DEVELOPMENT, IMPLEMENTATION AND TESTING OF AN ALTERNATIVE DDES FORMULATION BASED ON ELLIPTIC RELAXATION

A THESIS SUBMITTED TO THE UNIVERSITY OF MANCHESTER FOR THE DEGREE OF DOCTOR OF PHILOSOPHY IN THE FACULTY OF ENGINEERING AND PHYSICAL SCIENCES

2012

By
Neil Ashton
School of Mechanical, Aerospace & Civil Engineering
# Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>14</td>
</tr>
<tr>
<td>Declaration</td>
<td>15</td>
</tr>
<tr>
<td>Copyright</td>
<td>16</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>17</td>
</tr>
<tr>
<td>1 Introduction</td>
<td>26</td>
</tr>
<tr>
<td>1.1 Background</td>
<td>26</td>
</tr>
<tr>
<td>1.2 Study Objectives</td>
<td>28</td>
</tr>
<tr>
<td>1.3 Outline of thesis</td>
<td>29</td>
</tr>
<tr>
<td>2 Turbulence Modelling</td>
<td>31</td>
</tr>
<tr>
<td>2.1 Reynolds Averaged Navier-Stokes (RANS)</td>
<td>32</td>
</tr>
<tr>
<td>2.1.1 Algebraic models</td>
<td>34</td>
</tr>
<tr>
<td>2.1.2 One-equation models</td>
<td>36</td>
</tr>
<tr>
<td>2.1.3 Two-equation models</td>
<td>39</td>
</tr>
<tr>
<td>2.1.4 Reynolds Stress Models</td>
<td>45</td>
</tr>
<tr>
<td>2.1.5 Near-wall modelling</td>
<td>47</td>
</tr>
<tr>
<td>2.2 Large Eddy Simulation (LES)</td>
<td>60</td>
</tr>
<tr>
<td>2.3 Hybrid RANS-LES</td>
<td>63</td>
</tr>
<tr>
<td>2.4 Closure</td>
<td>78</td>
</tr>
<tr>
<td>3 Numerical solver</td>
<td>79</td>
</tr>
<tr>
<td>3.1 Introduction</td>
<td>79</td>
</tr>
<tr>
<td>3.2 Finite volume method</td>
<td>79</td>
</tr>
<tr>
<td>3.3 Spatial Discretisation</td>
<td>80</td>
</tr>
<tr>
<td>3.3.1 Convection term</td>
<td>81</td>
</tr>
</tbody>
</table>
3.3.2 Diffusion term .............................................. 85
3.3.3 Gradient reconstruction ...................................... 85
3.4 Boundary conditions ........................................... 86
3.5 Turbulence models .............................................. 87
  3.5.1 RANS models ............................................. 87
  3.5.2 DDES models ............................................. 88
  3.5.3 Validation - Fully developed channel flow at Re=395 ... 90
3.6 Closure .......................................................... 91

4 Development of the $\phi - f$ DDES model ....................... 94
  4.1 Introduction .................................................. 94
  4.2 The $\phi - f$ DDES model ..................................... 96
    4.2.1 Derivation ................................................ 97
    4.2.2 Calibration ............................................... 99
    4.2.3 Numerical scheme sensitivity study ...................... 101
    4.2.4 Computational expense .................................. 103
  4.3 Closure ........................................................ 104

5 2D Periodic hills .................................................. 106
  5.1 Introduction .................................................. 106
  5.2 Computational grid and boundary conditions .................. 107
  5.3 Results ........................................................ 108
    5.3.1 Re=10590 ................................................. 108
    5.3.2 Re=37000 ................................................. 113

6 NACA0021 at 60° incidence ......................................... 131
  6.1 Introduction .................................................. 131
  6.2 Computational grid and boundary conditions .................. 132
  6.3 Results ........................................................ 133
    6.3.1 Mesh resolution .......................................... 133
    6.3.2 DDES results ............................................ 133

7 2D wall-mounted hump ............................................... 139
  7.1 Introduction .................................................. 139
  7.2 Computational grid and boundary conditions .................. 140
  7.3 Results ........................................................ 141
    7.3.1 Mesh and reference data analysis ....................... 141
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.3.2 DDES Results</td>
<td>142</td>
</tr>
<tr>
<td>8 Ahmed car body</td>
<td>159</td>
</tr>
<tr>
<td>8.1 Introduction</td>
<td>159</td>
</tr>
<tr>
<td>8.2 Computational grid and boundary conditions</td>
<td>161</td>
</tr>
<tr>
<td>8.3 Results</td>
<td>162</td>
</tr>
<tr>
<td>8.3.1 Mesh resolution</td>
<td>162</td>
</tr>
<tr>
<td>8.3.2 DDES results</td>
<td>163</td>
</tr>
<tr>
<td>9 Conclusions and suggestions for further work</td>
<td>186</td>
</tr>
<tr>
<td>9.1 Conclusions</td>
<td>186</td>
</tr>
<tr>
<td>9.2 Suggestions for Future work</td>
<td>189</td>
</tr>
</tbody>
</table>
List of Tables

2.1 Model coefficients for the SA model . . . . . . . . . . . . . . . . . 38
2.2 Model coefficients for the $k - \omega$ SST model . . . . . . . . . . . 43
2.3 Model coefficients for the elliptic relaxation RSM model . . . . . . 53
2.4 Model coefficients for the EBRSM model . . . . . . . . . . . . . . 55
2.5 Further model coefficients for the EBRSM model . . . . . . . . . . 56
2.6 Model coefficients for the $\varphi - f$ model . . . . . . . . . . . . . 59
4.1 Summary of convective schemes applied to the momentum equations 102
4.2 Average time per iteration (in seconds) for each test case using the three DDES formulations . . . . . . . . . . . . . . . . . . . . . . . . 104
5.1 Reattachment points for the 2D periodic hill ($x/h$) . . . . . . . . . 116
5.2 Reattachment points for the 2D periodic hill ($x/h$) . . . . . . . . . 116
6.1 Lift and Drag coefficient results from DDES simulations and expe-

mental data for the NACA0021 airfoil . . . . . . . . . . . . . . . . . . . 136
## List of Figures

