subgrid scale viscosity.

The theoretical value calculated by [77] is \(C_S \approx 0.18\). However, this value is not universal and usually depends on the type of flow being simulated and on Reynolds’ number. The Smagorinsky constant must be reduced to \(C_S \approx 0.065\) for channel flows [78], and is usually \(\approx 0.12 - 0.15\) for free shear flows. In this study, the Smagorinsky constant was fixed to \(C_S = 0.1\) for all cases, a value which is used for a wide range of engineering flows. By keeping this value low, one minimises the affect of the model and allows more structures to be resolved rather than modelled. If different types of
flow are present in the same domain the standard Smagorinsky model would not be able to model accurately all the scales and $C_S$ would need to be determined for each type of flow.

### 2.4 Spatial Discretisation

A finite volume approach is employed by *Hydra* which subdivides the solution domain into a number of non-overlapping control volumes. The governing equations are then discretised on each control volume. The discretisation scheme described here is limited to second order accuracy and follows the work of Moinier [66], which is based on the MUSCL approach [79].

The numerical flux is integrated over the faces of the median dual control volume [80], which is generated from the original mixed element mesh. 2D grids will be considered for simplicity, with 3D grids following directly. The median-dual control volume is constructed around each node of the grid by joining the centroids of the neighbouring cells with the midpoint of each of the edges as shown in Fig. 2.1. By doing this, each node of the primal grid is associated with a cell of the dual grid. This allows the primal grid nodes to be used as storage locations of the median dual control volumes.

*Hydra* uses a functional representation of $Q$, the vector of conserved variables, within each control volume to solve a Riemann problem at the interface of the control volumes.
2.4. SPATIAL DISCRETISATION

Figure 2.1: Construction of Median Dual Control Volume around internal node.

2.4.1 Inviscid Flux $F_I$

In Moinier’s work [66] the flux is integrated over the faces of the median-dual control volumes according to the flux-differencing ideas of Roe [81], which combines central differencing with a smoothing flux based on one-dimensional characteristic variables. For an edge $ij$ connecting nodes $i$ and $j$ as depicted in Fig. 2.1, the flux $F_{ij}$ on the face at the mid-point of the edge $ij$ is defined by the second order scheme of Moinier [66] as:

$$F_{ij} = \frac{1}{2}[F(Q_i) + F(Q_j) - d_{ij}] \quad (2.36)$$

where the smoothing term $d_{ij}$ is defined as:

$$d_{ij} = \frac{1}{2}(1 - \kappa)|A_{ij}|(L^{lp}_j(Q) - L^{lp}_i(Q)) \quad (2.37)$$

where $\kappa \in [0, 1]$, $L^{lp}$ is the linearly-preserving pseudo Laplacian operator [66], and $|A_{ij}| = \partial F/\partial Q$.

Within Hydra, the constant $\frac{1}{2}(1 - \kappa)$ is replaced with a user-defined constant $\varepsilon$ so that the smoothing term is scaled solely by $\varepsilon$ in the following manner:
2.4. SPATIAL DISCRETISATION

\[ d_{ij} = |A_{ij}| \varepsilon (L_j^{bp}(Q) - L_i^{bp}(Q)) \] (2.38)

\( L^{bp} \) has been found to be accurate and robust [82, 83, 84] especially on grids which are not highly stretched.

2.4.2 Viscous Flux \( F^V \)

The viscous flux is approximated half-way along each edge and integration is performed around each volume. This requires an approximation of \( \nabla Q \) at this point which can be done by approximating the flow variables at the nodes using the existing edge-weights. This is defined by:

\[ \nabla Q_{ij} = \frac{1}{2} (\nabla Q_i + \nabla Q_j) \] (2.39)

Since the above equation is the average of two central differences, it will not damp high frequency modes and the inviscid flux numerical dissipation is insufficient in the boundary layer where the viscous terms are dominant. This can be resolved by using a simple difference along the edge as in [66] to give:

\[ \nabla Q_{ij} = \nabla Q_{ij} + \left( \frac{(Q_i - Q_j)}{|x_i - x_j|} - \nabla Q_{ij} \cdot \delta s_{ij} \right) \delta s_{ij} \] (2.40)

where

\[ \delta s_{ij} = \frac{x_i - x_j}{|x_i - x_j|} \] (2.41)
2.5. **INTEGRATION SCHEME**

This damps the high frequency error modes in the boundary layer.

### 2.5 Integration Scheme

Temporal discretisation in *Hydra* uses a standard third-order accurate, three stage Runge Kutta algorithm. A detailed description of the Runge Kutta method can be found in [85, 86, 87], but in general it provides high order of accuracy for non-linear equations. This can be expressed as:

\[
Q_j^{(0)} = Q_j^n \tag{2.42}
\]

\[
Q_j^{(k)} = Q_j^n - \alpha_k \Delta t_j R_j^{(k-1)} \quad k=1,2,3 \tag{2.43}
\]

\[
Q_j^{n+1} = Q_j^3 \tag{2.44}
\]

with

\[
R_j^{(k-1)} = C_j(Q_j^{(k-1)}) - B_j^{(k-1)} \tag{2.45}
\]

\[
B_j^{(k-1)} = \beta_k D_j(Q_j^{(k-1)}) - (1 - \beta_k) B_j^{(k-2)} \tag{2.46}
\]

where \(C_j(Q_j^{(k-1)})\) is the convective contribution to \(R_j\) arising from the inviscid terms and \(D_j(Q_j^{(k-1)})\) are the remaining parts due to the viscous fluxes, both physical and numerical, and source term if present. The coefficients are:

The viscous flux needs to be evaluated at each stage since \(\beta \neq 0\), making the process expensive.
Running incompressible flows on a compressible solver results in time-steps which are smaller than those required to resolve the length scales present. This means that stability was more constraining than the accuracy for time-step selection for the cases in this study.

When running steady state simulations, *Hydra* has low Mach number preconditioning based on eigenvalue scaling of the coefficient matrix in order to overcome the problem of degradation in convergence at low flow speeds. However, this was not available when running the code in true time-dependent mode as is necessary for LES.

### 2.6 Boundary Conditions

To solve the system of equations discussed in this chapter, the correct physical boundary conditions must be applied. The residual must be defined for all nodes, including those at the boundary. There are two ways of imposing boundary conditions at a boundary: the first involves directly applying the desired values to the flow variables, and the second involves modifying the fluxes at the boundaries to achieve the desired conditions. A combination of both is used by *Hydra*.

In single-grid methods, the boundary conditions are usually imposed on the update vector or the solution after the update has taken place. However, in *Hydra* the residual with the boundary conditions included is defined since the multigrid scheme for steady-state flows transfers residuals between grids. The boundary conditions implemented in this study were freestream condition, subsonic inlets, subsonic outlets, slip walls, and no-slip walls both stationary and moving. Following Moinier [66], the

\[
\begin{align*}
\alpha_1 &= \frac{1}{3} \quad \alpha_2 = \frac{1}{2} \quad \alpha_3 = 1 \\
\beta_1 &= 1 \quad \beta_2 = 1 \quad \beta_3 = 1
\end{align*}
\]

Running incompressible flows on a compressible solver results in time-steps which are smaller than those required to resolve the length scales present. This means that stability was more constraining than the accuracy for time-step selection for the cases in this study.

When running steady state simulations, *Hydra* has low Mach number preconditioning based on eigenvalue scaling of the coefficient matrix in order to overcome the problem of degradation in convergence at low flow speeds. However, this was not available when running the code in true time-dependent mode as is necessary for LES.

### 2.6 Boundary Conditions

To solve the system of equations discussed in this chapter, the correct physical boundary conditions must be applied. The residual must be defined for all nodes, including those at the boundary. There are two ways of imposing boundary conditions at a boundary: the first involves directly applying the desired values to the flow variables, and the second involves modifying the fluxes at the boundaries to achieve the desired conditions. A combination of both is used by *Hydra*.

In single-grid methods, the boundary conditions are usually imposed on the update vector or the solution after the update has taken place. However, in *Hydra* the residual with the boundary conditions included is defined since the multigrid scheme for steady-state flows transfers residuals between grids. The boundary conditions implemented in this study were freestream condition, subsonic inlets, subsonic outlets, slip walls, and no-slip walls both stationary and moving. Following Moinier [66], the

\[
\begin{align*}
\alpha_1 &= \frac{1}{3} \quad \alpha_2 = \frac{1}{2} \quad \alpha_3 = 1 \\
\beta_1 &= 1 \quad \beta_2 = 1 \quad \beta_3 = 1
\end{align*}
\]

Running incompressible flows on a compressible solver results in time-steps which are smaller than those required to resolve the length scales present. This means that stability was more constraining than the accuracy for time-step selection for the cases in this study.

When running steady state simulations, *Hydra* has low Mach number preconditioning based on eigenvalue scaling of the coefficient matrix in order to overcome the problem of degradation in convergence at low flow speeds. However, this was not available when running the code in true time-dependent mode as is necessary for LES.

### 2.6 Boundary Conditions

To solve the system of equations discussed in this chapter, the correct physical boundary conditions must be applied. The residual must be defined for all nodes, including those at the boundary. There are two ways of imposing boundary conditions at a boundary: the first involves directly applying the desired values to the flow variables, and the second involves modifying the fluxes at the boundaries to achieve the desired conditions. A combination of both is used by *Hydra*.

In single-grid methods, the boundary conditions are usually imposed on the update vector or the solution after the update has taken place. However, in *Hydra* the residual with the boundary conditions included is defined since the multigrid scheme for steady-state flows transfers residuals between grids. The boundary conditions implemented in this study were freestream condition, subsonic inlets, subsonic outlets, slip walls, and no-slip walls both stationary and moving. Following Moinier [66], the

\[
\begin{align*}
\alpha_1 &= \frac{1}{3} \quad \alpha_2 = \frac{1}{2} \quad \alpha_3 = 1 \\
\beta_1 &= 1 \quad \beta_2 = 1 \quad \beta_3 = 1
\end{align*}
\]
2.6. BOUNDARY CONDITIONS

Boundary conditions are described as follows:

**Freestream** - This requires specifying a free-stream density, an inlet to outlet static pressure ratio, a Mach number, and direction of velocity inlet vector. The freestream state vector is defined by \( Q_\infty \). The boundary condition is then implemented through the inviscid boundary flux term \( F_k \) which is evaluated by solving the Riemann problem given by:

\[
F_k = \frac{1}{2} \left( F_k^I(Q_k) + F_k^I(Q_\infty) - |A_k|(Q_k - Q_\infty) \right)
\]  

(2.47)

where \( F_k \) is the numerical flux associated with the boundary face \( k \) with no viscous contribution.

**Subsonic Inlet** - Inlets are specified through a total pressure and temperature condition that is used to modify the freestream condition at the particular boundary, as well as pitch and yaw angles which were 0° for the calculations in this study. The total pressure is calculated from the desired velocity at the boundary. Turbulence properties are also imposed at the inlet for RANS, but ignored for LES. The implementation is similar to that of the freestream boundary, where a state specified through the conditions specified here is used to replace \( Q_\infty \) in the Riemann problem above. The characteristic boundary treatment used in *Hydra* means that the resulting velocity at the inlet might differ slightly from the desired value.

**Subsonic Outlet** - This is defined through one state variable which is static pressure. The implementation is similar to that of inlet conditions.

**slip** - This condition is implemented on inviscid walls where boundary layer effects are ignored. The mass flux given by \( F_k \) at the boundary is set to zero and the normal momentum components of the residual are removed at all nodes which lie on the boundary by enforcing a vanishing normal velocity component, i.e. \((v.n)_{wall} = 0\). This ensures that the normal velocity components which are initialised to zero do not
2.7. GRID GENERATION

change.

\textit{no-slip} - This condition is implemented on viscous walls where a boundary layer is expected. All normal components of momentum of the residual are set to zero, i.e. \( v_{\text{wall}} = 0 \). In the case of the moving ground in this study, a value for the velocity component in the direction in the flow is specified in the same direction as the flow.

2.7 Grid Generation

The quality of the grids generated is crucial for the accuracy of the solution, especially in LES where the solution is very grid dependent. It also has a computational cost implication which means that there is a trade off between accuracy and cost. Although \textit{Hydra} is an unstructured solver, in practice it has been found that hexahedral elements are preferred as tetrahedral grids result in increased numerical damping of the eddies. The face normals of each element are aligned with the primary axes of the Cartesian coordinate system leading to better grid quality. However, hexahedral grids can be difficult to generate around complicated or sharp geometries and may in fact result in worse quality due to element skewness.

In this study, ICEM CFD Version 11 was used to generate hexahedral and hybrid (both hexahedral and tetrahedral elements) grids, whilst \textit{CENTAUR} was used to generate tetrahedral grids. A description and their application for the current studies are explained below.

2.7.1 ICEM CFD

\textit{ICEM CFD} is capable of generating body fitted grids through multi-block hexahedral grid generation, as well as unstructured hexahedral, tetrahedral and hybrid grids. In
hexa mode it creates an initial block around the whole domain which can then be further split into the desired number of blocks as depicted in Fig. 2.2a, depending on the complexity of the geometry. The blocks around the body are made to fit the shape of the geometry and the edges of the blocks are then associated with the curves of the geometry. Further divisions can be made in the blocking to control the expansion of element size into the domain (eg. to capture the wake of a flow). Nodes are set along each edge of the blocks with the desired distribution. ICEM is capable of creating O-grids which were used around the nose of the bluff body and around the vortex generator. Details of the grid will be discussed in further detail in the relevant chapters. O-grids are good at capturing curved surfaces and also allows greater control of the grid in areas of high gradients. The final grid can be exported as an unstructured Fluent format file (.msh) to be read into the Hydra pre-processor JM 52.
2.7. GRID GENERATION

2.7.2 CENTAUR

CENTAUR is an unstructured grid generator, capable of generating both hexahedral and tetrahedral grids. In the current study, it was used to generate a grid around the delta wing, as it was more appropriate than hexahedral grids to capture the sharp edges. Hexahedral grids for this case resulted in highly skewed cells in critical regions of the flow. CENTAUR allows the user to specify grid clustering in regions of interest by means of geometric or CAD sources. Tetrahedral cells can be generated on the surface as shown in Fig. 2.3 with a prism layer source to control the thickness of the prism cells around the wall. In this case the CAD itself is used as a source from which the prism layer grows.

A geometric source can be specified through a range of different shapes which is used to cluster elements within that geometry. For this study it was beneficial to have a frustum shaped source in the position of the delta wing leading edge vortex so that a high level of refinement could be achieved. Further details of meshing this geometry will be discussed in Chapter 4, but a side view image of the frustum shaped clustering is presented in Fig. 2.4.
2.8 Parallel Processing

Running on a single processor would be infeasible for challenging flow problems such as that of the current study. The grid refinement required to resolve the small eddies results in a very large number of grid points and the range of time scales between the large and small eddies requires the calculation to be run for a large number of time steps. The grid is partitioned at run time using ParMetis and the solver uses the OPlus and MPI library for parallel communication.

Tests were carried out for the different LES calculations to find the ideal number of processor cores to run on in order to manage the computational resources available efficiently. Figure 2.5a shows a comparison of four different cases that were run on different numbers of cores to assess this. The cases used for the test were the delta wing; 8 million and 15 million node grids, and the diffuser; 10 million and 20 million node grids. Each case was set to run for 500 time-steps on 8, 16, 24 and 32 computer nodes, each consisting of 12 processor cores. This results in 96, 192, 288 and 384 processor cores.

Ideally the speed-up is directly proportional to the number of cores that the calculation is running on, so that if the number of cores is doubled, the time for the

Figure 2.4: Frustum shaped source showing grid refinement.
2.8. PARALLEL PROCESSING

Figure 2.5: Plots measuring the benefit of parallel processing on the Hydra cluster.

Simulation is halved. However, Fig. 2.5b, which uses the 20 million node diffuser calculation as an example, shows that the speed-up for the calculation deviates from the ideal curve as the number of cores increases. Efficiency is lost due to the large amount of data that has to be communicated across the cores.