3.1 General configuration of a face between two adjacent cells .......................... 81
3.2 Notations for spatial discretisation ......................................................... 82
3.3 Energy spectra versus wavenumber ($\kappa$) for the velocity field, using varying values of $C_{DDES}$ for the $32^3$ and $64^3$ grids SA - DDES ........ 89
3.4 Energy spectra versus wavenumber ($\kappa$) for the velocity field, using varying values of $C_{DDES}$ for the $32^3$ and $64^3$ grids SST - DDES .... 90
3.5 (a) Mean stream-wise velocity and (b) Mean turbulent kinetic energy for fully developed channel flow at $Re=395$ .............................. 91
3.6 (a) Mean turbulent dissipation and (b) Mean wall-normal Reynolds Stress component for fully developed channel flow at $Re=395$ .... 92
3.7 (a) Mean turbulent viscosity and (b) mean $\varphi$ function for fully developed channel flow at $Re=395$ .................................................. 93
4.1 Demonstration of the functionality of the $\Psi$ correction term for the DIT case using a $64^3$ grid at $t=2\, s$ and $C_{DDES}=0.60$ ................. 99
4.2 Energy spectra versus wavenumber ($\kappa$) for the velocity field, using varying values of $C_{DDES}$ for the $32^3$ and $64^3$ grids (a) SST DDES, (b) $\varphi-f$ DDES .................................................. 101
4.3 Energy spectra versus wavenumber ($\kappa$) for the velocity field, with different numerical schemes using the $64^3$ grid (a) SST DDES, (b) $\varphi-f$ DDES .................................................. 104
4.4 Mean span-wise averaged skin friction coefficient ($C_f$) using different numerical schemes for the 2D-wall mounted hump (a) SST DDES, (b) $\varphi-f$ DDES .................................................. 105
4.5 Mean span-wise averaged skin friction coefficient ($C_f$) using different numerical schemes for the 2D periodic hills for the (a) SST DDES, (b) $\varphi-f$ DDES .................................................. 105
5.1 Case setup for the 2D periodic hills .................................................. 107
5.2 Mean skin friction coefficient ($C_f$) for the SST, $\varphi - f$ and Spalart-Allmaras RANS models on the 2D periodic hills for
(a) $Re = 10590$ .................................................................................. 116
and (b) $Re = 37000$ .............................................................................. 116
5.3 Wall-adjacent cell size along the bottom wall of the 2D periodic
hills at $Re=10590$ .................................................................................. 117
5.4 Ratio of filter width to Kolmorogov scale: $\Delta/\eta$ at
(a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic
hills at $Re=10590$ .................................................................................. 117
5.5 Mean turbulent viscosity ratio: $\nu_t/\nu$ at
(a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic
hills at $Re=10590$ using the reference LES and experimental data 118
5.6 Mean span-wise averaged stream-wise velocity profiles at
(a) $x/h = 0.05$ (b) $x/h = 2.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic
hills at $Re=10590$ using the reference LES and experimental data 118
5.7 Mean span-wise averaged turbulent shear stress profiles at
(a) $x/h = 0.05$ (b) $x/h = 2.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic
hills at $Re=10590$ using the reference LES and experimental data 119
5.8 Mean span-wise averaged skin friction coefficient ($C_f$) for the reference LES and experimental data on the 2D periodic hills at $Re = 10590$ .................................................................................. 119
5.9 Mean span-wise averaged skin friction coefficient ($C_f$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D periodic
hills at $Re = 10590$ .................................................................................. 120
5.10 Mean span-wise averaged pressure coefficient ($C_p$) for the SST-
DDES, $\varphi - f$ DDES and SA-DDES models on the 2D periodic
hills at $Re = 10590$ .................................................................................. 121
5.11 The location of the DDES zones (0=RANS 1=LES) and the blending
function $f_d$ for the SST DDES model ...................................................... 121
5.12 Instantaneous velocity shots using the $\varphi - f$ DDES model which
highlights the alternating separation and reattachment across the
flow at $Re=10590$ .................................................................................. 122
5.13 Mean streamlines for the SST DDES model for the 2D periodic
hills at $Re=10590$ .................................................................................. 122
5.14 Mean streamlines for the $\varphi - f$ DDES model for the 2D periodic
hills at $Re=10590$ .................................................................................. 122
5.15 Mean streamlines for the SA DDES model for the 2D periodic hills at Re=10590

5.16 Mean span-wise averaged stream-wise velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.17 Mean span-wise averaged transverse velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.18 Mean span-wise averaged turbulent shear stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.19 Mean span-wise averaged turbulent $u'u'$ Reynolds’s stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.20 Mean span-wise averaged turbulent $v'v'$ Reynolds’s stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.21 Mean span-wise averaged turbulent $w'w'$ Reynolds’s stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

5.22 Wall-adjacent cell size along the bottom wall of the 2D periodic hills at Re=37000

5.23 Ratio of filter width to Kolmorogov scale: $\Delta/\eta$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=37000

5.24 Mean turbulent viscosity ratio: $\nu_t/\nu$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=37000

5.25 Mean span-wise averaged skin friction coefficient ($C_f$) for the SST-DDES, $\varphi-f$ DDES and SA-DDES models on the 2D periodic hills at Re=37000

5.26 Mean span-wise averaged stream-wise velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=37000

8
5.27 Mean span-wise averaged transverse velocity profiles at (a) \( x/h = 0.05 \) (b) \( x/h = 3.0 \) (c) \( x/h = 6.0 \) (d) \( x/h = 8.0 \) for the 2D periodic hills at Re=37000.

5.28 Mean span-wise averaged turbulent shear stress profiles at (a) \( x/h = 0.05 \) (b) \( x/h = 3.0 \) (c) \( x/h = 6.0 \) (d) \( x/h = 8.0 \) for the 2D periodic hills at Re=37000.

5.29 Mean span-wise averaged turbulent \( \overline{u'w'} \) Reynold’s stress profiles at (a) \( x/h = 0.05 \) (b) \( x/h = 3.0 \) (c) \( x/h = 6.0 \) (d) \( x/h = 8.0 \) for the 2D periodic hills at Re=37000.

5.30 Mean span-wise averaged turbulent \( \overline{v'v'} \) Reynold’s stress profiles at (a) \( x/h = 0.05 \) (b) \( x/h = 3.0 \) (c) \( x/h = 6.0 \) (d) \( x/h = 8.0 \) for the 2D periodic hills at Re=37000.

5.31 Mean span-wise averaged turbulent \( w'w' \) Reynold’s stress profiles at (a) \( x/h = 0.05 \) (b) \( x/h = 3.0 \) (c) \( x/h = 6.0 \) (d) \( x/h = 8.0 \) for the 2D periodic hills at Re=37000.

6.1 Mesh and domain for the NACA0021 airfoil.

6.2 Running time-average of the span-averaged (a) lift coefficient and (b) drag coefficient by the SST DDES, \( \varphi - f \) DDES and SA DDES models for the NACA 0021 airfoil.

6.3 Power Spectral Density (PSD) of the lift coefficient for the (a) SST DDES model, (b) \( \varphi - f \) DDES model and (c) the SA DDES model for the NACA 0021 airfoil.

6.4 Mean span-wise averaged (a) pressure coefficient \( (C_p) \) and (b) skin-friction coefficient by the SST DDES, \( \varphi - f \) DDES and SA DDES models for the NACA 0021 airfoil. The upper line represents the lower surface of the airfoil for the skin-friction coefficient and vice versa for the lower line.

6.5 Mean resolved turbulent kinetic energy (TKE) for the (a) SST DDES model, (b) \( \varphi - f \) DDES model and (c) SA DDES model for the NACA0021 airfoil.

6.6 Mean modelled turbulent kinetic energy (TKE) for the (a) SST DDES model, (b) \( \varphi - f \) DDES model and the (c) SA DDES model for the NACA0021 airfoil.

6.7 Mean velocity streamlines for the (a) SST DDES model, (b) \( \varphi - f \) DDES model and the (c) SA DDES model for the NACA0021 airfoil.
6.8 Instantaneous field of Vorticity for the (a) SST DDES model, (b) \( \varphi - f \) DDES model and the (c) SA DDES model for the NACA0021 airfoil

7.1 (a) Case setup for the 2D wall-mounted hump (b) Mesh for the 2D wall-mounted hump

7.2 Mesh and domain for the 2D wall-mounted hump

7.3 Mean span-wise averaged skin-friction coefficient for both 2D wall-mounted hump meshes using the \( \varphi - f \) DDES model

7.4 Wall-adjacent cell size along the bottom wall of the 2D hump

7.5 Ratio of filter width to Kolmogorov scale: \( \Delta / \eta \) at (a) \( x/c = 0.65 \) (b) \( x/c = 0.66 \) (c) \( x/c = 0.80 \) (d) \( x/c = 0.9 \) (e) \( x/c = 1.0 \) (f) \( x/c = 1.1 \) (g) \( x/c = 1.2 \) (h) \( x/c = 1.3 \) for the 2D wall-mounted hump

7.6 Mean turbulent viscosity ratio: \( \nu_t / \nu \) at (a) \( x/c = 0.65 \) (b) \( x/c = 0.66 \) (c) \( x/c = 0.80 \) (d) \( x/c = 0.9 \) (e) \( x/c = 1.0 \) (f) \( x/c = 1.1 \) (g)

7.7 Mean span-wise averaged skin friction coefficient \( (C_f) \) (a) and pressure coefficient \( (C_p) \) (b) for the SST-DDES, \( \varphi - f \) DDES and SA-DDES models on the 2D hump

7.8 Mean span-wise averaged skin friction coefficient \( (C_f) \) for the SST-DDES, \( \varphi - f \) DDES and SA-DDES models on the 2D hump

7.9 Mean span-wise averaged pressure coefficient \( (C_p) \) for the SST-DDES, \( \varphi - f \) DDES and SA-DDES models on the 2D hump

7.10 Mean streamlines for the SST DDES model (top) and the SST DDES model (bottom) for the 2D wall-mounted hump

7.11 Mean streamlines for the \( \varphi - f \) DDES model for the 2D wall-mounted hump

7.12 Mean streamlines for the SA DDES model for the 2D wall-mounted hump

7.13 Iso-Q contours coloured by velocity magnitude for the SST DDES model for the 2D wall-mounted hump

7.14 Iso-Q contours coloured by velocity magnitude for the \( \varphi - f \) DDES model for the 2D wall-mounted hump

7.15 Iso-Q contours coloured by velocity magnitude for the SA DDES model for the 2D wall-mounted hump
7.16 $f_d$ function for the $\varphi - f$ DDES model on the 2D wall-mounted hump ................................................................. 154
7.17 Mean span-wise averaged stream-wise velocity profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.9$ for the 2D wall-mounted hump ................................................................. 154
7.18 Mean span-wise averaged transverse velocity profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.9$ for the 2D wall-mounted hump ................................................................. 155
7.19 Mean span-wise averaged turbulent shear stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump ................................................................. 155
7.20 Mean span-wise averaged turbulent $\overline{u' u'}$ Reynolds’s stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump ................................................................. 156
7.21 Mean span-wise averaged turbulent $\overline{v' v'}$ Reynolds’s stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump ................................................................. 156
7.22 Mean span-wise averaged stream-wise velocity profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump ................................................................. 157
7.23 Mean span-wise averaged transverse velocity profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump ................................................................. 157
7.24 Mean span-wise averaged turbulent shear stress profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump ................................................................. 157
7.25 Mean span-wise averaged turbulent $\overline{u' u'}$ Reynolds’s stress profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump ................................................................. 158
7.26 Mean span-wise averaged turbulent $\overline{v' v'}$ Reynolds’s stress profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump ................................................................. 158
8.1 Dimensions of the Ahmed car body .......................... 168
8.2 Coarse mesh for the Ahmed car body ......................... 168
8.3 Fine mesh for the Ahmed car body ............................ 168
8.4 Mean streamlines over the Ahmed car body from experimental results [3] .................................................. 169
8.5 Mean streamlines over the Ahmed car body for the SST DDES model .................................................. 169
8.6 Mean streamlines over the Ahmed car body for the $\varphi - f$ DDES model ............................................. 170
8.7 Mean streamlines over the Ahmed car body for the SA DDES model ................................................... 170
8.8 Iso-surfaces of the Q-criterion over the Ahmed car body for the SST DDES model, coloured by mean stream-wise velocity .......................................................... 171
8.9 Iso-surfaces of the Q-criterion over the Ahmed car body for the $\varphi - f$ DDES model, coloured by mean stream-wise velocity .......................................................... 171
8.10 Iso-surfaces of the Q-criterion over the Ahmed car body for the SA DDES model, coloured by mean stream-wise velocity ................................................... 172
8.11 Iso-surfaces of the Q-criterion over the Ahmed car body for the SST DDES model, coloured by mean stream-wise velocity ................................................... 172
8.12 Iso-surfaces of the Q-criterion over the Ahmed car body for the $\varphi - f$ DDES model, coloured by mean stream-wise velocity .......................................................... 173
8.13 Iso-surfaces of the Q-criterion over the Ahmed car body for the SA DDES model, coloured by mean stream-wise velocity ................................................... 173
8.15 Iso-contours of mean velocity at $x/H = 0$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body .......................................................... 174
8.16 Iso-contours of mean velocity at $x/H = 0.27$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body .......................................................... 175
8.17 Iso-contours of mean velocity at $x/H = 0.69$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body .......................................................... 176
8.18 Iso-contours of mean velocity at $x/H = 1.74$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body .......................................................... 177
8.19 Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4] .......................................................... 178
<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.20</td>
<td>Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].</td>
</tr>
<tr>
<td>8.21</td>
<td>Mean wall-normal velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].</td>
</tr>
<tr>
<td>8.22</td>
<td>Mean wall-normal velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].</td>
</tr>
<tr>
<td>8.23</td>
<td>Mean turbulent kinetic energy profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].</td>
</tr>
<tr>
<td>8.24</td>
<td>Mean turbulent kinetic energy profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].</td>
</tr>
<tr>
<td>8.25</td>
<td>Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models using the coarse 2.7 million cell mesh. Experimental data from [4].</td>
</tr>
<tr>
<td>8.26</td>
<td>Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models using the coarse 2.7 million cell mesh. Experimental data from [4].</td>
</tr>
</tbody>
</table>
Abstract

University of Manchester
Neil Ashton
Doctor of Philosophy
Development, Implementation and Testing of an Alternative DDES
Formulation Based on Elliptic Relaxation
December 31st, 2012

A new formulation of Delayed Detached-Eddy Simulation (DDES) based upon elliptic relaxation is derived and implemented within a finite-volume framework. This new formulation is based upon the $\varphi - f$ RANS model which has previously demonstrated both improved modelling of the near-wall physics and numerical robustness for industrial applications. The $\varphi - f$ DDES model is calibrated and validated using Decaying Isotropic Turbulence (DIT) to establish the validity of the derivation and to calibrate the model constants. In light of the numerical scheme requirements for DDES, a hybrid numerical scheme is proposed and implemented, which is shown to perform in the intended manner.