The calculations in this study were carried out over 288 processor cores since the benefit of running on a higher number of cores starts to decrease significantly beyond this point.
2.9 Closure

The present chapter has described the unstructured scheme employed by Hydra for which the governing equations were discretised. It has been shown that the numerical accuracy adopted by the algorithm is second order accurate. The Large Eddy Simulation method and sub-grid-scale Smagorinsky model were described. As mentioned above, the standard SGS model has its limitations in the near-wall region, consequently, care is required when generating the grid in this region and this must be kept in mind when analysing the results.

A brief description of the mesh generators used in this study was also given and the benefits of each explained. The unstructured nature of Hydra made it easily applicable to the cases in this study which covered a range of different grids that the different geometries could support.
Chapter 3

Automotive Underbody Diffuser
3.1 Introduction

Large Eddy Simulation (LES) is carried out for the flow around a bluff body equipped with an underbody rear diffuser in close proximity to the ground, representing an automotive diffuser. The scope of the time-dependent simulations is not necessarily to improve on the accuracy of Reynolds-Averaged Navier-Stokes (RANS), but to give further insight into vortex formation and progression at experimental Reynolds numbers ($1.01 \times 10^6$ based on model height and inlet velocity). This will lead to better understanding and control of the flow. Vortical flow structures in the diffuser region, along the sides and top surface of the bluff body will be modelled and investigated. Differences between instantaneous and time-averaged flow structures will be presented and explained. Comparisons to pressure measurements and Particle Image Velocimetry (PIV) data from wind tunnel experiments on an identical bluff body model will also be presented.

3.2 Experimental Configuration

The computational study of the automotive diffuser was based on experiments by Jowsey and Passmore [15] carried out in the Loughborough University $1.9m \times 1.3m$ open-circuit closed-working-section wind tunnel. The wind tunnel was equipped with a six-component underfloor balance for force and moment measurements, while pressure measurements were taken using a 64-channel high-speed pressure scanner. The repeatability of the measurements after a complete dismantle, removal, and re-installation of the model is $\pm 0.009$ for the drag coefficient $C_D$ and $\pm 0.035$ for the lift coefficient $C_L$. The repeatability of the difference between two configurations after the re-installation is $\pm 0.003$ for $C_D$ and $\pm 0.020$ for $C_L$. Tests were carried out at a wind speed of $40ms^{-1}$ at a range of diffuser angles and ride heights.
3.2. EXPERIMENTAL CONFIGURATION

The Loughborough University wind tunnel does not have a moving ground plane and so all tests were carried out over a fixed ground plane. Based on previous work on moving and stationary grounds [31, 6, 88, 89] the authors concluded that a fixed ground simulation was sufficient for studying trends of pressure distribution and forces and understanding mechanisms, but if magnitudes are required then a moving ground is essential.

A bluff body with a 25% rear diffuser was used to represent a car with an underbody diffuser. The dimensions of the model were a length of 0.8m, a width of 0.4m, a height of 0.31m, a nose radius of 0.064m and an end-plate width of 0.012m as shown in Fig. 3.1, giving a 5% blockage ratio. These dimensions gave a relatively large base area ensuring reasonable independence of the underbody and overbody flows. The diffuser plate was hinged and connected to the main model 0.04m upstream of the diffuser inlet as shown in Fig. 3.2a to avoid separation at the inlet due to surface discontinuity. The hinge was covered in a flexible plastic skin. Splitter plates were used on the diffuser plate to create the channels for the multi-channel diffuser set-up. The model was attached to the underfloor balance by means of a threaded bar as shown in Fig. 3.2b.

Figure 3.1 shows the arrangement of the pressure tappings along the plane-channel centre-line and the centre-line of each diffuser channel for the multi-channel set-up, allowing pressure measurements to be taken at these positions even when the model was in plane-channel configuration. The diffuser plate also had rows of pressure tappings placed at several streamwise intervals to create pressure maps (Fig. 3.1d).
3.3 Computational Set-Up

3.3.1 Geometry

The bluff body diffuser model used for the computational study was modelled using the same dimensions as those used in the experiments [15]. The dimensions are non-dimensionalised by the model height $H$, giving a total length of $L = 2.58H$, body width of $W = 1.29H$, nose radius of $r = 0.21H$, diffuser length of $N = 0.64H$ (25% length diffuser) and end plate width of $w = 0.04H$, which are labelled in the bluff body cross-section in Fig. 3.3. The diffuser angle chosen for the computations was $\alpha = 13^\circ$, measured from the horizontal as shown in Fig. 3.3. This angle generated the highest level of downforce out of all the angles tested in experiments but did not carry the largest drag penalty, giving it an efficient lift to drag ratio (Fig. 1.7). Three
3.3. COMPUTATIONAL SET-UP

(a) Hinge arrangement

(b) Model attachment

Figure 3.2: 3D wind tunnel model [15].

dimensional views of the bluff body are depicted in Fig. 3.4.

The domain, which is depicted in Fig. 3.5, was $18H$ long, $5H$ wide and $3.5H$ high. These dimensions are comparable to the size of the test work section in the wind tunnel and proved to be sufficient for the LES calculation. The inlet boundary was placed $5.2H$ upstream of the bluff body and the outlet was placed $12.8H$ downstream, measured from the front surface. The bluff body was modelled at a non-dimensional ride height of $h = 0.09H$ (Fig. 3.5a) and is placed laterally in the centre of the domain (Fig. 3.5b). At the chosen ride height, the experiments[15] showed that the flow was largely attached and provided reliable data for comparison. The $x$, $y$ and
3.3. COMPUTATIONAL SET-UP

Figure 3.3: Cross-section of the diffuser-equipped bluff body at the symmetry plane showing size parameters.

$z$ axis run in the streamwise, vertical and spanwise directions respectively, with $u$, $v$ and $w$ representing the velocity components in each direction.
3.3. COMPUTATIONAL SET-UP

Figure 3.4: 3D views of diffuser.
3.3. COMPUTATIONAL SET-UP

Figure 3.5: Domain with diffuser-equipped bluff body.
3.3. COMPUTATIONAL SET-UP

3.3.2 Grid Generation

Two grids, consisting of 10 million and 20 million nodes, were generated in ANSYS ICEM CFD. Finer grids would have required too much computational time and would have been infeasible to run on the resources at the time due to the resulting small time step which will be discussed later. The blocking tool within ANSYS ICEM CFD was used to create an unstructured hexahedral grid.

Figure 3.6a shows the 3-dimensional blocking of the fine grid consisting of 69 blocks in total, where the turquoise lines represented the edges of the blocks and all other colours represent the geometry. An O-type block was created around the nose (Fig. 3.6b) to capture the geometry of the model and enable more control over the expansion of cells normal to the surface. The blocking in the diffuser region is presented in Fig. 3.6d.

A block structured grid proved to be very challenging in the diffuser region, as one of the edges of a rectangular block had to be collapsed to form a wedge-shaped block in order to follow the geometry of the diffuser section. Figure 3.7a shows a 2-dimensional image of the blocking in the diffuser region for the 10 million node grid, showing the three edges of the block. This results in a layer of triangular cells at the inlet of the diffuser (Fig. 3.7b) since all cells in this region join at a merged edge. This issue was resolved for the 20 million node grid in which the wedge shaped block is replaced by three diamond shaped blocks as observed in Fig. 3.8a, which results in solely hexahedral cells (Fig. 3.8b).

Images of the final 20 million node grid are presented in Fig. 3.9. Although Hydra is an unstructured solver, hexahedral elements are preferred as tetrahedral grids have previously resulted in increased numerical damping of the eddies. Grid points are clustered in regions of high gradients, namely around the radiused nose of the body.
3.3. COMPUTATIONAL SET-UP

Table 3.1: Grid parameters for hexahedral grids of the diffuser-equipped bluff body.

<table>
<thead>
<tr>
<th>Nodes $\times 10^6$</th>
<th>$x$</th>
<th>$y$</th>
<th>$z$</th>
<th>$x^+$</th>
<th>$y^+$</th>
<th>$z^+$</th>
<th>Nodes along edge</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.3 - 12</td>
<td>0.3 - 14</td>
<td>15 - 600</td>
<td>3 - 15</td>
<td>60 - 650</td>
<td>240</td>
<td>60</td>
</tr>
<tr>
<td>20</td>
<td>0.1 - 5</td>
<td>0.1 - 6</td>
<td>5 - 250</td>
<td>1 - 5</td>
<td>20 - 280</td>
<td>280</td>
<td>118</td>
</tr>
</tbody>
</table>

and at the diffuser inlet. A slice of the grid at the symmetry plane is shown in Fig. 3.9b, with a close up of the front end shown in 3.9c. A rapid increase in cell width downstream of the rounded edge can be observed, which could not be avoided in order to keep the overall cell count feasible.

Table 3.1 gives details about the number of cells and their size along the surface of the bluff body in the $x$, $y$ and $z$ directions for both grids. The 20 million node grid had an increased number of nodes in all directions. The first cell height normal to the wall gives a $y^+$ which varies between 1 on entry to the underbody region and diffuser inlet and 5 at the end of the flat underbody (immediately upstream of the diffuser inlet). The streamwise spacing varies between $x^+ = 5$ at the entrance to the underbody region and $x^+ = 250$ mid-length of the flat underbody, while the spanwise spacing varies between $z^+ = 20$ and $z^+ = 280$. Although the streamwise and spanwise requirements are less than that of the normal spacing, they are still larger by a factor of 5 and 17 respectively when compared to a well resolved LES study of a cascade by McMullan and Page [90]. This resulted in high aspect ratio cells ($\approx 100$) in the near-wall region of the bluff body as can be seen in Fig. 3.9c.
3.3. COMPUTATIONAL SET-UP

Figure 3.6: Diffuser-equipped bluff body geometry showing blocking.
3.3. COMPUTATIONAL SET-UP

(a) wedge-shaped blocking

(b) wedge-shaped mesh

Figure 3.7: 2D wedge shaped blocking in diffuser section

(a) diamond shaped blocking

(b) diamond-shaped mesh

Figure 3.8: 2D diamond shaped blocking in diffuser section
3.3. COMPUTATIONAL SET-UP

(a) 3D front view

(b) Symmetry plane \((z/W = 0.5)\)

(c) Near wall grid spacing

Figure 3.9: Block-structured 20 million node hexahedral grid.
3.3. COMPUTATIONAL SET-UP

3.3.3 Boundary Conditions

A subsonic velocity inflow was applied at the inlet boundary and specified through a total pressure and temperature condition. Since this is a relatively low speed case, using a compressible solver like Hydra results in relatively small time-steps and longer running times than actually required. In order to decrease the computational time, the calculation was run at twice the speed of that in the experiments, halving the geometry in order to maintain the experimental Reynolds number of $1.01 \times 10^6$. This increases the Mach number from 0.12 to 0.24, and since the latter can still be considered incompressible, this does not introduce an error in the calculation.

A total pressure of $1.05 \times 10^5 Pa$ was specified at the inlet based on a velocity of $80 ms^{-1}$ (twice that of the experiments) and air density of $1.226 kg m^{-3}$, whilst temperature was set to $293 K$. A freestream static pressure was specified at outlet. The bluff body diffuser and the ground plane beneath it were modelled as viscous, stationary walls (since the experiments[15] made use of a stationary ground rather than a moving ground) and the sides and ceiling of the domain were set to inviscid walls. Since the aim of this study is not to measure absolute values, a stationary ground is sufficient as explained in section 3.2.

3.3.4 Calculation

The time step for the coarse mesh was $3.6 \times 10^{-8}$ seconds, while that for the fine mesh was $5.9 \times 10^{-8}$ seconds. The hexahedral cells at the inlet of the diffuser allowed for a larger time-step on the finer grid, whereas the highly skewed cells at the diffuser inlet of the coarser grid lead to an even smaller time step. The initial LES flow field was obtained from a reasonably converged RANS solution in order to reduce computational time. A Smagorinsky subgrid scale constant of $C_S = 0.1$ was used in
3.4. RESULTS

The calculation ran over 288 processor cores on 24 nodes each with 24GB memory and connected by an Infiniband network. Since the LES was initiated from a RANS calculation, the finest grid was run for 10 ‘flow passes’ before averaging was initiated, and a reliable mean flow was achieved over a further 15 ‘flow passes’, where ‘flow pass’ is defined as the time taken for a particle travelling at the freestream velocity to traverse the length of the bluff body. The quality of the mean was judged by means of surface pressure plots at different cross-sections of the body. The calculation ran for around 400 elapsed hours representing 0.1416 seconds in physical time.

3.4 Results

An assessment of the LES is carried out through analysis of the sub-grid scale ratio which compares the resolved and modelled viscosity. Results are initially shown for the 10 million 20 million node grids, however, the chapter continues with results mostly from the latter grid since it showed improved results. Comparisons between the LES from this study and the experiments of Jowsey and Passmore [15] are drawn for velocity and pressure in the underbody region. Both instantaneous and time-averaged ‘streamlines’ in the diffuser region are presented and explained. The wake flow is compared to a previous study by Krajnović and Davidson [11] and resolved flow structures around the model are presented through iso-surfaces of pressure.

3.4.1 Subgrid Scale Viscosity Ratio

The influence of the subgrid scale (SGS) model is presented in Fig. 3.10 for the finest grid (20 million nodes) through the mean SGS viscosity ratio ($\mu_{SGS}/\mu$), where $\mu$ is the
3.4. RESULTS

molecular viscosity and $\mu_{SGS}$ is the subgrid scale viscosity. By combining equations 2.31 and 2.32 one can observe that the modelled viscosity varies by the square of the filter width:

$$\mu_{sgs} = \rho (C_s \Delta)^2 \sqrt{2S_{ij}S_{ij}}, \quad (3.1)$$

where,

$$\Delta = (vol)^{1/3} \quad (3.2)$$

Taking equation 3.2 into consideration, it is important for the cell to have small dimensions in all directions in order to keep the modelled stress low. At the wall the modelled stress should go to zero (as turbulence is suppressed by the presence of the wall) and all stress should be resolved. In turbulent regions this is expected to increase due to the presence of high gradients.

In Fig. 3.10, the SGS ratio in the freestream, away from the body, is approximately 0 since the flow is not turbulent in this region. High values (above 20) are observed near the ground and underbody surfaces. These regions coincide with areas of high aspect ratios as observed in Fig. 3.9c, up to 50 in the centre of the underbody region and up to 100 at the wall. Although the normal distance away from the wall is reasonably fine, the streamwise and spanwise dimensions could be refined to avoid unrealistic large length scales.

Figure 3.10a shows SGS ratios below 20 throughout most of the diffuser region, indicating reasonable grid size in this region. However, Figure 3.10b shows high ratios in the largely separated regions over the top surface and behind the bluff body, indicating that further grid refinement would lead to a better resolved LES.
3.4. RESULTS

3.4.2 Pressure Distribution

Figure 3.11 shows a comparison of pressure distribution, for the experiments and the two grids, along the centre line \((z/W = 0.5)\) and quarter line \((z/W = 0.25)\) of the underbody and top surface. In Fig. 3.11a and Fig. 3.11b stagnation is observed at the front of the body \((C_p = 1)\) followed by a pressure drop as the flow accelerates around the radiused nose. Pressure is recovered until a second suction peak is observed at the diffuser inlet which is caused by the discontinuity in surface gradient. This recovers along the surface of the diffuser. The 20 million node grid shows significant
3.4. RESULTS

improvement over the coarser grid, in particular achieving the correct suction around the nose and along the flat underbody.

In the experiments, a small laminar separation bubble is reported at the start of the flat underbody ($x/L = 0.1$) [15] in the region of adverse pressure gradient. This is possibly indicated by the plateau region in the measured pressure coefficient shown in Fig. 3.11a and Fig. 3.11b. In the LES, a separation bubble is not observed on the underside of the bluff body, although separation is evident on the upper surface (Fig. 3.14) which results in transition to turbulence.

A calculation was run on the finer grid with all turbulence models switched off, so effectively modelling a laminar flow. Separation was observed in the same position recorded in the experiments which is indicated by the blue contours in Fig. 3.12b. For the LES calculations, transition to turbulence on the lower surface occurs further downstream in the diffuser region, whilst separation over the front top corner is captured. This is likely due to the fact that the flow over the top edge is a separated shear layer, whereas the bottom wall (ground plane) is producing a damping effect which hinders the formation of the small separation bubble on the bottom edge. Whilst this likely difference in behaviour between experiment and simulation will have an impact on the overall flow features this is not apparent in the underbody pressure coefficients at the quarter plane where suction at the diffuser inlet is in good agreement.