Initially, three DDES formulations (SA-DDES, SST-DDES and $\varphi - f$ DDES) are compared on the 2D periodic hills test case at $Re = 10590$ and $Re = 37000$. This test case primarily serves as a validation case to evaluate whether the implementation and calibration were correct. The flow over a NACA0021 airfoil post-stall at $60^\circ$ incidence is then evaluated; a test case that DDES was originally devised for (i.e. massive separation from an airfoil). The three formulations are then evaluated on a 2D wall-mounted hump which exhibits largely geometry induced separation, but is still sensitive to the modelling of the initial separated shear layer and upstream turbulence levels. The final case is the Ahmed car body which combines both geometry and pressure-induced separation from a 3D surface. This complex flow is challenging for any turbulence modelling approach and is sensitive to the underlying RANS model.

A general sensitivity to the underlying RANS model is demonstrated for the majority of the test cases investigated. The $\varphi - f$ DDES model is shown to have encouraging performance on these wide range of test cases compared to the established SST-DDES and SA-DDES models. Whilst the $\varphi - f$ DDES model is not a fix for the shortcomings of DDES, it is shown to be a practical and robust alternative to the established SST-DDES and SA-DDES variants that have become the de facto choice for many DDES users.
Declaration

No portion of the work referred to in this thesis has been submitted in support of an application for another degree or qualification of this or any other university or other institute of learning.
Copyright

i. The author of this thesis (including any appendices and/or schedules to this thesis) owns certain copyright or related rights in it (the “Copyright”) and he has given The University of Manchester certain rights to use such Copyright, including for administrative purposes.

ii. Copies of this thesis, either in full or in extracts and whether in hard or electronic copy, may be made only in accordance with the Copyright, Designs and Patents Act 1988 (as amended) and regulations issued under it or, where appropriate, in accordance with licensing agreements which the University has from time to time. This page must form part of any such copies made.

iii. The ownership of certain Copyright, patents, designs, trade marks and other intellectual property (the “Intellectual Property”) and any reproductions of copyright works in the thesis, for example graphs and tables (“Reproductions”), which may be described in this thesis, may not be owned by the author and may be owned by third parties. Such Intellectual Property and Reproductions cannot and must not be made available for use without the prior written permission of the owner(s) of the relevant Intellectual Property and/or Reproductions.

iv. Further information on the conditions under which disclosure, publication and commercialisation of this thesis, the Copyright and any Intellectual Property and/or Reproductions described in it may take place is available in the University IP Policy (see http://www.campus.manchester.ac.uk/medialibrary/policies/intellectual-property.pdf), in any relevant Thesis restriction declarations deposited in the University Library, The University Library’s regulations (see http://www.manchester.ac.uk/library/aboutus/regulations) and in The University’s policy on presentation of Theses.
Acknowledgements

First and foremost I would like to thank Dr Alistair Revell for his guidance, motivation and understanding. I am forever grateful to the opportunities he gave me, especially becoming involved with the ATAAC project that gave me the much needed motivation and structure that were so influential in my work. I would also like to thank my other supervisor Dr Robert Prosser. Although we may have had differing opinions along the way, I am very grateful for the opportunity he gave me to undertake the PhD, and without this I know I would not have become the person I am today.

The PhD has been much more than just research and I want to say a special thank you to Alastair and Nick who have been alongside me for every bit of this journey. The many games of squash, Wetherspoons breakfasts and lunchtime discussions have truly made my PhD experience something to cherish. I would also like to take this opportunity to thank Dr Juan Uribe, Dr Flavien Billard and Dr Stefano Rolfo who were always there to help when I needed them, their everlasting patience for the many questions I asked them is much appreciated. I would also like to thank the rest of the George Begg office who have been great company, particularly Mark, Dalila, Umair, Imran, Mahmoud, James, Hossein and Mina.

I would like to acknowledge the Engineering and Physical Sciences Research Council for funding my studies as well as the ATAAC project for providing the financial support for the final 6 months of my studies[1]. I would also like to thank Dr Mariano Vazquez, Dr Herbert Owen and Dr Guillaume Houzeaux at the Barcelona Supercomputing Centre (BSC) for hosting me for a very enjoyable month. I would also like to thank the EDF team working on Code_Saturne, in particular Dr Martin Ferrand and Dr Yvan Fournier who were always willing to help me out with my work.

A special thanks goes out to Dr Simon Hood who has had to live with my almost constant-use of the HPC facilities for nearly 3 years. His patience and support is much appreciated. I would also like to thank Dr Charles Moulinec for his help during my time at BSC and also for his company on the football pitch!

Of course, all of this would not have been possible without the truly selfless and forever caring support of my family. My Mum and Dad have always been there, never doubting, always encouraging, and the most supportive parents I could ever wish for. I would also like to thank my Nana for her love and encouragement, I would like to think that this PhD is testament to all the help and care she gave me over my childhood. I also owe a special thank you to my Sister, Laura who has been there for me my entire life, the help she gave to me when I first came to Manchester made a huge difference. Finally I want to say a big thank you to Sabina for her love and support, and for putting up with me being glued to my laptop for the past 4 years! I look forward to our future together.
To Albert Wilson, forever in my thoughts
Nomenclature