The late transition to turbulence has been the subject of research for many years [91] and could be due to several factors: insufficient grid resolution around the front of the body; poor inflow turbulence disturbances; inadequacies in the standard Smagorinsky SGS model [92]. The first factor is probably the most important as can be seen by the large change in pressure coefficient achieved when improving longitudinal and spanwise resolution when moving to the finer grid. Estimates of turbulent length scales from a RANS $k - \varepsilon$ prediction [93] were found to be as small
3.4. RESULTS

Figure 3.11: Time-averaged $C_p$ distribution along the underbody and top surface of bluff body.

as 0.0015m in the underbody region, requiring the cell size in the $x$ and $z$ directions to be at least halved in order to capture 80% of the turbulence energy. Aspect ratios of the grid in this region were quite large which is not desirable for LES calculations. Some calculations were carried out using a numerical ‘trip’ similar to that used in the predictions of a jet nozzle flow [94] and analogous to that employed in experimental work. However, the disturbances did not sustain into full turbulence and there was no improvement in the solution.

Pressure recovery along the first half of the flat underbody matches up very well with experiments but degrades in the second half, especially at the centre-line. This is due to the suction at the diffuser inlet not being as strong as that in the experiments. The strength of the suction at this point will determine pressure recovery on either
3.4. RESULTS

Figure 3.12: Laminar test showing separation immediately downstream of front nose.

side of it, meaning that pressure recovery is not modelled accurately if the suction peak is not captured well. The LES shows laminar separation at the entrance to the diffuser, at which point the flow is expected to be already turbulent, followed by reattachment of the flow and pressure recovery along the rest of the diffuser.

Over the top surface of the body (Fig. 3.11c and Fig. 3.11d), flow accelerates over the radiused edge giving rise to the pressure peak in the plot. The flow separates as indicated by the plateau region in the LES plots as well as was observed earlier by the velocity contours in Fig. 3.14. The flow then reattaches as a turbulent flow and pressure is recovered to a point where it remains constant along the length of the
3.4. RESULTS

top surface. Although the suction peak is not modelled accurately, the separation is captured by both grids with the 20 million node mesh in good agreement along the flat surface.

The suction over the top radiused edge is stronger than that over the bottom edge. This may be due to the fact that there is more flow deflected over the top edge as the stagnation point on the front surface is at 0.39$H$ from the bottom surface of the model. The pressure distribution over the top surface is in much better agreement with the experiments than the bottom surface. This may be due to the fact that over the top surface the flow is dominated by larger scale flow features which are more easily captured by the grid.

3.4.3 Flow Structure

Figure 3.13 shows the path taken by the flow as it travels around the bluff body based upon the mean velocity field. The body is inverted in order to easily view the flow in the underbody region. The streamlines approaching the body are deflected around the radiused edges. Twisting of the streamlines along the lower longitudinal edges indicate the presence of a rotating structure along the side of the bluff body. As it reaches the end-plates of the diffuser, it winds up around them to form a pair of counter-rotating vortices. As the flow exits the diffuser, large rotating structures are observed in the wake of the model. The pressure contours in Fig. 3.13 show areas of low pressure regions concentrated around the radiused front edges and diffuser inlet.

Figure 3.14a shows instantaneous velocity contours at the symmetry plane. A stagnation point forms at the front of the body as the oncoming flow (left to right) hits the nose of the bluff body. Acceleration is observed around the top and bottom edges, with the flow over the top separating and transitioning to turbulence, indicated
3.4. RESULTS

Figure 3.13: Bluff body in inverted position showing time-averaged $C_P$ on the surface and path taken by the 3-dimensional streamlines.

by the unsteady velocity contours. Figure 3.14b shows a close up of the separation and onset to turbulence over the top surface, while Fig. 3.14c shows similar behaviour in the diffuser region. The flow on the underside remains attached and unsteadiness only appears around half way along the diffuser region.

There are signs of flow reversal near the diffuser inlet which was not observed in the experiments[15]. This laminar separation is being picked up due to the late transition to turbulence in the underbody region. Although the location of transition is not specified in the experiments, the flow is expected to have transitioned further upstream, however, the LES calculations show transition to turbulence on the ramp of the diffuser as the separated flow reattaches to the surface. Recirculating flow is observed behind the bluff body (Fig. 3.14a), which is being fed by the flow exiting the diffuser region.

3.4.4 Diffuser Region.

Figure 3.15 presents a series of mean streamwise velocity contours with streamlines constrained to the slice. Streamlines are not necessarily parallel to the wall due to
3.4. RESULTS

discarding the out of plane velocity component. At the diffuser inlet, $x/L = 0.75$ (Fig. 3.15a), flow enters the underbody region from the side and rolls up into a small vortex at the edge of the underbody. A little further downstream at $x/L = 0.813$ (Fig. 3.15b) the vortex splits around the inner edge of the end-plate to form two vortices. At $x/L = 0.875$ (Fig. 3.15c) the vortex on the inside of the end-plate has grown larger and continues to do so as flow progresses downstream through the diffuser. The rotating structure occupies a relatively large region at the diffuser exit $x/L = 1$ (Fig. 3.15e). Smaller vortices are observed under the sharp edge of the end plate.

Three instantaneous images at the position $x/L = 0.875$ are presented in Fig. 3.16. When comparing them to the time-averaged image at the same position (Fig. 3.15b), the vortex core changes position and direction in each image, while the time-averaged image shows a steady vortex structure at the edge of the diffuser. The presence of the vortices keeps the diffuser from stalling by stopping separation along the ramp as low-energy flow near the surface, which would otherwise lead to separation, is re-distributed away from the surface and high energy flow is introduced.
3.4. RESULTS

![Contour plots](image)

(a) $z/W = 0.5$

(b) top corner

(c) diffuser region

Figure 3.14: Contours of non-dimensional, instantaneous, streamwise velocity $u/U$ at symmetry plane ($z/W = 0.5$).

86
3.4. RESULTS

Figure 3.15: Time-averaged streamlines constrained to $y-z$ plane at different axial positions along the diffuser. View from front of the lower left edge of the body.
3.4. RESULTS

Figure 3.16: Instantaneous ‘streamlines’ constrained to $y-z$ plane at $x/L = 0.813$. Time difference between three images is $tU/H = 1.5$. View from front of the lower left edge of the bluff body.
3.4. RESULTS

3.4.5 Diffuser Exit Flow

Figure 3.17 shows a comparison of the vertical component of velocity $v$, the spanwise component $w$, velocity magnitude $V_{yz}$ based on these two components and axial vorticity $\Omega_x$ between the LES calculations and PIV data from the experiments at the diffuser exit plane ($x/L = 1.0$). Only vertical and spanwise velocity components were available from the experiments hence streamwise velocity could not be compared. The contours from the experiments are clipped in the near wall region where reliable data could not be recorded.

Figure 3.17b and Fig. 3.17c show that velocity magnitude compares reasonably well with the largest difference being the high velocity captured by the LES under the end-plate. Contours of the individual components of the velocity magnitude are analysed to understand the discrepancy.

Figure 3.17d and Fig. 3.17e show a good comparison of the vertical component of velocity with the red contours indicating an upward direction and the blue contour near the end plate indicating a downward direction. A discrepancy arises in the spanwise component of velocity (Fig. 3.17f and Fig. 3.17g) with a region of higher velocity under the end-plate captured by the LES. The red contour shows velocity from left to right, indicating that the vortex on the left hand side of the diffuser is rotating in an anti-clockwise manner when viewed from the back. The fact that the experiments do not capture the high velocity underneath the end-plate is suspicious since the vortices in the diffuser form due to flow being ‘sucked’ into the underbody region due to the pressure difference, accelerating around the end-plate and rolling up into a vortex. Since the laser sheet in the PIV method is very thin and the particles being tracked will travel through it very quickly due to high axial velocities which are not being captured, it is possible that the particles are not being picked up at all by the cameras and hence the PIV data misses the high spanwise velocity component.
3.4. RESULTS

under the end-plate.

The high acceleration around the end-plate leads to the vortices in the LES to be more tightly wound to the inner sides of the end plates when compared to the experiments. This leads to a larger area of flow travelling in the streamwise direction (out of the paper) in between the vortex pair. In the experiments, the streamwise vortices occupy a larger volume in diffuser region forcing the oncoming flow through the gap between the vortex pair. This leads to acceleration along the centre of the underbody, leading to low pressures. The fact that the LES shows a larger unaffected area may be the reason why the LES shows less suction in the pressure plots in Fig. 3.11a than that observed in the experiments. The fluid passing in between the vortex pair is not as constrained as it is in the experiments, resulting in lower acceleration and suction levels.

Streamwise vorticity ($\Omega_x$) compares relatively well as similar vortical structures are observed for the LES and PIV in Fig. 3.17h and Fig. 3.17i respectively. The largest discrepancy arises from the higher spanwise velocity underneath the end-plate as discussed above.
3.4. RESULTS

(a) Rear view of bluff body showing section of diffuser selected for images below

(b) LES - $V_{yz}$

(c) PIV - $V_{yz}$

(d) LES - $v$

(e) PIV - $v$

(f) LES - $w$

(g) PIV - $w$

(h) LES - $\Omega_x$

(i) PIV - $\Omega_x$

Figure 3.17: Time-averaged contours of velocity magnitude $V_{yz}$, vertical component $v$, spanwise component $w$ and streamwise vorticity $\Omega_x$ for LES (left) and PIV (right) at the diffuser exit plane.
3.4. RESULTS

(a) Time-averaged; Inverted model showing underbody

(b) Instantaneous; Inverted model showing underbody

(c) Time-averaged; View of rear surface

(d) Instantaneous; View of rear surface

(e) Time-averaged; View of front surface

(f) Instantaneous; View of front surface

Figure 3.18: Iso-surface of pressure coefficient $C_p = -0.35$ and $C_p = -1$ coloured by non-dimensional freestream velocity comparing time averaged (left) and instantaneous (right) structures.
3.4 RESULTS

3.4.6 Iso-surfaces of Pressure.

Figure 3.18 shows a comparison of iso-surfaces of pressure coefficient coloured by axial velocity around different parts of the bluff body for the time-averaged and instantaneous flow. In Fig. 3.18a and Fig. 3.18b the bluff body is inverted with the underbody facing upwards, as indicated in the image, to allow a better view of the flow in the underbody region. A pair of vortices, analogous to the rotating structure observed in Fig. 3.15 is seen exiting the back of the diffuser region in the mean flow. The instantaneous image shows a complicated flow with numerous resolved structures in the diffuser region. Some small flow structures can be observed around the sides of the bluff body but are not present on the flat region of the underbody. This is an indication that laminar to turbulent transition in the underbody region does not happen until the flow reaches the diffuser region.

Figure 3.18c and Fig. 3.18d show a view of the back end of the bluff body with the diffuser in the correct orientation. The mean shows a vortex ring of very slow moving flow in the near wake with two distinct structures exiting the diffuser region. It is noticeable that the vortex ring is asymmetrical as more samples would need to be collected for a completely symmetrical mean flow. Although the flow around a symmetric body is not necessarily symmetric due to interactions of the flow from either side, in this case the asymmetry is visible behind the bluff body where velocities are low, indicating that longer time averaging is required. This is an issue in high Reynolds number LES, especially in the wake where the time scales are large, as a true mean would be very time consuming and costly to achieve. The instantaneous flow shows a high level of unsteadiness as opposed to a single, distinct vortex ring.

Figure 3.18e and Fig. 3.18f again emphasise the difference between the time-averaged and instantaneous flow. The time-averaged flow shows some signs of separation, however, it is the instantaneous flow (Fig. 3.18f) that shows how the separation
3.4. RESULTS

transitions to turbulence through the numerous resolved vortex structures around the top and side of the diffuser. Separation was captured on three (top and sides) of the four surfaces around the nose of the bluff body.

3.4.7 Recirculating Flow in the Wake.

Streamlines constrained to the $x - y$ and $z - y$ planes which cut through the ring vortex observed in Fig. 3.18c are plotted in the wake of the bluff body diffuser to give further insight into the base flow. Figure 3.19b and Fig. 3.19c shows streamlines at the centre and quarter planes respectively. Krajnović and Davidson[10] carried out similar LES calculations to those in this study but on a bus-shaped body, which consisted of a bluff body without a diffuser. The streamlines on the centre-plane in the wake for their flow (3.19a) show two counter-rotating vortices labelled $F_1$ and $F_2$, with the lower $F_2$ being much smaller than upper $F_1$, the upper vortex, due to the small ground clearance which limits the amount of fluid entering the lower vortex.

The presence of the rear diffuser alters the wake of the bluff body flow. The lower vortex is dominating the near wake because of the larger amount of flow being fed into it from the diffuser. Consequently, the stagnation point and saddle point indicated by $S_b$ and $D$ respectively in Fig. 3.19a form higher up in the positive $y$ direction as observed in Fig. 3.19b. The large vortex pushes the upper vortex, reducing it in size, nearly causing it to disappear as the flow runs off the top surface of the bluff body. If more samples were collected, it is thought that only one upper vortex would be present rather than the two small ones present in Fig. 3.19b. At the quarter plane ($z/W = 0.25$) in Fig. 3.19c, the upper vortex is larger in size than at the centre plane and the stagnation point is slightly lower. This is due to the slice being taken on the outer section of the ring vortex observed in Fig. 3.18c.
Figure 3.18d and Fig. 3.18e show streamlines constrained to planes on the vertical axis, passing through the vertical mid-point of the base and lower vortex respectively. As observed in Fig. 3.18c, longer time-averaging would be required to achieve a better mean in the base region as can be observed from the small vortices at the surface of the bluff body, typical of the instantaneous flow. These would disappear with longer time-averaging and the two small vortices at the top of Fig. 3.18d would coalesce into one forming the sides of the mean ring vortex. Figure 3.18d captures some of the rotating flow from the lower vortex moving in the negative $x$ direction towards the base of the bluff body. Since Fig. 3.18e shows a plane passing straight through the core of the lower vortex, all the streamlines are observed moving downstream. At this plane, the flow in the vortex ring is either moving upwards or downwards, and hence not captured by the constrained streamlines.
3.4. RESULTS

(a) Krajnović and Davidson[11] at $z/W = 0.5$

(b) $z/W = 0.5$

(c) $z/W = 0.25$
3.4. RESULTS

(d) \( y/H \) = (plane through vertical mid-point of base)

(e) \( y/H \) = (plane through lower part of ring vortex)

Figure 3.18: Time-averaged streamlines constrained to different planes plane.
3.5 Closure

LES of a bluff body in ground effect with a diffuser has been carried out at experimental Reynolds numbers. Previous studies of underbody diffuser flows have been experimental or Reynolds-Averaged CFD and this is the first time that unsteady structures in the diffuser region have been investigated. Time-averaged and instantaneous results are compared in the diffuser region where an unsteady, continuously moving and changing longitudinal vortex averages in the mean to be a circular steady vortex. Unsteady vortex structures that are shed into the wake of the flow average out to give a vortex ring behind the rear face of the bluff body. The small computational time-step inherent in LES means that longer time-averaging is required to achieve a more accurate mean in the wake of the flow where the time scales are large.

The main weakness in the simulation is the late laminar to turbulence transition in the underbody region. Grid refinement in the underbody region would have enabled the capture of small turbulent structures expected along the underbody, as was simulated around the sides and top of the bluff body. Lower aspect ratios, especially around the front nose and entry to the underbody would also allow more structures to be resolved.

It is thought that the effect of the ground hindered these structures from being resolved, possibly because the structures present in the tight underbody region are smaller than those resolved around the sides and back of the body. A finer grid might solve this but would lead to smaller time-steps and an increased number of cells which may lead to run times which are infeasible. Testing with a numerical trip similar to that used in experimental work did not succeed in forcing an early turbulent boundary simulation. The available experimental data underneath the bluff body is limited to mean surface pressures and non-intrusive measurements in the underbody flow region are needed to validate the simulation technique.
3.5. CLOSURE

The results from this study provide a good understanding of the unsteady flow in the underbody diffuser region of a bluff body. This can form the basis of further LES studies on bluff bodies equipped with a rear diffuser where improved modelling of the laminar to turbulence transition may lead to a better representation of the flow. Based on current resources available for such calculations, a study which concentrates on an isolated underbody section is more feasible as a finer grid can be generated for a significantly smaller region, enabling more structures to be resolved on the grid.
Chapter 4

Vortical Flow from a Delta Wing
4.1 Introduction

Prior to simulating a vortex generator in ground effect, it is necessary to validate the methodology for strong vortical flows. In aerospace engineering, the vortex from the sharp leading edge of a delta wing has been studied extensively [18, 20, 95, 19] due to its relevance to aircraft high angle of attack aerodynamics. This makes it an ideal case to validate the methodology and to evaluate the benefit of LES.

4.2 Background

An analogy can be made between the vortices created by an automotive vortex generator and the leading edge vortices of a delta wing [8]. Whilst a delta wing’s angle-of-attack determines the strength and breakdown position of the leading edge vortex, the same can be said for the sweep angle of a vortex generator. In both cases, higher angles lead to earlier breakdown. The delta wing has been studied extensively both experimentally and computationally and therefore provides a good case for validation of the LES methodology.

The delta wing is a slender, highly swept, low aspect ratio wing normally found on high performance aircraft such as military aircraft. It can produce large values of lift due to its leading-edge vortices [96] which form as a shear layer separates from the sharp leading edge. These vortices keep the flow attached on the suction side of the wing, enabling it to sustain high values of lift at very high angles of attack as opposed to the stalling behaviour found on the wings of civil aircraft. This results in a highly manoeuvrable, agile aircraft.

Déléry [97] explains that the sharp geometry imposes a separation line which is always fixed along the leading edge of the delta wing. A separated shear layer rolls up
to form a vortical structure over the wing as depicted in Fig. 4.1a, called the primary vortex. The vorticity which was previously contained in the turbulent boundary layer is transferred to the flow away from the wall and due to the rolling up of the separation surface, it concentrates in the core of the vortical structure. With the aid of the diagram in Fig. 4.1a, Breitsamter [16] explains that the vortex sheet reattaches further inboard of the wing and a secondary separation line is observed close to this, leading to a secondary vortex rotating in the opposite sense which reattaches close to the leading edge. However, one must also note that the adverse pressure gradient in the transverse direction on the surface of the delta wing may also be leading to the secondary separation forming the secondary vortex. Very low pressure is present underneath the leading edge vortices, which increases in the outboard direction, leading to the adverse pressure gradient. The vortex core contains high axial velocities [16], reaching values as high as three times the freestream velocity. As angle of attack (\(\alpha\)) increases, lift increases, until at a critical angle, a disorganisation of these vortical structures is observed, referred to as vortex breakdown [22], and the extra lift is lost.

Vortex breakdown results in a loss of lift coefficient, hence limiting the operation of the wing. Many attempts have been made at modelling these vortices in experiments such as those by Harvey [98] and Garg and Leibovich [99]. Many experiments were carried out at much lower Reynolds number than those typical of aerospace and automotive application and a detailed review on the subject can be found in Lucca-Negro and O’Doherty[100]. Although the mechanism behind vortex breakdown is not universally accepted, there are a few characteristics of vortex breakdown which have been recorded in several studies.

Vortex breakdown on delta wings was first observed in 1954 by Werlé [101] in his water-tunnel experiments. This was quickly confirmed by Peckham and Atkinson [102], and many experiments followed. Vortex breakdown is characterised by a rapid deceleration of the vortex in both axial and swirl directions due to an adverse
4.2. BACKGROUND

(a) Vortex formation; main flow features

Figure 4.1: Vortex formation and vortex breakdown [16]

Vortex bursting: Sudden expansion of vortex core flow

Jet-type core flow: \( \frac{u}{U_\infty} \)

Wake-type core flow: \( \frac{u}{U_\infty} \)

Pressure gradient at high angles of attack, leading to a stagnation point ahead of the breakdown, and a sudden expansion of the vortex core (Fig. 4.1b). The stagnation point marks the vortex breakdown location; it is unsteady and oscillates about a mean position [95]. The flow upstream of the breakdown has a jet-like velocity distribution, while downstream of the breakdown, the flow is wake-like [99][103] [17] [16]. Early experiments by Wentz and Kohlman [104] also show that higher leading edge sweep can sustain higher \( \alpha \) before breakdown occurs. The full span delta wing calculations of Le Roy et al. [105] outlined the interaction of the starboard and portside vortices, which resulted in different breakdown locations on each side.
4.2. BACKGROUND

In experiments on a 76\degree swept delta wing with a Reynolds number of $1.07 \times 10^6$ (based on wing mean aerodynamic chord), Breitsamter [16] identified a dominant frequency of the large scale unsteadiness which is associated with the high turbulence intensity in the vortex core on the onset of breakdown as seen in Fig. 4.2. The helical mode instability of the breakdown flow is characterised by narrow band amplitudes of the fluctuating pressure coefficient that increase with angle of attack, causing unsteady loading (buffeting) on the surface of the wing.

In experiments on 60\degree, 65\degree, 70\degree and 75\degree delta wings, Gursul [106] associates the helical mode instability to the pressure fluctuations observed on the delta wings. The vortex breakdown position moves upstream towards the apex as angle-of-attack increases. No dominant frequency was observed in the spectra of pressure fluctuations on the wing surface after breakdown moved forward to the apex of the wing. This further proves that the unsteady loading when breakdown has occurred is due to the helical mode instability rather than the vortex shedding which is present after the breakdown has reached the apex of the wing.

Gursul also found that for his range of Reynolds numbers (25,000 to 100,000), the dominant frequency was not Reynolds number dependent. This was also confirmed in experiments by Mitchell et al. [95] which were carried out at much higher Reynolds numbers ($9.7 \times 10^5$ to $2.6 \times 10^6$). The separation at the leading edge is dominated by the sharp edge leading-edge geometry as noted by Délery [97] rather than Reynolds number and angle of attack. Despite this, De Luca and Guglieri [96] confirm in their experiments that Reynolds number and angle of attack have an influence on the transition region of the vortex, where the boundary layer changes from laminar to turbulent. The important role of the leading edge geometry is also evident in a study by Cummings and Schutte [107], in which a rounded leading-edge was modelled. This geometry resulted in two primary vortices on each side of the wing as opposed to one primary and one secondary vortex for the sharp leading-edge geometry. A similar
flowfield was obtained on a blunt edge delta in a study by Luckring [108].

Jones et. al [109] performed a computational comparison of vortex flow above a delta wing at Reynolds number \(2.7 \times 10^5\) for angle-of-attack between 10° and 40° to that in an open pipe flow at an Reynolds number of 600. Although the characteristics of a low Reynolds number flow differ to those at higher Reynolds numbers, some comparisons can be made. It is evident from the computational tests that azimuthal vorticity is present for both cases just before the breakdown position. It was concluded that negative azimuthal vorticity is one of the criteria for vortex breakdown, created when axial vorticity is turned into azimuthal vorticity by turning and stretching. It was also confirmed that the critical helix angle criterion marks the transition
4.2. BACKGROUND

Figure 4.3: Total pressure coefficient contours for the delta wing $\alpha = 40^\circ$, $\text{Re}=0.9 \times 10^6$ [17]

from a supercritical flow upstream of the breakdown to subcritical downstream of the breakdown. The critical helix angle for the delta wing was $45^\circ$. No breakdown was observed on the delta wing for angles-of-attack which were less than $20^\circ$.

Hsu and Liu [17] performed a RANS CFD analysis on vortex breakdown, in which they were able to capture an example of the rapid changes in the structure of the vortex. Figure 4.3 shows pressure contours upstream and downstream of the breakdown. Upstream of the breakdown, there is a strong pressure gradient between the inner and outer core. After breakdown occurs, $C_p$ in the inner region is higher and there is less pressure loss across the core. It is the strong negative pressure in the centre of the core which is desirable for increased suction on the suction surface, resulting in more lift in the delta wing case or more downforce in the underbody vortex case. However, the unsteadiness of the flow could not be captured through the use of the RANS equations which can only model a time-averaged flow. A time-dependent simulation is needed to model the turbulent eddies in the flow. LES calculations were carried out by Mary [110], which emphasise the need for very fine grids in LES at the cost of computational time.
Figure 4.4: Evolution of breakdown location as a function of angle-of-attack ($U_\infty = 24m/s$) [18]

The experiments documented by Mitchell [18] and Mitchell et al. [111] on the ONERA 70° delta wing provide a good benchmark for validation of computational work. The authors provided laser sheet visualisation, surface oil flow visualisation, surface pressure data and LDV results to give more detail on the vortex breakdown phenomena and breakdown location. The breakdown locations obtained from the laser sheet visualisation are presented in figure 4.4, showing that vortex breakdown moves upstream over the delta wing as angle of attack increases, as previously reported by Werlé [101]. At higher values of $\alpha$, the suction on the surface of the wing was found to be stronger [112], whilst at angles of 20° or less breakdown did not occur on the surface of the delta wing [95]. LDV measurements performed by Mitchell [18] were able to estimate the breakdown location through the jet-like to wake-like velocity behaviour of the vortices. The authors also observed vortical substructures that are spatially stationary and form in the shear layer around the vortices.
4.3 EXPERIMENTAL CONFIGURATION

The CFD predictions were based on the ONERA experiments [18]. Figure 4.5 shows the geometry of the model used in the experiments. The delta wing had a sweep angle $\Lambda$ of 70° and a root chord $c$ of 0.95 m. The wing span $b$ was 0.73$c$ at its trailing edge, which was 0.02$c$ thick and was bevelled on the windward side at an angle of 15° to form sharp leading edges, whilst the trailing edge was blunt without any bevel.

The experiment was conducted inside the Onera F2 subsonic wind tunnel, which is a closed-return atmospheric wind tunnel with a rectangular test section of dimensions 1.4 m $\times$ 1.8 m $\times$ 5 m. Tests were run at freestream velocities between 10 ms$^{-1}$ and 75 ms$^{-1}$. The mean turbulence intensity inside the wind tunnel was 0.1%. The delta wing model was mounted on a sting attached to a horizontal support and placed in the centre of the test section as shown in Fig. 4.6a. The angle-of-attack could be altered and a range of different angles between 20° and 40° were tested in the experiment. The yaw angle was set to 0° with respect to the flow. A picture of the delta wing model inside the wind tunnel is shown in Fig. 4.6b.

The model used to collect steady surface pressure data had 232 pressure taps in total placed in rows at 16 chordwise stations normal to the root chord. Their
4.3. EXPERIMENTAL CONFIGURATION

(a) Wind tunnel configuration

(b) Delta wing model in test section

(c) Arrangement of pressure tappings

Figure 4.6: Delta wing model in the wind tunnel.

Arrangement is shown in Fig. 4.6c. Another model without pressure tappings was used to collect flowfield and oil flow data. The measurement techniques used were: Laser Sheet Visualisation; to measure the vortex breakdown position, Surface Oil Flow Visualisation; to study the interaction of the surface effects on the leeward surface and along the leading edges of the wing, and Laser Doppler Velocimetry (LDV); to measure the average velocity flowfield.
4.4 Computational Set-Up

4.4.1 Geometry

A half-model with its symmetry plane at the root chord was used for the numerical calculations (Fig. 4.7). It is understood that the flow is not symmetrical and that interaction of the port and starboard vortices lead to different breakdown positions on each side. This cannot be modelled assuming symmetry, however, the aim of this study is to test the capability LES to model the structures that make up the leading edge vortex. Also, interaction between the vortices is likely to occur downstream, once breakdown has occurred as the vortices grow considerably in size after this point. The formation of the vortices upstream of the breakdown is of more relevance to this study, at which point there is little or no interaction. The geometry of the vortex generator case which follows on from the delta wing study will resemble half a delta wing.

The size of the domain used for the simulation was $20c \times 5c \times 10c$, which was found to be sufficient by Morton[20]. As depicted in Fig. 4.8, the inlet and outlet boundaries were placed $10c$ lengths upstream and downstream of the apex. The symmetry plane of the delta wing coincided with one of the side boundaries, with all other boundaries placed $5c$ from the apex. The numerical calculation was set-up similar to the experiments, with a delta wing at $27^\circ$ angle-of-attack, and freestream velocity $U_\infty$ of $24ms^{-1}$, giving a Reynolds number of $1.56 \times 10^6$ based on root chord.

4.4.2 Grid Generation

Grids consisting of tetrahedral cells and prisms were generated using the software CENTAUR which is capable of generating unstructured grids with geometric sources for grid refinement in desired regions. The grids were created so that there was
4.4. **COMPUTATIONAL SET-UP**

![Figure 4.7: Delta wing model for computation](image)

Figure 4.7: Delta wing model for computation

![Figure 4.8: Domain for delta wing](image)

(a) side view

(b) front view

Figure 4.8: Domain for delta wing.

...a concentration of nodes near the surface to capture the wall boundary layer and a concentration of nodes in the expected vortex position using geometric sources. Although hexahedral grids are normally preferred for LES calculations due to the increased numerical damping of tetrahedral grids, the latter was more suitable for the delta wing geometry as it allowed for a high level of grid refinement in the vortex region. Attempts to generate hexahedral grids led to poor cell quality due to the triangular shape and sharp edges of the wing. Cells around the apex were of very poor quality and a stable calculation was not achieved.

Placing geometric sources in the domain is well suited for the refinement of separated flow regions such as that over a delta wing. A recent DES study by Morton...
Figure 4.9: Delta wing showing frustum shaped geometric sources for refinement.

[20] showed that a tetrahedral grid of 3.5 million cells refined in the core of the vortex through Adaptive Mesh Refinement (AMR) gave similar results to a $1 \times 10^7$ cell grid that was refined across the whole domain. This method allows the cells in the vortex core to be very fine whilst keeping within reasonable grid sizes that are feasible to run on current computing resources.

For this study, two frustum shaped sources were created as shown in Fig. 4.9: one along the leading edge of the delta wing, where the primary vortex was predicted to form and the second around the sharp leading edge where the shear layer separates. The position of the sources was based on the Adaptive Mesh Refinement (AMR) grids of Morton [22] followed by a few test runs to ensure that the vortex was completely captured by the refinement. CENTAUR does not have AMR capabilities, although such grids carry the risk of the vortex changing as the grid changes, which was also observed during the test runs. To ensure the vortex was captured completely, the geometric sources were constructed as large as possible.

Details of the 8 million and 15 million node grids that were generated are presented in Table 4.1. The former consisted of 13 layers of prisms around the delta wing with a first cell size of $0.05mm$ and a stretching ratio of 1.8. This ratio was quite high since it took a few iterations to reach a combination of parameters that did not result
4.4. COMPUTATIONAL SET-UP

Table 4.1: Grid parameters for tetrahedral grids of the delta wing.

<table>
<thead>
<tr>
<th>Nodes ×10^6</th>
<th>Surface</th>
<th>Prism Layers</th>
<th>Geometric Source</th>
<th>Tetrahedral</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Prisms</td>
<td>Stretch</td>
<td>Layers</td>
<td>Max size</td>
</tr>
<tr>
<td>8</td>
<td>3.5mm</td>
<td>0.05mm</td>
<td>1.8</td>
<td>13</td>
</tr>
<tr>
<td>15</td>
<td>1.2mm</td>
<td>0.1mm</td>
<td>1.1</td>
<td>13</td>
</tr>
</tbody>
</table>

Figure 4.10: 15 million node grid showing frustum shaped source with node clustering.

in a grid too large to be generated by CENTAUR. The size of the tetrahedral cells in the geometric source was set to 1.7mm. A stretching ratio of 1.35 was set for the tetrahedral grid growing into the freestream, resulting in a maximum cell size of 60mm. A cross section of the grid is presented in Fig. 4.11b.

For the finer mesh, the first cell was doubled in size, but the growth of the prism layers was controlled by setting the stretch ratio to a value of 1.1. The radius of the frustum shaped source was increased so that it captured a larger area around the vortex, the cell size within the source was scaled by 0.7 and the stretch ratio for the tetrahedral grid was increased to 1.8. This resulted in an increased level of refinement around the delta wing and in the vortex region (Fig. 4.11a) where gradients are high, with a quick expansion into the freestream (Fig. 4.11c).