Acronyms

ATAAC Advanced Turbulence Simulation for Aerodynamic Application Challenges
BDS Blended Differencing Scheme
BPG Best Practice Guidelines
CDS Central Differencing Scheme
CEASM Compact Explicit Algebraic Stress Model
CFD Computational Fluid Dynamics
CFL Courant-Friedrich-Lewy Condition
CPU Central Processing Unit
DDES Delayed Detached-Eddy Simulation
DES Detached-Eddy Simulation
DESider Detached-Eddy Simulation for Industrial Aerodynamics
DIT Decaying Isotropic Turbulence
DNS Direct Numerical Simulation
DSM Dynamic Smagorinsky Model
EBRSM Elliptic Blending Reynolds Stress Model
EDF Electricite de France
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ERCOFAC</td>
<td>European Research Community on Flow, Turbulence and Combustion</td>
</tr>
<tr>
<td>FLOMANIA</td>
<td>Flow Physics Modelling An Integrated Approach</td>
</tr>
<tr>
<td>GGDH</td>
<td>Generalised Gradient Diffusion Hypothesis</td>
</tr>
<tr>
<td>GIS</td>
<td>Grid-Induced Separation</td>
</tr>
<tr>
<td>HNS</td>
<td>Hybrid Numerical Scheme</td>
</tr>
<tr>
<td>IDDES</td>
<td>Improved Delayed Detached-Eddy Simulation</td>
</tr>
<tr>
<td>LDA</td>
<td>Laser Doppler Anemometry</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LLR</td>
<td>Launder Reece and Rodi</td>
</tr>
<tr>
<td>LLR</td>
<td>Linear Local Realizable</td>
</tr>
<tr>
<td>MDS</td>
<td>Modelled-Stress Depletion</td>
</tr>
<tr>
<td>PANS</td>
<td>Partially Averaged Navier-Stokes</td>
</tr>
<tr>
<td>PITM</td>
<td>Partially Integrated Transport Models</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds Stress Model</td>
</tr>
<tr>
<td>SA</td>
<td>Spalart-Allmaras</td>
</tr>
<tr>
<td>SALSA</td>
<td>Strain-Adaptive Linear Spalart-Allmaras</td>
</tr>
<tr>
<td>SAS</td>
<td>Scale-Adaptive Simulation</td>
</tr>
<tr>
<td>SGS</td>
<td>Sub-grid Scale</td>
</tr>
<tr>
<td>SOLU</td>
<td>Second Order Linear Upwind</td>
</tr>
<tr>
<td>SSG</td>
<td>Speziale Sarkar and Gatski</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
</tbody>
</table>
TKE Turbulent Kinetic Energy
UDS Upwind Differencing Scheme
URANS Unsteady Reynolds-Averaged Navier-Stokes
WMLES Wall-Modelled Large Eddy Simulation
ZDES Zonal Detached-Eddy Simulation

Greek letters
\( \delta_{ij} \) Kronecker symbol
\( \alpha \) Elliptic variable in the elliptic blending approach
\( \Delta x^+ \) Non-dimensional grid spacing in the stream-wise direction
\( \Delta z^+ \) Non-dimensional grid spacing in the span-wise direction
\( \Delta \) LES filter width
\( \delta \) Boundary layer thickness
\( \eta \) Kolmogorov length scale
\( \kappa \) Wavenumber or Von Karman constant
\( \mu \) Molecular viscosity
\( \mu_f \) Friction velocity
\( \nu_{SGS} \) Sub-grid scale viscosity
\( \nu_t \) Turbulent viscosity or eddy viscosity
\( \omega \) Dissipation rate per unit of turbulent kinetic energy
\( \Omega_{ij} \) Rotation rate tensor
\( \phi_{ij} \) Pressure strain-rate correlation tensor
\( \Psi \) SA-DES/DDES correction function
\( \Psi_{\varphi f} \) \( \varphi - f \) DDES correction function
\( \rho \)  \hspace{1em} \text{Density}

\( \sigma_\omega \)  \hspace{1em} \text{Turbulent Prandtl number for the } \omega \text{ transport equation}

\( \sigma_\varepsilon \)  \hspace{1em} \text{Turbulent Prandtl number for the } \varepsilon \text{ transport equation}

\( \sigma_k \)  \hspace{1em} \text{Turbulent Prandtl number for the } k \text{ transport equation}

\( \tau_{ij} \)  \hspace{1em} \text{Viscous stress}

\( \tau_w \)  \hspace{1em} \text{Wall shear stress}

\( \tilde{\nu} \)  \hspace{1em} \text{SA model eddy viscosity}

\( \varepsilon \)  \hspace{1em} \text{Dissipation rate of turbulent kinetic energy}

\( \varepsilon_{ij} \)  \hspace{1em} \text{Tensor of the turbulent dissipation of the Reynolds Stresses}

\( \varphi \)  \hspace{1em} \text{Wall-normal turbulent anisotropy } \varphi = \frac{\nu^2}{k}

**Roman letters**

\( \overline{u_i u_j} \)  \hspace{1em} \text{Reynolds Stress tensor}

\( \overline{u' v'} \)  \hspace{1em} \text{Turbulent shear stress}

\( \overline{v'^2} \)  \hspace{1em} \text{Wall-normal Reynolds Stress}

\( \hat{U} \)  \hspace{1em} \text{Spatially filtered velocity}

\( a_{ij} \)  \hspace{1em} \text{Turbulence anisotropy tensor}

\( c \)  \hspace{1em} \text{Airfoil chord length}

\( C_{\text{DDES}} \)  \hspace{1em} \text{DDES model constant}

\( C_{\text{DES}} \)  \hspace{1em} \text{DES model constant}

\( C_D \)  \hspace{1em} \text{Drag coefficient}

\( C_f \)  \hspace{1em} \text{Skin-friction coefficient}

\( C_L \)  \hspace{1em} \text{Lift coefficient}

\( C_p \)  \hspace{1em} \text{Pressure coefficient}
\[ d \quad \text{Wall distance (SA model)} \]

\[ D_{ij} \quad \text{Tensor of the diffusion of the Reynolds Stresses} \]

\[ D_{ij}^\nu \quad \text{Tensor of the molecular diffusion of the Reynolds Stresses} \]

\[ D_{ij}^p \quad \text{Tensor of the pressure transport of the Reynolds Stresses} \]

\[ D_{ij}^t \quad \text{Tensor of the turbulent transport of the Reynolds Stresses} \]

\[ d_k \quad \text{Diffusion of turbulent kinetic energy} \]

\[ f_d \quad \text{DDES boundary layer shield function} \]

\[ f_{ij} \quad \text{Elliptic variable in the elliptic relaxation approach} \]

\[ k \quad \text{Turbulent kinetic energy} \]

\[ k_{SGS} \quad \text{Sub-grid scale energy} \]

\[ L \quad \text{Length scale} \]

\[ l \quad \text{Mixing length scale} \]

\[ L_{\nu K} \quad \text{von Karman length scale} \]

\[ L_{DDES} \quad \text{DDES length scale} \]

\[ L_{DES} \quad \text{DES length scale} \]

\[ L_{IDDES} \quad \text{IDDES length scale} \]

\[ L_{LES} \quad \text{LES length scale} \]

\[ L_{RANS} \quad \text{RANS length scale} \]

\[ P \quad \text{Mean pressure} \]

\[ p \quad \text{Pressure} \]

\[ p' \quad \text{Fluctuating pressure} \]

\[ P_{ij} \quad \text{Turbulent production tensor of Reynolds Stresses} \]

\[ P_k \quad \text{Production of turbulent kinetic energy} \]
$r_d$  Modelled turbulence boundary layer sensor function

$Re$  Reynolds number

$S_{ij}$  Strain rate tensor

$St$  Strouhal number

$T$  Time scale

$t$  Time

$U_\infty$  Free-stream velocity

$U_i$  Mean velocity vector

$u_i$  Velocity vector

$u'_i$  Fluctuating velocity vector

$x_i$  $i^{th}$ component of the position vector

$y$  Wall distance or coordinate in the wall-normal direction (except for Ahmed car body where it is the span-wise direction)

$y^+$  Non dimensional wall distance
Chapter 1

Introduction

1.1 Background

Computational Fluid Dynamics (CFD) has developed over the past 50-60 years from a complex tool limited to a small group of academic researchers to something that has found widespread use for a vast range of applications. From simulating the flow of medicines within the blood stream to designing a Formula 1 car, the use of CFD is becoming ever more available to a greater number of users and applications. The rise in cheap computer power is a large factor in this expansion and has enabled the simulation of complex geometries using hundreds of millions of cells; a feat that would have been beyond the wildest dreams of many of the early developers of CFD.