At the apex, the 15 million node grid has a first cell height that gave a y^+ ≈ 3 at the leading edge and and y^+ ≈ 6 at the root chord. Towards the trailing edge, y^+ ≈ 2 at the leading edge and and y^+ ≈ 3 at the root chord. The minimum y^+ ≈ 0.5 at
4.4. COMPUTATIONAL SET-UP

Figure 4.11: Comparison of 15 million and 8 million node grids at $x/c = 0.48$ showing different levels of refinement in the vortex region and top section of the domain.

approximately 60% span on the pressure side of the wing. The streamwise spacing varies between $x^+ \approx 36$ and $x^+ \approx 72$, as does $z^+$. These values compare reasonably well to the grid size applied by McMullan and Page [90] with $x^+$ and $z^+$ approximately a factor of 4 smaller than the grid size for the diffuser in Chapter 3.
4.4. COMPUTATIONAL SET-UP

4.4.3 Boundary Conditions

A subsonic velocity inflow was applied at the inlet boundary and specified through a total pressure and temperature condition. The total pressure was set to $1.027 \times 10^5 \text{ Pa}$ based on an inlet velocity of $48 \text{ ms}^{-1}$ (twice the value of experiments) and air density of $1.226 \text{ kgm}^{-3}$, whilst temperature was set to $293 \text{ K}$. Since Hydra is a compressible solver, the time-step tends to be relatively small leading to longer running times. In order to reduce computational time, the velocity used in the calculations was twice the value of that used in the experiments and the geometry used in the experiments was halved so that the root chord $c$ was 0.475 ensuring the calculation was run at the experimental Reynolds number. This increases the Mach number from 0.075 to 0.15, and since the latter can still be considered incompressible, this does not introduce an error in the calculation. The delta wing was modelled as a viscous, stationary wall, while the sides, ceiling and floor of the domain were set to inviscid walls.

4.4.4 Calculation

The Smagorinsky constant was initially set to $C_s = 0.15$ and reduced to $C_s = 0.1$ once the calculation was stable. The time step for the coarse grid and fine grid was $4 \times 10^{-9}$ seconds and $1.5 \times 10^{-7}$ seconds respectively. The larger time step for the finer mesh is due to a larger first cell height on the surface of the wing. The initial LES flow field was obtained from a reasonably converged RANS solution in order to reduce computational time.

The calculation ran over 288 processor cores on 24 nodes each with 24GB memory and connected by an Infiniband network. If one ‘flow pass’ is defined as the time for a particle to travel from the apex to the trailing edge of the wing, then the finest grid was run for 10 ‘flow passes’ before averaging commenced to ensure that the flow was
completely unsteady before the sampling started. It took a further 25 ‘flow passes’ to obtain a reliable mean with the computation taking around 650 hours in total.

4.5 Results

4.5.1 Subgrid-scale Viscosity Ratio

The LES of the delta wing was assessed by means of subgrid-scale (SGS) ratio, similar to the diffuser study. Although the SGS ratio normally increases in vortex regions due to high gradients and insufficient grid resolution, Fig. 4.12 shows areas of SGS ratio between 0 and 6 in the vortex position. This is due to the high level of refinement that was achieved with the geometric sources used in the grid. This means that unrealistic large length scales normally modelled in such regions, were not an issue in this simulation. The SGS ratio is observed to increase as the size of the cells increases quite rapidly from the surface of the geometric source. This is due to the Smagorinsky model having SGS viscosity that varies with the square of cell size, however, the high SGS ratio is not in the area of interest. Low levels of modelled viscosity is evidence of reliable LES results.

4.5.2 Vorticity Magnitude

Figure 4.13 shows an iso-surface of vorticity magnitude coloured by spanwise vorticity for the instantaneous flow. When comparing it to the DES iso-surfaces in Fig. 4.14, one can appreciate that the fine structures could only be captured by LES and cannot be captured as well by DES or RANS. Numerous structures were resolved along the leading edge of the wing, making up the leading edge vortex. Grids of higher resolution were used in the LES, with the finer grid having 6.3 times more cells than that used in
4.5. RESULTS

Figure 4.12: Mean subgrid-scale (SGS) viscosity ratio ($\mu_{SGS}/\mu$) for LES on a perpendicular plane to the wing at four chordwise stations.

The DES and the coarse grid being 3.3 times finer. A structure that can be observed spiralling around the primary vortex was also captured clearly by the LES. This is formed due to shedding from the sharp leading edge of the wing which is entrained into the main vortex. The iso-surface also shows signs of vortex breakdown at some position in the rearward half of the wing, where turbulence levels increase. Further details on the structures mentioned in this section will be described in detail in the following sections.
Figure 4.13: LES instantaneous iso-surfaces of vorticity magnitude coloured by axial vorticity for three views of 70° delta wing half model.
Figure 4.14: DES instantaneous iso-surfaces of vorticity magnitude coloured by axial vorticity for three views of 70° delta wing G9A4 in freestream (no wind tunnel walls, no sting) [20].
4.5. RESULTS

Figure 4.15: Constant $C_P$ lines on the leeward surface of the delta wing

4.5.3 Surface Pressure

Figure 4.15 shows the mean surface pressure coefficient on the leeward surface of the wing for the 15 million node grid and the experiments [18]. The LES results were mirrored to give a complete wing, hence the starboard and portside of the wing are identical. The surface pressure for the experiments is an interpolation of the pressure recorded from the pressure tappings depicted in Fig. 4.6c. A strong suction peak is observed at around 60% of the wing span for the LES, slightly further inboard than that recorded for the experiments which was situated at 66% of the wing span. This suction is coincident with the location of the leading edge vortex.

The LES shows a second suction peak between the primary vortex and the leading edge indicating the existence of a secondary vortex. The experiments [18] do discuss secondary peaks near the leading edge, however, they are not reflected in their surface pressure contours. This is probably because of an insufficient number of pressure tappings. Figure 4.15a shows that in the experiments stronger levels of suction were sustained towards the trailing edge when compared to the LES. Further analysis of surface pressure coefficient can be made through the plots in Fig. 4.16.
Figure 4.16: Pressure coefficient along the wing span at four chordwise stations on the leeward surface of the delta wing.

Figure 4.16 shows the pressure distribution along the span of the wing at four chordwise positions (0.53L, 0.63L, 0.74L and 0.84L). The span of the wing is non-dimensionalised by $e$ a function of the span of the wing, such that $z/e = 0$ is the root chord and $z/e = 1$ is always at the leading edge. Pressure coefficient plots are presented for the for the LES grids, the DES[20] and the experiments[18].

At $x/c = 0.53$ and $x/c = 0.63$ (Fig. 4.16a and 4.16b), two negative pressure peaks are observed along the span of the wing, the strongest one in the position of the leading edge vortex and the weaker one in the position of the secondary vortex as previously observed in Fig. 4.15. At $x/c = 0.74$ and $x/c = 0.84$ (Fig. 4.16c and 4.16d), the pressure increases (becomes less negative), with that of the secondary vortex nearly disappearing.
4.5. RESULTS

The 8 million node grid is in good agreement with the levels of pressure achieved by the DES, whilst the 15 million node grid reports higher levels of pressure (less suction). Both LES grids predict a vortex position which is slightly further inboard than that for the DES. The exact position of the pressure peak, and hence vortex position for the experiments is not clear from the pressure plots as there aren’t enough pressure tappings along the span of the wing to capture such detail.

The 15 million node LES grid underpredicts the pressure peak by 32% and the DES by 24% when compared to the lowest recorded pressure reading from the experiments. This is not necessarily an error though as several authors of numerical delta wing studies [20, 113, 105] have also failed to obtain similar values of surface pressure coefficient with a discrepancy ranging between 22.4% and 24% from the ONERA experiments [18]. Morton [20] has suggested that the discrepancy might be due to different scaling of dynamic pressure [20]. In an attempt to match the experiments, he scales his surface pressure coefficient by a factor of $\sqrt{2}$ to achieve identical suction to the experiments. It should also be noted that the experiments did not make any blockage or wall corrections which may also be contributing to the discrepancy.

One of the reasons that may cause the difference in pressure between the two LES grids is the pressure peak from the secondary vortex, which is captured more strongly by the finer grid. Le Roy et al. [105] suggested that an over-prediction of the secondary vortex may lead to an under-prediction of the primary vortex. The fine grid may be predicting structures which were not captured by coarser grids and hence is not necessarily an over-prediction. Although the DES plots do not reflect the existence of a secondary vortex, evidence of one is present in the flow visualisation. Oil flow visualisation from the experiments (Fig. 4.17b) also show signs of a secondary vortex, however, this is not reflected in the pressure coefficient plots either.
4.5. RESULTS

(a) LES

(b) Surface oil-flow visualisation [18]

Figure 4.17: Comparison of streamlines on the surface of the wing.

4.5.4 Surface Streamlines

Figure 4.17 shows a comparison of streamlines on the surface of the wing for the LES and experiments [18]. The LES shows a primary separation line which is fixed at the leading edge of the wing due to the sharp geometry and a primary attachment line at the root chord as observed in the experiments. Streamlines are swept outboard and a secondary separation line as denoted in Fig. 4.17a occurs inboard of the leading edge. This is similarly seen in the oil flow visualisation by the dark border between the green and red colours. A secondary reattachment line is observed immediately inboard of the leading edge and a tertiary separation which is not very obvious is present between the secondary separation and reattachment lines. The LES shows disorganisation of streamlines towards the trailing edge of the wing, indicating that breakdown has occurred. The front half of the wing compares reasonably well with the oil flow visualisation, however, the disorganisation is not captured by the oil flow visualisation. In the experiments [18] the change of direction of the skin friction lines near the secondary separation lines is said to indicate laminar to turbulent transition.
4.5. RESULTS

4.5.5 Mean Axial Velocity

In Fig. 4.18, mean axial velocity \( u/U_\infty \) for the LES (left) and experiments (right) is compared in the core of the vortex at a series of slices perpendicular to the leeward surface of the wing. In Fig. 4.18a and Fig. 4.18c, upstream of the breakdown, a circular region of high velocity (up to three times freestream velocity) is observed, with velocities being slightly lower than those of the experiments (fig. 4.18b and fig. 4.18d). At the edge of the wing there appears to be another circular (blue) region with lower values of velocity from the secondary vortex. This is not visible in the LDV results at the same spanwise positions as the images are trimmed and exclude the region right above the wing. However, secondary separation close to the tip is visible in the oil flow visualisation in fig. 4.17b, at the border of the green and pink regions.

As described in section 4.2, vortex breakdown is characterised by a rapid deceleration of axial velocity in the vortex core, followed by recirculation of the flow. At \( 0.74L \) vortex breakdown has occurred and recirculation can be observed in Fig. 4.18e and Fig. 4.18g, with the recirculation region growing in diameter as it progresses downstream. The velocity contours at these two positions compare very well with the experiments (fig. 4.18f and fig. 4.18h) both in size and contour level.

In Fig. 4.19, a plane along the vortex core was extracted so that \( u/U_\infty \) can be studied along the vortex core. \( u/U_\infty \) compares well with experiments and the main characteristics of vortex breakdown can be observed. A rapid decrease in axial velocity and rapid increase in \( TKE/U_\infty^2 \) can be seen at the break down position, followed by a wake like flow. The vortex breakdown position for the experiments, taken at the point where axial velocity reached zero, occurred at \( x/c = 0.65 \) (±0.05), whereas in the LES this occurred at \( x/c = 0.73 \), 4.3% downstream of the upper limit.
4.5. RESULTS

Figure 4.18: Mean non dimensional axial velocity component \((u/u_\infty)\) for LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations.
4.5. RESULTS

(a) LES

(b) LDV - Mitchell [18]

(c) DES - Morton [20]

Figure 4.19: Mean non-dimensional freestream velocity \( \frac{u}{U_\infty} \) for the LES and experiments in a horizontal plane passing through the vortex core

4.5.6 Axial Vorticity

Figure 4.21 shows the axial component of mean vorticity \( \Omega_x = \frac{\partial W}{\partial y} + \frac{\partial V}{\partial z} \) at each of the perpendicular planes described above for the LES (left) and the experiments (right), both representing averaged data. The axial vorticity was non-dimensionalised by root chord and velocity. The LES is presented using the same scale as the DES contours for comparison purposes. There is a difference in scale between the numerical (LES and DES) and the experiments of a factor of \( 10^3 \). This must be down to a scaling error rather than large discrepancy in results since the vorticity is composed of velocity
4.5. RESULTS

Gradients which match up very well with experiments as observed in Fig. 4.18.

At the 0.53L and 0.63L (Fig. 4.21a and Fig. 4.21c) the traces reveal regions of negative axial vorticity in the primary vortex core and a smaller region of positive vorticity very close to the leading edge of the wing, suggesting that a secondary vortex is rotating in the opposite direction to that of the main vortex. Although not discussed in detail, evidence of a secondary vortex has appeared in previous delta wing studies. The vorticity contours of Mitchell et al. [22] show similar regions of positive axial vorticity, whilst oil-flow visualisation from Mitchell’s experiments [18], presented in Fig. 4.17b. At 0.74L and 0.84L (Fig. 4.21e and Fig. 4.21g) vorticity in the core is much weaker than it was upstream which is evidence that breakdown has occurred, analogous with the lower velocity contours in figure 4.18 at the same positions.

Vortical substructures consisting of negative vorticity are observed in Fig. 4.23, around the vortex core in the same position that the ‘pockets’ of velocity were observed in Fig. 4.18a. These substructures were observed in the experiments [18] and DES study by Mitchell et al. [22]. They are spatially stationary and originate from the separated shear layer. They follow a helical trajectory which was captured in Fig. 4.13. At the 0.53L plane the LES resolves 4 vortical substructures around the primary vortex, whereas 5 can be observed in the experiments [18] (Fig. 4.21b).

In the experiments, the vorticity levels in the substructures pre and post break down were similar (Fig. 4.21), however, in the LES vorticity in the substructures decreased in the downstream direction after break down occurred, similar to that observed in the DES study[22]. After break down, the vortex has grown significantly in size and the geometric source containing the cell refinement is not large enough to capture the substructures accurately. Figure 4.23 shows axial vorticity contours at the final two positions using a more suitable scale for the resolved substructures.

Mitchell et al. [22] suggested that the sub-structures formed due to a local in-
stability near the leading edge and were entrained downstream by the axial and rotational velocity of the leading-edge vortex, giving them a helical trajectory. These sub-structures in the shear layer may be linked to the discrete vortices in Fig. 4.20 previously observed by Gad-el-Hak and Blackwelder [21] on a delta wing at 10° angle-of-attack at a Reynolds number of $1.3 \times 10^4$ based on root chord. The authors proposed that discrete vortices shed off the leading edge of the delta wing either set up the initial shear layer for the primary vortex or possibly merged to give a large vortex.

Spatially stationary discrete vortices in the shear layer were also observed by Honkan and Andreopoulos [114]. The authors proved experimentally for the first time, the existence of fluctuating vorticity of alternating sign generated on the surface of the wing which resulted in secondary vortices. They also found that the re-attachment point of the shear layer, which moved closer to the centre of the wing with increasing $\alpha$, gave rise to intense turbulence. The mechanism generating the vortical substructures within the shear layer and their influence on the leading-edge vortex is still not clearly understood with numerical results [22] being very mesh dependent.

Figure 4.22 compares instantaneous axial vorticity from the LES (left) and DES (right). The results are quantitatively in very good agreement with each other both
4.5. RESULTS

pre breakdown and post breakdown. The most significant difference is that the LES has resolved much finer structures than the DES, especially in the secondary vortex region at the leading edge of the wing. At the first two positions, the vortical substructures appear in the same position as they did in the mean flow, confirming that they are spatially stationary. At the final two positions, the disorganisation in the vortex core indicates that breakdown has occurred and the sub-structures have become weaker as observed in the mean flow.
4.5. RESULTS

Figure 4.21: Mean non-dimensional axial vorticity component ($\Omega_x c / U_\infty$) for LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations.
4.5. RESULTS

Figure 4.22: Instantaneous non-dimensional axial vorticity component \( \Omega_x c / U_\infty \) for LES (left) and DES [22] (right) on a perpendicular plane to the wing at four chordwise stations.
4.5. RESULTS

4.5.7 Turbulent Kinetic Energy

Contours of Turbulent Kinetic Energy extracted from the mean flow at the four perpendicular planes are presented in Fig. 4.24 for the LES (left) and experiments [18] (right). When using identical scales, the LES compares well with the experiments [18] predicting even the low levels of TKE observed in the core of the vortex at the 0.53L (Fig. 4.24b), which regular DES methods struggled to do. At 0.63L, TKE for the LES (Fig. 4.24c) increases rapidly and is much higher than that observed in the experiments (Fig. 4.24d). This is consistent with the lower axial velocity contours in the vortex core at this position which indicates a more dissipative core.