Whilst the rise of CFD has brought much more accurate representations of fluid flow, there is still a long way to go before it can be used to accurately predict high-Reynolds number flow, i.e when inertial forces far outweigh viscous forces and turbulence occurs. At such high-Reynolds numbers the high grid resolution requirements for the most accurate methods remain beyond current computer resources and are likely to stay this way for many decades to come [5]. The operational need for fast, yet accurate, simulations has spurred on much of the development of CFD, and perhaps the key limiting factor is how best to represent the effect of turbulence on the flow. Although the Navier-Stokes equations fully describe the motion of a turbulent flow, an analytical solution is only possible for a limited number of very simple cases at low-Reynolds numbers. To solve these equations for more complex geometries at higher-Reynolds numbers an iterative approach is required. Unfortunately this is extremely computationally expensive.
because of the requirement for the spatial discretization to be small enough to capture the smallest eddies and the smallest time scales at which they evolve. This approach, Direct Numerical Simulation (DNS) is thus at present limited to relatively simple flows at modest Reynolds numbers and is unlikely to become available for general use until computational resources have become sufficiently more powerful.

It is for this reason that the past 50 years of turbulence modelling research has focused on developing methodologies to model and approximate turbulence such that it can become viable to capture its influence using the practically available computational resources. Two broad approaches have formed to tackle this problem; Large Eddy Simulation (LES) where the smallest eddies are modelled (as they are considered isotropic and only weakly dependent on the flow) and the rest of the eddies resolved, and then Reynolds-Averaged Navier-Stokes (RANS) models where the flow equations are time-averaged and all the turbulent content is modelled.

Each of these approaches have their own advantages and disadvantages; RANS models can be solved in a steady fashion and can often be solved within short timescales, which has made them attractive to industry. In contrast LES models offer a higher level of accuracy, but compared to steady RANS simulations, the computational cost is considerably higher; which has to date limited much of its use to academic cases. Thus the choice between these approaches is dependent on the available computer resources and the type of information that is required from the simulation (i.e. mean or unsteady information).

Hybrid RANS-LES approaches combine the best elements of each of the aforementioned methodologies; by using a RANS model in regions where the flow is more universal and isotropic, i.e. the steady attached regions, and then switching to a LES mode for the unsteady regions, where modelling the turbulence is more difficult and less accurate. How to couple these two methods together and the choice of the underlying RANS model is not a trivial issue and is a major focus of research.

The development of CFD over the past 50 years has seen a shift from solving meshes of hundreds of cells on a single computer core, to hundreds of millions of cells being computed on thousands of computer cores running in parallel. Although it is not clear where the precise direction of CFD development will lead to over the next 50 years, it is without doubt that a simulation which seems complex
and computationally expensive using current computer resources will be deemed trivial in 50 years time.

1.2 Study Objectives

Hybrid RANS-LES approaches have seen a surge in popularity in the past decade as academia and industry are increasingly interested in instantaneous information of high-Reynolds number turbulent flows. For these flows, Large-Eddy Simulation (LES) is still not a practical option for most industrial applications; considering the currently available computational resources (and the time allocated for a simulation). Reynolds-Averaged Navier-Stokes (RANS) models are well established as numerical tools for industrial applications, but when applied to highly unsteady separated flows their limitations are well-known. Their inability to capture the large scale unsteadiness and a tendency to under-predict the shear-stress in the separated layer (which leads to a longer separation length) mean they are unsuitable for these types of flows [6].

Hybrid RANS-LES methods attempt to merge these two modelling approaches and apply them either zonally or non-zonally. Hybrid methods aim to relax grid resolution requirements relative to original LES, by providing improved sub-grid scale RANS modelling, in particular to capture the near wall turbulence (therefore enabling a coarser near-wall grid to be applied). In addition, applying LES in the regions of highly unsteady flow allows the large scale instabilities to be resolved, which in turn can provide more information about the flow structure.

This work is focussed on one such hybrid RANS-LES method, namely Delayed Detached-Eddy Simulation (DDES) [7]. In the present work, several versions of DDES have been implemented into the finite volume code Code_Saturne to highlight the importance of the underlying RANS model within DDES. A new formulation of DDES has been derived based upon the principle of elliptic relaxation and after initial calibration and validation work, it has been evaluated on a range of test cases to compare its performance against current DDES formulations.

The specific objectives of this work are thus as follows:

- Implement, calibrate and validate the main formulations of DDES (SST and SA based DDES models) using simple test cases.
• Develop as necessary any numerical solvers to facilitate the use of DDES within Code_Saturne.

• Derive, implement and carefully validate a new DDES formulation based upon elliptic relaxation for improved near-wall modelling.

• Carefully evaluate these DDES formulations on a range of test cases, included pressure-induced separation, geometry-induced separation and highly 3D flows that combine both types of separation.

• Investigate the sensitivity of each DDES formulations to different numerical schemes for a variety of flows to assess the importance of selecting the correct numerical scheme.

The overriding theme of this work is therefore not only to develop a new version of DDES based upon a novel RANS model but to assess whether or not there is any merit in employing more capable RANS models into hybrid RANS-LES schemes.

1.3 Outline of thesis

This thesis is presented as follows: after this introduction Chapter 2 provides an overview of turbulence modelling, from the earliest RANS models to the latest research in hybrid RANS-LES methods. Chapter 3 goes on to describe the finite volume CFD code that was used as well as details of the turbulence models employed within this work and their validation and calibration work. Chapter 4 describes the development of the φ − f DDES model in more detail, and includes the derivation, implementation and calibration of the model for decaying isotropic turbulence. The sensitivity to the numerical scheme and the hybrid numerical scheme developed during the course of this work is also discussed. Chapter 5 presents the work undertaken to evaluate the performance of DDES formulations based upon the SA, SST and φ − f RANS models on the 2D periodic hills at \( Re = 10590 \) and \( Re = 37000 \). This case was selected as the starting point for the model validation, as there is an extensive range of experimental and LES results available. Chapter 6 describes the work done to evaluate the aforementioned DDES formulations on the NACA0021 airfoil post-stall at 60° incidence. This test case has been extensively used to evaluate new DDES formulations and
is precisely the type of flow for which DES was originally designed. Chapter 7 presents the extensive work completed on the 2D wall-mounted hump at a high-Reynolds number. Although this flow exhibits separation that is geometry induced, it has been shown to be challenging to predict accurately. Chapter 8 describes the work done for the Ahmed car body using each of the DDES formulations used throughout the thesis. It is a good example of a complex flow at high-Reynolds number that is subject to a mixture of pressure and geometry-induced separation. It is also a case that has exhibited a high sensitivity to the underlying RANS model within a hybrid RANS-LES approach in previous studies. Chapter 9 brings together the conclusions from throughout the thesis and describes the possible directions of future investigation.
Chapter 2