Figure 4.23: Mean non-dimensional axial vorticity component ($\Omega_x c/U_\infty$) for LES on a perpendicular plane to the wing at four chordwise stations.
4.5. RESULTS

At 0.74L, immediately downstream of the breakdown, the LES (Fig. 4.24e) and experiments (Fig. 4.24f) compare very well with a relatively large red region of high TKE in the core. TKE levels up to 1.2 were observed in the LES which questions whether the experiments also captured higher values since the contours are in such good agreement and the highest level of TKE occupies a relatively large area around the core. At the final position, 0.84L (Fig. 4.24g and Fig. 4.24h), further downstream of the vortex breakdown, TKE levels decrease and a disorganised flow is observed.

Figure 4.25 shows a plot of TKE levels along the vortex core for the LES and the finest grid (without AMR) used for the DES [20]. A rapid increase in TKE was observed for both methods around the region where breakdown occurs due to the oscillation of the vortex breakdown position. The peaks of the plots for the two simulations are very close to each other with regards to position, however, the LES value is more than twice the DES value. The DES study [20] showed that the peak value of TKE increased significantly with grid refinement which suggests that higher refinement would have probably resulted in higher maximum TKE values. One of the DES studies [20] states that the experimental value for TKE at the breakdown position was approximately 0.5, however, this value has been assumed based on the TKE contour in Fig. 4.26c which caps the contour variable at 0.5. Based on the observation that when capped at this value the TKE contours for the LES and experiments look very similar, it is assumed that higher values were probably achieved in the experiments also.

In Fig. 4.26, TKE contours on a horizontal plane passing through the core for the LES, DES and experiments showed very similar results. The LES captured the asymmetry that appeared in the experiments and was also successful in picking up the lower levels of TKE along the leading edge of the delta wing, which did not appear in the DES. This is the TKE being produced by the separated shear layer, which is then entrained into the vortex, energising the flow.
Figure 4.24: Mean non-dimensional total turbulent kinetic energy ($k/U_\infty^2$) for the LES (left) and experiments [18] (right) on a perpendicular plane to the wing at four chordwise stations.
4.5. RESULTS

Figure 4.25: Resolved turbulent kinetic energy \( (k/U_\infty^2) \) along the core for the LES and Morton’s DES [20]

![Graph showing resolved turbulent kinetic energy](image)

Figure 4.26: Mean non-dimensional turbulent kinetic energy \( (k/U_\infty^2) \) for the LES and experiments in a horizontal plane passing through the vortex core.

![Images of LES and LDV data](image)

(c) DES - Morton [20]
LES of a delta wing was successfully carried out at experimental Reynolds numbers. This is the first LES study to present such extensive results on this case which are in good agreement with experiments. Very fine structures (braiding) around the vortex core were captured, showing improvement over DES studies. Axial velocity, axial vorticity and turbulent kinetic energy for the LES, experiments and DES were compared at perpendicular slices to the wing.

High levels of agreement were achieved between the LES and experiments. The primary vortex and secondary vortex were modelled successfully with opposite signs of vorticity showing the different sense of rotation between the two. Substructures around the vortex core were also captured in the mean flow of the LES.

The main weakness in the calculation was the discrepancy in surface pressure along the span of the wing. The LES seems to overpredict the suction of the secondary vortex as compared to the DES, leading to lower suction of the primary vortex. The experiments do not record the suction of the secondary vortex because there were not enough points along the span to capture such detail. The inability to capture the correct suction is a trait of all computational studies so far, with DES studies being in error by a factor of $\sqrt{2}$.

The position of the sudden increase in TKE at the breakdown position was captured accurately, but values were much higher than those observed in the experiments and DES studies. However, contours of TKE in the vortex core compare very well with experiments.

The results from this study gave insight into the vortical structures on the leeward surface of a delta wing and provided evidence of LES being able to capture such flows accurately. The successful simulation of the vortical flow problem provides confidence.
that the numerical approach will be suitable for the prediction of a vortex generator in ground effect.
Chapter 5

Vortex Generator in Ground Effect
5.1 Introduction

Vortex Generators are increasingly being used on race cars, especially in the underbody region, to generate vortices that interact with the flow. When placed at an angle to the onset flow, a tip vortex forms that extends along the underside of the car as shown in Fig. 5.1. An example of these vortex generators is the pair of ‘L’ shaped aerodynamic devices depicted in Fig. 5.2 that are attached to the underside of the front nose, commonly referred to as turning vanes. One of their roles is to capture small vortices that are generated from the numerous elements of the front wing, which then coalesce with the larger vortex from the vortex generator itself. These devices are set so that one of the tips directs ‘dirty’ flow away from the underbody region and the other tip is used to generate a strong vortex that extends into the flat underbody region to supply the rear diffuser with an attached flow and increased downforce. In this chapter, a simplified version of such a vortex generator will be modelled in order to investigate the vortex in close proximity to the ground, and also to assess the effect of modelling the ground as a rough surface with cavities to represent the tarmac on a real race track. The vortex will decay in strength as it travels under the car, however, the factor of increasing decay due to the rough ground is unknown.
5.2. EXPERIMENTS

Figure 5.2: Red Bull Renault rb6 F1 with ‘L’ shaped vortex generators under front nose.

5.2 Experiments

Although this study does not replicate the geometry of any particular experimental study, studies on trapped vortices in ground effect [38, 8, 24] were used to define relevant dimensions for the geometry and to give some insight into the flow that is generated by such devices. The vortex generators in the above mentioned studies are aimed at representing devices attached to the lower surface of the vehicle as seen in Fig. 5.3.
5.2. EXPERIMENTS

Figure 5.3: Typical application of large scale vortex generators on the lower surface of an open wheel race car [24].

The tests [24] were carried out using the wind tunnel set-up depicted in Fig. 5.4a, where the vehicle’s underfloor was represented by a flat plate with rectangular vortex generators mounted on top of it to generate the streamwise vortices. The plate was mounted onto a six-component balance by three struts. The inverted set-up allowed for direct measurement of forces since the plate is mounted directly onto the force balance, however, this eliminates the possibility of having a moving ground plane. The ground plane could move up and down by means of actuators to vary the ride-height, which was measured from the upper tip of the vortex generator to the ground plane above it.

The wind tunnel blockage for the model was less than 1%. Two vortex generators on each side of the symmetry plane were used as seen in Fig. 5.4b as it was previously suggested by Rossow [36, 37] that this was required to stabilise the vortex. The yaw angle and spanwise position of the vortex generator were adjustable, but their vertical orientation was fixed at 90 degrees to the flat plate. The vortex generators were tested at 10, 20 and 30 degrees for a range of ride heights, and the spacing between vortex generators was also varied.

Tests were carried out in the San Diego State University low-speed wind tunnel.
5.2. EXPERIMENTS

(a) Wind tunnel test section

Figure 5.4: Experimental set-up showing flat plate and vortex generators.

which has a test section of 0.813m in height and 1.143m in width. The wind speed was set to 53.6ms$^{-1}$ and turbulence levels were about 2%. The vortex generators had a length of 0.152m and height of 0.025m, and the flat plate had a length of 0.762m and a width of 0.406m. The Reynolds number based on vortex generator length and wind speed was 543,000. The accuracy of the six-component balance was ±0.004 for $C_L$, ±0.002 for $C_D$, and ±0.003 for $C_M$.

The experiments found that a pair of vortex generators on each side of the car was not necessary to stabilise the vortex as previously thought but having an additional
vortex generator lead to higher levels of downforce [8]. Increasing the vortex generator
yaw angle resulted in increased downforce, however, the lower angles were less sensitive
to stall at low ride-heights [8]. This is an important consideration in motor sport
since the ride height of a race car varies as it laps round the race track. Gothic, ogive,
parabolic and triangular shaped vortex generators were also tested [24] and a relation
between surface area and downforce generation was established. The larger surface
areas such as the rectangular and gothic shapes gave higher values of downforce than
the triangular shapes which have half the surface area [24]. Similar vortex generators
were modelled for the computational case, however, the parameter ratios were based
on Formula 1 car ‘L’ shaped vortex generators.

5.3 Computational Set-Up

5.3.1 Geometry

Vortex Generator

The computational case takes the information gained from the experiments [38, 8, 24], but is also highly linked to the delta wing study as the half model of the delta
wing is similar in geometry and also generates similar flow features to that of the
vortex generators described above. Garcia and Katz [8] found that the streamwise
vortices generated by the vortex generators are analogous to the leading edge vortices
generated by a delta wing. If the orientation of the delta wing half model is rotated
so that it protrudes vertically downwards like a fin, where the angle of attack becomes
the yaw angle, then it would resemble a vortex generator found on the underside of
race cars.

Since the LES method was validated against experimental work for the delta wing
5.3. COMPUTATIONAL SET-UP

In Chapter 4, it could be readily applied to a similar geometry set-up. This was important since the experimental data available on vortex generators is not extensive enough for the LES to be compared to. For this reason, the geometry chosen for the computational work did not have to be identical to that of previous trapped vortex studies and an appropriate set-up based on analytical observation of current full scale Formula 1 cars and information from the experiments [38, 8, 24] was chosen to create a simplified, generic version of a vortex generator.

In 2010 a pair of ‘L’ shaped vortex generators (one on each side) attached to the underside of the front nose was introduced by several Formula 1 teams, such as those seen on the Red Bull Renault RB6 F1 shown in Fig. 5.2, Team Lotus T127 and Williams FW32. Similar rectangular devices were observed on the Sauber C29 and McLaren MP4-25 also raced in 2010, which resemble the bottom section of the ‘L’ shape. For simplicity the simpler version was modelled for the LES simulations, including the slope in the vertical direction seen on most cases. This resulted in a gothic shaped vortex generator similar to that tested by Katz and Morey [24]. The elimination of one of the two sharp edges from the ‘L’ shape avoids the generation of an additional vortex and any subsequent interactions, since the scope of this study is to investigate the interaction of the vortex with different ground surfaces.

One side of the underbody was modelled as this was sufficient for the purpose of this study. The underfloor was modelled as a flat surface with the vortex generator attached to it a yaw angle of 20 degrees. This angle was chosen based on findings from the delta wing and vortex generator studies. In a delta wing study by Mitchell [18], the author found that lift increased and vortex breakdown position moved forward as angle-of-attack increased, until at a critical angle the wing stalled. However, at angles of 20 degrees and lower, breakdown occurred downstream of the wing rather than on the surface of the wing. Similarly, in their underbody vortex generator study, Garcia and Katz [24] found that at larger angles of yaw (30 degrees), the vortex generator
generated higher levels of downforce but became very sensitive as ride-height was decreased and stalled at higher ride-heights than the lower angles did.

The dimensions for the vortex generator were based on the Reynolds number desired for the calculation. The vortex generator depicted in Fig. 5.5 had a length of 0.05\(m\), the height at the leading edge was 0.23\(L\), the height at the trailing edge was 0.5\(L\) and a radius of 0.04\(L\) was applied front and rear corner. The angle in the vertical direction from the leading to trailing edge was 15 degrees. The thickness of the vortex generator was 0.015\(L\) and the edge had a fillet of 0.0075\(L\) all the way round.

The domain shown in Fig. 5.6 was modelled to represent a section of the underbody region, with the top surface representing the underfloor, the bottom surface representing the ground and the inlet and outlet representing a position upstream and downstream of the vortex generator. The size of the domain was 7\(L\) × 3\(L\) × 0.75\(L\), where the height of the domain represents the ride-height of the car above the ground. This was based on the lowest possible plane that any part of the car can lie on as required by the FIA regulations. The inlet and outlet were placed 1\(L\) upstream and 6\(L\) downstream from the front of the vortex generator respectively. The front end is positioned in the centre of the spanwise direction and attached to the top surface of the domain.

The dimensions of the modelled vortex generator resulted in a Reynolds number of 220,000 based on vortex generator length and freestream velocity. The Reynolds number of a full sized vortex generator of this kind on a full sized car travelling at 180\(mph\) would be 1.6 \(\times 10^6\), similar to the diffuser-equipped bluff body and delta wing cases. This case was run at a lower Reynolds number than the diffuser case since previously the grid was not fine enough to capture all the fine vortical structures that occur at such high Reynolds number. In the delta wing case, although the grid was capable of resolving even very fine structures, Centaur allowed for very high grid
refinement in the vortical regions. The same method of grid generation could not be achieved in the vortex generator case, since the STL file format could only be read by **ICEM CFD** in which the hexahedral meshing tool is more effective than tetrahedral meshing. Other LES studies on automotive flows have shown that running at such cases at lower Reynolds numbers can still give successful results [9, 44, 45]. The Reynolds number for this case is still representative of several aerodynamic devices found on the car, such as the small winglets found on the front wing.

**Rough Ground**

A rough ground plane representing the tarmac of a race track was modelled by generating a three dimensional scan of a small section of it using a *Handyscan 3D* by *Creaform* [115]. This is a portable 3D scanner based on the alignment of 3D curves of the part being scanned which are used to generate a model that can be exported as an STL file. *Handyscan* self-positions itself by means of triangulation to determine its relative position to the part in real time. It uses a positioning model made up of several positioning features (silver dots) which are affixed to the object before scanning. Whilst scanning, the positioning targets are added to the positioning model. The surface is created in STL format by using many small triangles formed.
5.3. COMPUTATIONAL SET-UP

Figure 5.6: Domain representing underbody region of a vehicle with a vortex generator attached to underbody surface.

by 3 vertices in a 3D coordinate system determined by the first sensor position. A scanning volume bounding box can be resized depending on the object being scanned and the resolution of the scan can be adjusted according to the level of detail required. A medium surface resolution of 1.95\,mm was used for this model as the finer resolution required too much memory.

Before scanning, the scanner was configured to set the sensor’s laser power and camera shutter time which is specific to the surface being scanned. The sensor was calibrated by taking measurements of a calibration plate that comes with the scanner. Reflective targets are randomly distributed on the object being scanned so that the scanner can identify its position in space.
5.3. COMPUTATIONAL SET-UP

Figure 5.7: Generation of rough ground geometry to replicate a tarmac surface.

The process followed to obtain CAD for the tarmac is depicted in Fig. 5.7. The scanner was sensitive to dark surfaces so initially, a layer of white paint was applied to a small section of tarmac in order to scan the tarmac directly. However, the cavities in the tarmac created shadows which resulted in holes (missing pieces) in the scanned surface. Since a direct scan of the tarmac was not possible, a mould of a section approximately $0.05m \times 0.05m$ was made using Siligum mould. Siligum has a plasticine texture which enabled the material to be worked into the surface to achieve the detailed structure of the tarmac and was then left to dry to form a strong mould.
5.3. COMPUTATIONAL SET-UP

Initially reflective targets were placed on the mould, however, since the mould was relatively small, it was difficult to obtain the correct distribution. Instead, the mould was placed in the centre of the calibration plate which already contained a distribution of targets, enabling the scanner to identify its position. After the scanning was complete, the STL file was read into *ICEM CFD* so that the geometry could be tidied up. The geometry was trimmed so that a section without any holes was selected, inverted and mirrored 70 times in the streamwise direction and 30 times in the spanwise direction in order to create a rough ground surface for the domain as shown in Fig. 5.7f.

Figure 5.8 represents a side view of the rough surface domain, showing the edges that were created to generate the boundaries. Since the lower edge of the domain is not straight, it was not possible to generate the side of the domain as one surface as this resulted in holes at the intersection of the side and ground surfaces. A number of vertical edges were set up at intervals in the streamwise direction which allowed the side surface to be created in sections. A horizontal edge is observed splitting the domain in the vertical direction, which was created for meshing purposes and will be discussed in the following section.