Turbulence Modelling

The Navier-Stokes equations represent the fundamental motion of a fluid. They are derived from the conservation of mass and momentum, and can be expressed in Cartesian form for an incompressible Newtonian flow with constant density and variable viscosity as follows:

\begin{align}
\frac{\partial u_i}{\partial x_i} &= 0 \quad (2.1) \\
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2.2)
\end{align}

where the viscous stress \( \tau_{ij} \) is defined as:

\[ \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.3) \]

While these equations fully represent any fluid flow, an analytical solution is only possible for basic flows that are subject to many assumptions. When an analytical solution is not possible, numerical methods may be used to achieve an approximate solution. In order to obtain a solution, these differential equations must be discretized to form integral equations which can be solved by a numerical procedure.

By solving these equations directly without further modelling or assumptions, the fluid velocity and pressure can be obtained, this method is called Direct Numerical Solution (DNS). Whilst it is the most complete numerical method as
it involves no further assumptions or modelling, it is not practical for industrial applications using the current generation of computational resources. Three alternative approaches are discussed below; Reynolds Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES) and hybrid RANS-LES.

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

This method relies on providing a statistical representation of the turbulence via mathematical models, in which case none of the turbulence is resolved. All such methods are based on the Reynolds temporal decomposition; this involves applying an ‘ensemble-averaging’ process to the whole turbulent flow field, $\phi(x,t)$, which, for the case where the flow is statistically stationary, is given by:

$$\phi(x,t) = \Phi(x) + \phi'(x,t) \quad (2.4)$$

where $\phi'(x,t)$ is the fluctuating component and $\Phi(x)$ is the mean component, which can be defined as:

$$\Phi_i = \lim_{\tau \to \infty} \frac{1}{\tau} \int_{t_0}^{t_0+\tau} \phi_i \, dt \quad (2.5)$$

When this temporal decomposition is applied to the Navier-Stokes equations (Equations 2.1 and 2.2), the Reynolds-Averaged Navier-Stokes (RANS) equations are attained, which can be expressed as the following:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (2.6)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{1}{\rho} \tau_{ij} - \frac{\partial \overline{u_i u_j}}{\partial x_j} \quad (2.7)$$

This averaging process produces an extra term that is not present in the Navier-Stokes equations. This term, $\overline{u_i u_j}$, is the Reynolds Stress tensor, which represents the effect of the turbulence fluctuations on the mean flow. It leads to a further 6 variables and therefore modelling is required in order to obtain full closure of Equation 2.7. In the following, only incompressible flows are considered and as such $\rho$ is assumed to take a value of unity.

There are two main approaches to modelling the Reynolds Stresses. The first approach is to model each Reynolds Stress individually and solve a transport
equation for each. This is the most complete method as individual components are accounted for independently and several terms of the transport equations are exact in their form. However as there are six Reynolds stresses, an additional six transport equations are required, as well as a transport equation for the length scale, such as $\varepsilon$ or $\omega$ which can increase the computational cost of a simulation.

An alternative approach to closing Equation [2.7] is to model these Reynolds Stresses using the Boussinesq approximation \[8\]. This assumes that the effect of turbulence on the main flow can be treated phenomenologically as a viscosity, a ‘turbulent viscosity’. This latter approach is now discussed in more detail.

**Boussinesq approximation**

In the Navier-Stokes equations, the viscous stresses in the momentum equations are modelled by Newton’s theory of viscosity. This states that the viscous stresses are linearly proportional to the mean rate of strain, $S_{ij}$. Boussinesq \[8\] proposed that the Reynolds stresses that result from the turbulent fluctuations in the velocity field can be modelled in a similar fashion, as they are shown to increase with the rate of deformation. It is logical to assume that as the molecular viscosity links the viscous stresses and strains, then the stresses due to turbulence (the Reynolds stresses) can be linked by a turbulent viscosity, $\nu_t$, to the mean strain rate. While physically, the molecular viscosity is a thermodynamic property of a fluid, the turbulent viscosity is dependent on the local flow itself, and this is only an engineering approximation rather than an actual physical property;

$$-\overline{u_i' u_j'} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij},$$

(2.8)

where

$$\delta_{ij} = \begin{cases} 1, & \text{if } i = j \\ 0, & \text{if } i \neq j \end{cases}$$

(2.9)

$$k = \frac{1}{2} \overline{u_i' u_i'}$$

(2.10)

$$S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$

(2.11)
In order to solve the equations, the turbulent viscosity must be determined. It is the product of a turbulent length and velocity scale and the specification of these scales is the focus of the following discussion.

2.1.1 Algebraic models

One of the earliest examples of a model that provides an approximation for the turbulent viscosity is the Mixing-Length Hypothesis (MLH) [9]. It is based upon an algebraic relationship rather than through the solution of a differential equation and thus is termed an algebraic or ‘zero equation’ model. Through analogy with the kinetic theory of gases (following Boussinesq), this model assumes that in a simple 2D shear flow where the only significant velocity gradient is $\frac{\partial U}{\partial y}$ and the only significant Reynolds stress is $-\overline{u'v'}$, the turbulent viscosity has the form:

$$\nu_t = l^2 \left| \frac{\partial U}{\partial y} \right|$$  \hspace{1cm} (2.12)

where $l$ is the mixing length that must be specified for each flow being investigated (e.g. a boundary layer, a mixing layer or jet). For a simple shear flow, the mixing length would be proportional to the width of the shear layer while for a boundary layer it can be split into three regions; the viscous sub-layer, the ‘log layer’ and the buffer layer.

In the log-layer $l = \kappa y$, where $\kappa = 0.41$ is the Von Karman constant. Outside of this region the mixing length is proportional to the boundary layer thickness:

$$\text{for } y < 0.22\delta : \quad l = \kappa y$$ \hspace{1cm} (2.13)

$$\text{for } y \geq 0.22\delta : \quad l = 0.22\kappa y\delta$$ \hspace{1cm} (2.14)

In order to take into account the effect of the wall in the viscous sub-layer, van Driest proposed a damping function $f_d$, for the mixing length:

$$f_d = \left[1 - e^{-y^+/\delta^+}\right]$$ \hspace{1cm} (2.15)

where $y^+ = y\mu_r/\nu$ is the dimensionless wall distance in which the friction velocity
\( \mu_r \) is related to the wall shear stress \( \tau_w \) as \( \mu_r = \sqrt{\tau_w / \rho} \) and \( A^+ = 26 \) is the damping coefficient.