5.3.2 Grid Generation

For the smooth ground calculations, a hexahedral grid of 40 million nodes was generated using *ICEM CFD* in the same way as the diffuser case in Chapter 1.3.1. An O-grid type mesh was created around the vortex generator in order to control the
5.3 COMPUTATIONAL SET-UP

Table 5.1: Grid Parameters for hexahedral grid of vortex generator over smooth ground.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>1st Cell Normal to Wall</th>
<th>Stretch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vortex Generator</td>
<td>0.02</td>
<td>1.15</td>
</tr>
<tr>
<td>Smooth Ground</td>
<td>0.1</td>
<td>1.2</td>
</tr>
<tr>
<td>Underbody surface</td>
<td>0.1</td>
<td>1.2</td>
</tr>
</tbody>
</table>

Table 5.2: Grid Parameters of unstructured tetrahedral grid near rough ground.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Maximum Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rough Floor</td>
<td>0.2</td>
</tr>
<tr>
<td>Smooth Ground</td>
<td>0.2</td>
</tr>
<tr>
<td>Lower Inlet</td>
<td>0.8</td>
</tr>
<tr>
<td>Lower Outlet</td>
<td>0.8</td>
</tr>
<tr>
<td>Volume</td>
<td>1.0</td>
</tr>
</tbody>
</table>

growth of the mesh in the near wall region and also to capture the radiused edges. Blocking was easily achieved, unlike the delta wing case, due to the vortex generator being a quadrilateral shape. Figure 5.9a shows the blocking around the vortex generator. The block was split in the spanwise direction as observed in Fig. 5.9b to enable refinement in the vortex region.

Details of the grid are found in Table 5.1. A first cell size of 0.02mm was set normal to the wall of the vortex generator resulting in $y^+ \approx 5$ near the root chord (top edge) to $y^+ \approx 7$ at the tip chord (lower edge). It should be noted that in this case $y^+$ is measured in the global z direction due to the orientation of the vortex generator. The minimum $y^+ \approx 0.7$ at approximately 75% span of the wing. Values of $y^+$ are similar from the leading to the trailing edge. 230 nodes were distributed uniformly along the length of the vortex generator leading to a streamwise spacing of $x^+ \approx 19$. There were 115 nodes distributed uniformly along the leading and trailing edges resulting in a spacing that varied from $z^+ \approx 12.6$ to $z^+ \approx 126$ (global y direction).

The rough ground calculation required a hybrid grid consisting of both tetrahedral elements around the rough surface, and hexahedral elements in the rest of the domain.
5.3. COMPUTATIONAL SET-UP

(a) Blocking around vortex generator

(b) Plan view of domain

Figure 5.9: Blocking of vortex generator domain.

Ideally, a similar mesh to that created for the delta wing would have been used for the vortex generator since the flow is comparable. However, the STL format of the scanned surface limited the grid generators that could be used to *ICEM CFD*.

The domain was split in the vertical direction at $y = 0.15H$ from the ground surface, where $H$ is the height of the domain, resulting in an upper and lower volume as observed in Fig. 5.7f. At the interface of the two volumes, a new surface was created. The upper volume was meshed using the blocking method and parameters used for the smooth ground calculation described in Table 5.1, and the lower section
was meshed using tetrahedral elements. First, the volume near the rough ground was
eliminated so that a hexadral grid could be generated in the upper volume, creating
a surface mesh of hexadral elements at the interface of the two volumes. The grid
on the common surface was then imported into a different file containing the lower
volume and a tetrahedral grid was generated from the existing surface elements by
applying the maximum sizes specified in Table 5.2. By using the grid on the common
surface, both grids had identical elements at the interface which were imported into
a single file and merged into a single domain. An additional 10 million nodes were
generated in the rough ground region resulting in a total of 48 million nodes.

Figure 5.10 shows cross-sections of the smooth and rough ground grids, with the
intersection of the hexahedral and tetrahedral volumes clearly visible in Fig. 5.10b.
A close up of the grid near the rough ground is depicted in Fig. 5.11. Attempts
5.3. COMPUTATIONAL SET-UP

Figure 5.11: Cross-section of tetrahedral mesh near rough ground surface.

Figure 5.12: Plan view of 0.01m section of rough ground surface.

to generate finer tetrahedral grids in the lower volume by controlling the stretching ratio in the domain were not successful due to limitations in computer resources. The ratio of nodes to elements was approximately $1:5$, which leads to a large number of elements to be generated. The plan view of a 1cm section of the grid on the rough surface in Fig. 5.12 shows a high level of refinement. The spacing normal to the rough ground surface varies since the grid is unstructured, however, an average of 0.1mm lead to a $y^+ \approx 1$ at the troughs and $y^+ \approx 5$ at the crests. The streamwise spacing varied between $x^+ \approx 2$ to $x^+ \approx 12$ and the lateral spacing from $z^+ \approx 1.5$ to $z^+ \approx 7$. 

153
5.3. COMPUTATIONAL SET-UP

5.3.3 Boundary Conditions

A subsonic velocity inflow was applied at the inlet boundary and specified through a total pressure and temperature condition. A total pressure of $1.045 \times 10^5 Pa$ based on an inlet velocity of $65 ms^{-1}$ and air density of $1.226 kgm^{-3}$, whilst temperature was set to $293 K$. The vortex generator, the flat underbody to which it was attached and the ground plane beneath it were modelled as viscous walls and the sides of the domain were set to inviscid walls since the domain represents a section of the underbody and boundary layer effects are ignored. The outlet was specified through a freestream static pressure.

Three ground conditions were investigated: 1) Stationary smooth ground, 2) Moving smooth ground and 3) Stationary rough ground. For the moving ground a velocity equal to the freestream velocity was applied to the ground plane to eliminate the boundary layer which does not exist in reality and develops in the computations as a result of the air rather than the car moving in relation to the ground. Since interaction with the ground appears to commence from the rear section of the vortex generator, the rough surface was modelled from the second half of the vortex generator downstream, whilst the upstream section was kept as a smooth ground. This delayed excessive growth of the boundary layer due to the rough ground and also saved computational time since the cavities require increased grid refinement. A moving rough ground was not modelled since it adds a lot of complexity which is not feasible with any current solvers. This would require an immersed mesh technique around the vortex generator which can be moved across a stationary rough ground, hence modelling the relative movement between the vehicle and the ground.
5.3.4 Calculation

The Smagorinsky constant $C_s$ was initially set to 0.15 and reduced to 0.1 once the calculation was stable. The user defined smoothly parameter $\varepsilon$ was set to 0.15. The time step for the 40 million node stationary and moving smooth ground calculations was $5.1 \times 10^{-8}$ seconds, whilst the 47 million node rough ground calculation had a timestep of $4.1 \times 10^{-7}$ seconds. The initial LES flow field for the stationary smooth ground was seeded from a reasonably converged RANS solution, and the moving ground flow field was seeded from the stationary ground solution. Similarly, the flow field for the rough ground case was seeded from a RANS solution.

The calculation ran over 288 processor cores on 24 nodes each with 24GB memory and connected by an Infiniband network. If one ‘flow pass’ is defined as the time for a particle to travel from leading edge to the trailing edge of the vortex generator, then the stationary and moving smooth ground calculations were run for 10 ‘flow passes’ before averaging commenced to ensure that the flow was completely unsteady before sampling. It took a further 80 ‘flow passes’ to obtain a reliable mean with the computation taking an elapsed time of around 700 hours in total. The rough ground calculation was run for 10 flow passes before sampling, and a further 40 flow passes to achieve a reliable mean, taking approximately 600 hours.

5.4 Results

Results of the vortex generator over a smooth ground (stationary and moving) and a rough ground representing a tarmac surface are presented for a Reynolds number of 220000 (based on vortex generator length and inlet velocity). Analysis of both the mean and instantaneous flow is presented through iso-surfaces and contours of velocity and vorticity at several streamwise planes. Plots of circulation and maximum vorticity
5.4. RESULTS

are used to study the vortex decay of the primary vortex over the three different boundary conditions, and insight into the flow near the rough ground is given.

5.4.1 Subgrid-Scale Viscosity Ratio

Figure 5.13 shows the subgrid scale (SGS) viscosity ratio on a plane passing through the vortex over the stationary smooth ground and Fig. 5.14 shows the SGS viscosity ratio at two streamwise planes passing through the vortex generator for the different boundary conditions. The SGS ratio is relatively low in all cases, with values not higher than 4.5 in the vortex regions where turbulence levels are expected to be high.

An SGS ratio of approximately 0.12 is observed at the wall of the vortex generator where the value should tend to 0 since turbulence is suppressed by the presence of the wall. This indicates a good level of grid refinement for an LES calculation. A maximum value of approximately 6 is observed in Fig. 5.14e and 5.14f close to the rough ground. This coincides with the region of the tetrahedral grid with a maximum cell size of 1mm. The value at the wall itself is similar to that observed on the vortex generator surface. Low levels of SGS ratio were achieved since the domain is smaller than those modelled in the previous chapters which enabled a good level of grid refinement throughout.
Figure 5.13: Subgrid-scale viscosity ratio at $z/L = 0.17$ for stationary ground
5.4. RESULTS

Figure 5.14: Subgrid-scale (SGS) viscosity ratio at streamwise planes through the vortex generator for different boundary conditions.
Figure 5.15 and Fig. 5.16 show a side view and bottom view of the flow structures from the mean flow, for the smooth stationary and moving grounds, and for the rough stationary ground. An iso-surface of vorticity magnitude coloured by streamwise velocity is used to give insight into the flow structures that are present in the flow. Due to the yaw angle of the vortex generator, flow separates off the length of the vortex generator and rolls up to form a primary vortex that runs through the length of the domain. Secondary vortices can be observed near the trailing edge of the vortex generator.

The vortices generated by the vortex generator are similar for the three different boundary conditions, however, differences can be observed near the floor. In Fig. 5.15b one can observe that the vorticity from the boundary layer has been greatly reduced along most of the ground surface by simulating a moving ground. However, 5.15c shows a greater area of the flow affected by vorticity from the stationary rough ground than that observed from the smooth stationary ground. As the oncoming flow interacts with the cavities in the ground, turbulence is generated in the near wall region leading to larger boundary layers. A moving rough ground would solve this issue but was not possible in the timescale of this research.

It is observed in Fig. 5.16 that the primary vortex aligns itself with the flow in the streamwise direction, even though the vortex generator is at a 20 degree angle to the flow. This implies that on a car, if such a vortex generator was placed at an angle high enough to generate a strong vortex and low enough to avoid vortex breakdown in the underbody region, it can be used to re-align turbulent flow which has been generated from other parts of the car. An example of this would be to re-align flow coming from the numerous devices on the front wing of a Formula 1 car. The same figure also shows the secondary vortex starting to be wrapped around the primary
5.4. **RESULTS**

vortex immediately downstream of the vortex generator.

Figure 5.17 shows an iso-surface of instantaneous vorticity magnitude for the three cases. Numerous turbulent structures are observed, making up the streamwise vortex that appears in the mean flow (Fig. 5.15 and Fig. 5.16). The vortex occupies most of the space between the underfloor and the ground. The position of the vortex core moves as it progresses downstream, and its interaction with the ground increases, which is not obvious from the mean flow images. The stationary ground cases (Fig. 5.17a and Fig. 5.17c) show vorticity generated from the boundary layer which interacts with the primary vortex. This interaction makes it difficult to separate vorticity generated by the boundary layer from vorticity of the streamwise vortex. In the moving ground case (Fig. 5.17b), where the boundary layer is eliminated, vortical structures from the streamwise vortex are visible close to the ground. This supports the idea that in reality one would expect vortex to ground interaction at such low ride-heights. This emphasises the importance of modelling a tarmac surface to achieve results which are closer to reality.

Figure 5.18 shows contours of instantaneous vorticity magnitude on a plane passing through the centre of the vortex for the three different boundary conditions. Numerous fine structures are resolved with different levels of vorticity. Similar to the iso-surfaces, these images also show how the vortex occupies a larger section of the underbody region as the vortex progresses downstream, increasing its interaction with the ground surface. In the rough ground case, the tetrahedral grid near the ground was not as fine as the hexahedral grid in the upper region, in fact the structures are not as well resolved near the ground. The point at which the rough ground starts, has acted as a trip and the boundary layer is observed to transition from a laminar to a turbulent boundary layer. The moving ground plane eliminates the boundary the vorticity generated by the boundary layer, but still shows vorticity from the vortex interacting with the ground.
5.4. RESULTS

(a) smooth, stationary ground

(b) smooth, moving ground

(c) rough, stationary ground

Figure 5.15: Side view of vortex generator showing iso-surface of mean vorticity magnitude, coloured by non-dimensional streamwise velocity $u/U_\infty$ for the three different ground boundary conditions.
5.4. RESULTS

Figure 5.16: View from bottom showing iso-surface of mean vorticity magnitude, coloured by non-dimensional streamwise velocity $u/U_\infty$ for the three different ground boundary conditions.
Figure 5.17: Side view showing iso-surface of instantaneous vorticity magnitude coloured by non-dimensional streamwise velocity $u/U_\infty$, for different ground boundary conditions.
5.4. RESULTS

Figure 5.18: Contour of instantaneous vorticity magnitude $\Omega L/U_\infty$ at lateral plane ($z/L = -0.17$) passing through the streamwise vortex, for different ground boundary conditions.
5.4. RESULTS

5.4.3 Flow near Rough Ground Surface

Figure 5.19 shows detail of the instantaneous flow near the rough ground surface. An $x - y$ plane clipped to the height of the boundary layer is depicted in Fig. 5.19a, with velocity vectors constrained to the 2-dimensional plane. The vectors show that some recirculation is being captured near the surface, however, the grid size increases rapidly with distance from the wall as can be seen in Fig. 5.19b. This implies that this region may not be as well resolved as other regions in the flow. The vorticity contours in Fig. 5.19c show much coarser structures than those observed in Fig. 5.18 in the vortex region. The limitations of the grid near the ground region are understood.

5.4.4 Mean Streamwise Velocity

Figure 5.20 shows a series of streamwise slices in the domain of the vortex generator from $0.5l$ to $1.6l$, with contours of streamwise velocity. Length $l = L\cos\theta$, the effective length of the vortex generator in the streamwise direction. In the figures, the term ‘fixed’ refers to the smooth stationary ground, ‘moving’ to the smooth moving ground and ‘rough’ to the rough stationary ground. A vortex is observed to form near the edge of the vortex generator, similar to that observed on the delta wing in Fig. 4.18. The vortex grows in size as the slices progress downstream on the surface of the vortex generator. Although velocity in the core decreases, vortex breakdown is not observed on the vortex generator. A smaller, secondary vortex is observed further outboard, which coalesces with the primary vortex in the wake of the vortex generator.

The difference in the size of the boundary layers on the smooth and rough ground surfaces is also noted. The height of the boundary layer over the rough ground is a factor of 2 larger than that over the smooth stationary ground. The moving ground was effective in eliminating the boundary layer.
5.4. RESULTS

Figure 5.19: Contours of instantaneous streamwise velocity and instantaneous vorticity magnitude near the rough ground surface showing velocity vectors and grid.
5.4. RESULTS

Figure 5.20: Contours of mean streamwise velocity ($u/U_\infty$) at $x = 0.5l - 0.8l$. 

167
5.4. RESULTS

\( u/U_\infty = 0.9 \)

\( u/U_\infty = 1 \)

\( u/U_\infty = 1.1 \)

\( u/U_\infty = 1.2 \)

Figure 5.20: Contours of mean streamwise velocity \( (u/U_\infty) \) at \( x = 0.9l - 1.2l \).
Figure 5.20: Contours of mean streamwise velocity ($u/U_\infty$) at $x = 1.3l - 1.6l$. 

5.4. RESULTS
5.4. RESULTS

Figure 5.20: Contours of mean streamwise velocity \((u/U_\infty)\) at \(x = 2l - 5l\).
5.4. RESULTS

5.4.5 Mean Streamwise Vorticity

Figure 5.21 shows contours of streamwise vorticity \( \Omega_x = \frac{\partial W}{\partial y} + \frac{\partial V}{\partial z} \) at the same positions as those in Fig. 5.20. Negative vorticity is observed in the position of the primary vortex. Similar to what was observed in Chapter 4, the negative vorticity originates due to an instability at the edge of the vortex generator. Negative vorticity is generated in the separated shear layer at the edge which feeds into the primary vortex, generating a large region of negative vorticity. The flow from the primary vortex reattaches to the surface of the vortex generator and separates due to the large pressure gradient between that in the position of the vortex and that at the edge. The secondary vortex is rotating in the opposite direction of that of the primary vortex and hence contains positive streamwise vorticity.

As the slices progress downstream, off the surface of the vortex generator, the secondary vortex is observed to first follow its own path with a different sign of axial vorticity, but then coalesce with the primary vortex. At 1.6l, only negative vorticity can be observed on the plane. The primary vortex is relatively larger than the secondary vortex and causes the secondary vortex to be wound into it. Similarly a vortex generator of this size can ‘collect’ and change direction of significantly smaller vortices that are generated from small winglets and other vortex generating devices ahead of it and coalesce them into a single steady vortex that will not break down if the correct geometry and angle of vortex generator are chosen.

When comparing the smooth stationary and moving ground contours, in the first few slices there is no axial vorticity on the ground plane region since axial vorticity is based on gradients of vertical and spanwise velocity, and the flow is travelling in the streamwise direction. At this point, the vortex which is creating velocity components in the latter two directions is not large enough or close enough to have an effect on the ground plane. However, when comparing these to the rough ground, small regions of
5.4. RESULTS

Figure 5.21: Contours of mean streamwise vorticity ($\Omega_x L/\infty$) at $x = 0.5l - 0.8l$. 172
5.4. RESULTS

Figure 5.21: Contours of mean streamwise vorticity ($\Omega_x L/U_\infty$) at $x = 0.9l - 1.2l$. 
5.4. RESULTS

Figure 5.21: Contours of mean streamwise vorticity $\left(\Omega_x L/U_\infty\right)$ at $x = 1.3l - 1.6l$. 

174
5.4. RESULTS

Figure 5.21: Contours of mean streamwise vorticity ($\Omega_x L/U_\infty$) at $x = 2l - 5l$. 

175
positive axial vorticity are observed on the rough ground. These regions are observed in the region vertically beneath the primary vortex, and hence it is confirmed that these regions of vorticity are arising due to the interaction of the vortex with the ground, which may have an effect on the strength of the primary vortex. Analysis of circulation in the next section will be used to confirm this.

5.4.6 Circulation

One method of measuring the effect of the rough ground on the vortex is to measure the circulation of the vortex and study the decay or otherwise as the vortex progresses downstream. The circulation around a closed contour is equal to the surface integral of vorticity:

\[ \Gamma_C = \oint_C u \, dl = \iint_S \omega \, dS \]  \hspace{1cm} (5.1)

where \( S \) is an arbitrary surface bounded by \( C \).

The surfaces in Fig. 5.22 were created by identifying the centre of the vortex and specifying a radius around it so that circulation could be calculated on each plane downstream of the vortex generator. It had to be ensured that the plane was capturing all of the circulation of the vortex but that vorticity from the boundary layer was not being included in the calculation. This was done by repeating the calculation for a range of different radii and identifying the size that contains circulation only of the vortex. Figure 5.23 shows plots of circulation for each of the different ground boundary conditions. The larger radii which included boundary layer vorticity gave significantly higher values of circulation. Once the critical radius was reached, circulation continues to decrease gradually as the integration plane gets smaller since less vorticity is being captured.
Figure 5.22: Planes showing vorticity magnitude used to calculate circulation for the three different ground boundary conditions.
5.4. RESULTS

To retain consistency across the different ground boundary conditions, the plane that had the largest difference in circulation from the next larger radius up was selected to be the best radius which was sufficiently large enough to capture all the circulation in the vortex, but small enough to clip the vorticity from the boundary layer. Using this criterion, the 17\text{mm} radius was selected for the smooth stationary and moving grounds, whilst the 16\text{mm} radius was selected for the rough ground.

The plots in Fig 5.24 show very similar circulation values for the three different boundary conditions which makes it difficult to make definite conclusions. The plots suggest the the vortex above the moving ground decays at a slower rate than that over the stationary smooth ground as it retains higher values of circulation at the downstream positions. This is due to the fact that the vortex is not interacting with the boundary layer from the ground.

The same trend cannot be assumed for a moving rough ground if it had to be simulated, eliminating the boundary layer. This is because although the vortex will not be interacting with the boundary layer generated on the ground, the vortex itself will interact with the cavities in the ground. The stationary rough ground is not able to show this interaction due to the large boundary layer that grows over the surface.

Due to the vortex being so close to the top and bottom boundaries, it was difficult to ensure that the plane on which the integration was carried out did not contain boundary layer vorticity. This method of assessment is only meant to give an initial indication and does not draw definite conclusions. In the case of the rough ground, since a moving ground was not simulated, the boundary layer was significantly larger than that of the smooth rough ground which made it even more difficult to separate the vorticity from the primary vortex from the vorticity of the boundary layer. This means that the circulation values of the rough ground may be contaminated with vorticity from the rough boundary, which makes the results unreliable.
5.4 RESULTS

Figure 5.23: Plots of circulation calculated at each non-dimensional position $x/l$ on planes of different radii for the different boundary conditions.
5.4. RESULTS

Figure 5.24: Circulation at each non-dimensional position $x/l$ for the optimum radius for each boundary condition.

5.4.7 Maximum Vorticity Magnitude

Another way by which the strength of the primary vortex could be assessed is by measuring the maximum vorticity magnitude in the core at each streamwise plane. Figure 5.25 shows plots of this from $1l$ to $6l$ at 0.5$l$ intervals for the three different boundary conditions. When comparing the smooth boundary conditions, the moving ground case retains higher values of vorticity magnitude in the primary vortex core, especially between $1l$ and $2.5l$. This is the region where the secondary vortex merges with the primary vortex. The rough ground simulation has the highest maximum vorticity at most streamwise positions.

As previously noted, the primary and secondary vortex are rotating in opposite directions, hence generating opposite signs of streamwise vorticity. When the secondary vortex merges with the primary vortex, the net amount of vorticity in the latter is reduced, hence reducing the total vorticity magnitude in the core. In Fig. 5.21, at the $1.5l$, the vortices have already merged in the case of the stationary smooth and rough grounds, whereas traces of two separate vortices can still be observed in the stationary moving ground case. For this reason, the maximum vorticity in the moving
5.4. **RESULTS**

ground case is higher than that observed in the other two cases.

From Fig. 5.15a and Fig. 5.15c it is also evident that in this region, the vorticity from the stationary ground, both smooth and rough, may be hindering the progression of the secondary vortex, forcing it to coalesce earlier than if the ground was moving.

Downstream of the 3/l position, the vorticity magnitude in the core is very similar for all three cases and it becomes difficult to separate the different mechanisms causing changes in the vortex progression. In this region, a boundary layer has started to form even in the moving ground case which makes it similar to the stationary case. In the case of the rough ground, it is not possible to separate the contribution of the vorticity due to the ground being stationary and the interaction of the vorticity of the primary vortex with the cavities in the ground. It is believed that the interaction of the vortex with the cavities will reduce the net vorticity in the core causing it to decay quicker than that over a smooth ground. A moving rough ground simulation would be required to give a quantitative analysis of this.

However, a qualitative analysis can still be made to support the conclusions of the findings. On investigating the series of slices of streamwise vorticity in Fig. 5.21, a small layer of positive vorticity is present near the ground for the smooth stationary and moving ground, whilst small structures of positive vorticity are present near the cavities of the rough ground. This vorticity is a result of the interaction of the rotating flow with the ground rather than vorticity being generated from the boundary layer itself. This is supported by the fact that the smooth stationary and moving ground show nearly identical contours of streamwise vorticity near the ground. As the primary vortex progresses downstream and grows in size, there is more interaction with the ground and the layer of positive vorticity becomes more apparent. The structures present near the rough ground prove that the cavities in the tarmac have some effect on the underbody flow, however a rough moving ground would have to be modelled.
5.5 Closure

This study has given insight into the flow structures generated by vortex generators in the underbody region. Comparisons between three different ground conditions have been given and the interaction of the vortex with the grounds has been shown.

In the case of the smooth ground, plots of vorticity magnitude and circulation have shown that having a moving ground results in the vortex retaining higher values of vorticity magnitude in the core immediately downstream of the vortex generator. Circulation values show similar results, with the vortex in the moving ground decaying slower than over a stationary ground.

It was difficult to quantify the effect of the rough ground on the decay of the vortex due to turbulent boundary layer that forms over the surface. A moving rough ground case would be required to quantify this effect. Although the rough ground shows higher values of circulation, this is thought to be due to the boundary layer merging...
5.5. **CLOSURE**

with the vortex, and hence it was not possible to calculate circulation only for the vortex.

A moving ground condition was easily applied to a smooth ground by specifying a velocity at the boundary. However, it is not as easily applied to a rough surface. One method which is being proposed in order to model the relative movement of the vortex generator with the ground is an immersed mesh, where the mesh is wrapped around the vortex generator and moved through the fixed ground and air.
Chapter 6

Conclusion
6.1. ACCOMPLISHMENTS AND FINDINGS

Large Eddy Simulation (LES) has been successfully carried out at experimental Reynolds numbers for a number of automotive flow applications. Unsteady turbulent structures around a diffuser-equipped bluff body, a delta wing and a vortex generator have been computed and presented in this study, giving the reader further insight into the instantaneous and mean flow. Tetrahedral, hexahedral and hybrid grids have been used, all showing comparable results to previous experiments and computational studies. The culmination of the project was the modelling of a rough ground, which has shown promising results.

### 6.1 Accomplishments and Findings

Unsteady structures in the diffuser region at a Reynolds number of $1.01 \times 10^6$ have been investigated for the first time using LES, providing a good understanding of the flow. Comparisons of time-averaged and instantaneous flow on a 20 million node hexahedral grid revealed unsteady vortex structures in the wake of the flow which have little resemblance to the mean flow. However, results in the diffuser region showed that although the core of the streamwise vortices moves around in the instantaneous flow, similar structures are observed in the mean flow, since separation is strongly defined by the end plates of the diffuser. The small computational time-step inherent in LES means that longer time-averaging is required to achieve a more accurate mean in the wake of the flow where the time scales are large.

The main weakness in the simulation is the late laminar to turbulence transition in the underbody region. Turbulent structures were resolved around the sides and top of the bluff body, but not in the underbody region. It is thought that the effect of the ground hindered these structures from being resolved, possibly because the structures present in the tight underbody region are smaller than those resolved around the
6.1. ACCOMPLISHMENTS AND FINDINGS

sides and back of the body. Testing with a numerical trip similar to that used in experimental work did not succeed in forcing an early turbulent boundary simulation.

Large Eddy Simulation of a delta wing was also successfully carried out at experimental Reynolds numbers \((1.56 \times 10^6)\) and served as a reliable validation study for the LES method. This is the first LES delta wing study to present such extensive results which are in very good agreement with experiments.

A high level of agreement was achieved between the LES and experiments, mainly due to the high level of grid refinement that was achieved by the 15 million node tetrahedral grid in the vortex regions by means of geometric sources. Detail of the leading edge and secondary vortices was given by means of velocity, vorticity and turbulent kinetic energy (TKE) contours as well as surface pressure measurements. Substructures around the vortex core were also captured in the instantaneous and mean flow, confirming that they are spatially stationary. The substructures originate at the leading edge and are entrained into the main vortex, resulting in a braiding around the leading edge vortex. The braiding appears to be finer than that captured by previous DES studies since more of the eddies are being resolved rather than modelled. When cutting through the flow at perpendicular planes to the wing, the braiding appears as an arrangement of circular substructures around the leading edge vortex.

The main weakness in the delta wing calculation was the discrepancy in surface pressure along the span of the wing. It has previously been suggested that this may be due to a scaling error in the experiments since all other comparisons are in very close agreement. Pressure distribution for the coarse grid (8 million nodes) compared reasonably well with previous DES studies, however, the finer grid showed increased detail and was able to capture the suction at the position of the secondary vortex. Although surface pressures from the experiments and DES studies do not capture the
secondary vortex, a secondary vortex is present in their flow visualisation, indicating that there are not enough pressure tappings near the leading edge to capture this in the experiments.

The position of the sudden increase in Turbulent Kinetic Energy (TKE) at the breakdown position was captured accurately, but values were a factor of 2.4 higher than those quoted in the experiments and DES studies. It is unclear whether the correct values for maximum TKE have been quoted by previous studies since the DES studies have quoted the highest value shown on the experiments TKE contour scale as being the maximum TKE. Although they were able to match this in their computations, this study has shown that further grid refinement could have predicted the flow better. The LES contours of TKE in the vortex core compare very well with the experiments when using the same scale, another indication that the scale in the experiments may have been capped at a value of 0.5, but may have achieved higher values.

The results from the delta wing study were able to validate the methodology and provided evidence of LES being able to capture such flows accurately. This strengthens the findings of the ‘Vortex Generator in Ground Effect’ for which experiments are not available. LES for the vortex generator case was carried out at a Reynolds number of \(2 \times 10^5\), reducing the computational effort required.

For the first time, detailed flow visualisation of the flow around a vortex generator has been presented. Insight into the flow structures generated by vortex generators in the underbody region, as well as comparisons between a stationary and moving smooth ground, and a stationary rough ground was presented. Due to the onset angle of the vortex generator, a vortex forms along the tip and extends along the length of the domain. At an angle of 20 degrees, vortex breakdown did not occur on the surface, unlike the delta wing, since the angle is lower, and pressure gradients along the core
are less adverse. Instead, a vortex is sustained throughout the domain. This provides a means of managing the flow by supplying the rear underbody with attached flow. In cases where underbody rear diffusers are used in motorsport, a strongly attached flow allows it to operate at higher angles and lower ride-heights, generally leading to increased downforce.

The smooth ground domain consisted of a 40 million node hexahedral grid, whilst the rough ground case required a hybrid grid which resulted in 48 million nodes. In the case of the smooth ground, plots of vorticity magnitude and circulation have shown that having a moving ground results in the vortex retaining higher values of vorticity magnitude in the core immediately downstream of the vortex generator. Circulation values show similar results, with the vortex in the moving ground decaying slower than over a stationary ground. It is difficult to translate this directly to the rough ground simulation, since it is not possible to isolate the effect of the large boundary layer from the effect of the interaction of the vortex with the cavities in the ground. For this reason, it was difficult to quantify the effect of the rough ground on the decay of the vortex due to the turbulent boundary layer that forms over the surface.

A moving rough ground case would be required to quantify this effect. Although the rough ground shows higher values of circulation, this is thought to be due to the boundary layer merging with the vortex, and hence it was not possible to isolate the vortex completely when performing calculations for maximum vorticity in the core and circulation of the vortex. Nevertheless, this study has been show that the modelling of a rough ground in LES is possible.
6.2 Further Work

This study has been effective in investigating and giving insight into unsteady flow features that have not been explained in such detail before. However, there are a few suggestions for future work which are thought could improve the LES.

In the diffuser-equipped bluff body case, the main issue seemed to be the inability of the LES to predict laminar to turbulent transition in the underbody region. Although it was captured over the top surface due to shear layer separation, in the underbody it was unable to capture the transition along the wall because of the high levels of subgrid scale viscosity predicted by the Smagorinsky model. The Smagorisky model might not be able to capture this accurately even with a fine enough grid. The Wall Adapting Local Eddy-viscosity (WALE) [116] model is thought to improve the laminar to turbulent transition since it does not produce spurious high levels of subgrid scale viscosity in a steady laminar shear layer. This might lead to a better representation of the flow without having to refine the grid much further.

Refinement of the delta wing grid in the substructures region may stop the structures from becoming weaker as the vortex progresses downstream. Confirmation of pressure coefficient and Turbulent Kinetic Energy values from the experiments and previous DES studies would tie up the results. Private communication [117] indicates that improved DES methods that are now available might yield similar results to the LES results in this study.

For the vortex generator case over a rough ground, a moving ground is required in order to draw quantitative conclusions on the decay of the vortex when compared to the smooth ground case. That would allow a direct comparison of a moving smooth and rough ground. One method which is being proposed in order to model the relative movement of the vortex generator with the ground is an immersed mesh, where the
6.2. FURTHER WORK

mesh is wrapped around the vortex generator and moved through the fixed ground and air. This would eliminate the turbulent boundary layer that forms on the rough ground and enable direct measurement of vortex decay due to its interaction with the cavities in the ground.
References


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