Several algebraic models have developed Prandtl’s ideas further in order to generalise them for a wider range of flows. Smagorinsky [10] proposed a version of Equation 2.12 based on the rate of strain tensor, \( \hat{S}_{ij} \) where here \( \hat{\cdot} \) refers to a spatially filtered quantity:

\[
\nu_t = l^2 \sqrt{2 \hat{S}_{ij} \hat{S}_{ij}} \quad (2.16)
\]

A further modification of the Mixing-Length model is the Baldwin-Lomax model [11] which is based on the rotation tensor, \( \Omega_{ij} \).

\[
\nu_t = l^2 \sqrt{2 \Omega_{ij} \Omega_{ij}} \quad (2.17)
\]

\[
\Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \quad (2.18)
\]

In this model a two-layer mixing-length approach is taken, where an inner and outer turbulent viscosity is given to further sensitise the turbulent viscosity to the position in the flow. The inner turbulent viscosity is defined as Equation 2.17 whereas the outer turbulent viscosity is a function of the wall distance and velocity which through the use of several calibrated functions adjusts the turbulent viscosity to better capture the outer region of the boundary layer.

The simplicity of this modelling approach makes it attractive for the first analysis of very simple flows. However if a case involves a mildly complex geometry, defining the mixing length can be impossible as the flow may change a lot in character in different areas of the domain, and may become separated altogether. The second drawback is that the velocity scale is taken to be a local quantity, i.e the local mean velocity gradient. This assumption leads to the assertion that when the velocity gradient is zero then so is the eddy viscosity (through the velocity scale). In cases such as decaying turbulence and the centreline of a round jet, the eddy viscosity is not zero even when the velocity gradients are zero [12].
2.1.2 One-equation models

One-equation models include some of the history effects of the flow through solving a transport equation for a velocity scale. Prandtl [13] and Kolmogorov [14] independently concluded that it is better to base the velocity scale on the turbulent kinetic energy $k$, rather than the local mean velocity. Taking the turbulent length scale to be the mixing length, then the turbulent viscosity becomes:

$$\nu_t = k^{1/2}l$$  \hspace{1cm} (2.19)

where a transport equation is solved to model $k$:

$$\frac{Dk}{Dt} = P_k + \varepsilon + d_k$$  \hspace{1cm} (2.20)

In which $P_k$, $\varepsilon$ and $d_k$ represent the production, dissipation and diffusion of $k$ respectively. The production of $k$ is of the form:

$$P_k = -\overline{u'_i u'_j}S_{ij}$$  \hspace{1cm} (2.21)

The dissipation rate $\varepsilon$ in high-Reynolds number flows away from the wall scales as $u^*^3/l^*$, where $u^*$ and $l^*$ are the velocity and length scales of the flow. Thus $\varepsilon$ is modelled as:

$$\varepsilon = \frac{C_D k^{3/2}}{l}$$  \hspace{1cm} (2.22)

Finally the diffusion of $k$ is modelled using a gradient-diffusion model:

$$d_k = \frac{\partial}{\partial x_i} \left[ (\nu + \nu_t) \frac{\partial k}{\partial x_i} \right]$$  \hspace{1cm} (2.23)

where $\sigma_k = 1$. Substituting Equation 2.22 into Equation 2.19 results in the turbulent viscosity being of the form:

$$\nu_t = C_\mu k^2/\varepsilon$$  \hspace{1cm} (2.24)

Where $C_\mu = 0.09$ is calculated from examining the evolution of $\overline{u'v'}/k$ in a fully
developed channel flow and taking the value that is approximately constant away from the wall.

While a one-equation model based on $k$ does show improvement over algebraic models [15], a major drawback is the need for a mixing length to be prescribed a priori (as for the MLH).

An alternative one-equation model is the Spalart-Allmaras model [16] which is based on a different modelling strategy, motivated by industrial use in the aerospace industry in particular, and is derived using dimensional analysis and a term-by-term modelling approach.

The Spalart-Allmaras solves a transport equation for a term $\tilde{\nu}$, that is equal to the turbulent viscosity far from any walls:

$$
\frac{D\tilde{\nu}}{Dt} = \left[ c_{b1}(1 - f_{t2})\tilde{S}\tilde{\nu} - \left[ c_{w1}f_{w} - \frac{c_{b1}}{\kappa^2 f_{t2}} \right] \left( \frac{\tilde{\nu}}{d} \right)^2 \right] + 
\tilde{\nu} \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( (\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right]
$$

(2.25)

Which is solved to calculate the turbulent viscosity:

$$
\nu_t = f_{v1}\tilde{\nu}
$$

(2.26)

Where,

$$
f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}
$$

(2.27)

$$
\chi = \frac{\tilde{\nu}}{\nu}
$$

(2.28)

Where $\nu$ is the molecular kinematic viscosity, and additional terms are defined as follows:

$$
\tilde{S} = \Omega + \frac{\tilde{\nu}}{\kappa^2 d^2 f_{v2}}
$$

(2.29)
In which $\Omega = \sqrt{2\Omega_{ij}\Omega_{ij}}$ is the vorticity magnitude, $d$ is the distance to the nearest wall and $\kappa = 0.41$ is the Von Karman constant. There are several damping functions introduced which are calibrated to match the experimentally observed variation of $\nu_t$ near the wall:

$$f_{c2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (2.30)$$

The damping function $f_w$ damps the sink term of Equation 2.25 near the wall and also provides a sensitivity to the pressure gradient.

$$f_w = g \left[ \frac{1 + c_w}{g^b + c_w^b} \right]^{1/6} \quad (2.31)$$

$$g = r + c_{w2}(r^b - r) \quad (2.32)$$

$$r = \frac{\tilde{\nu}}{S_\kappa^2 d^2} \quad (2.33)$$

The additional term $f_{t2}$ is designed to account for transition and enables to model ‘trip’ itself from laminar to turbulent flow:

$$f_{t2} = c_{t3} \exp(-c_{t4} \chi^2) \quad (2.34)$$

The model constants are as follows:

<table>
<thead>
<tr>
<th>$c_{t1}$</th>
<th>$c_{t2}$</th>
<th>$\sigma$</th>
<th>$c_{w1}$</th>
<th>$c_{w2}$</th>
<th>$c_{w3}$</th>
<th>$c_{v1}$</th>
<th>$c_{t3}$</th>
<th>$c_{t4}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1355</td>
<td>0.622</td>
<td>2/3</td>
<td>$c_{t1}/\kappa^2 + (1 + c_{t2})/\sigma$</td>
<td>0.3</td>
<td>2</td>
<td>7.1</td>
<td>1.2</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Table 2.1: Model coefficients for the SA model

Several modifications to the Spalart-Allmaras model have been proposed to improve some of the weaknesses of the original model. One such modification is a modified $f_{v1}$ and $\tilde{S}$ term to improve the near-wall numerical behaviour [17]. Another version of the model; the strain-adaptive linear Spalart-Allmaras model (SALSA), attempts to bring improvement for non-equilibrium flows by sensitising the production term to highly strained flows [18]. This is achieved by making the constant $C_{t1}$ in the production term a function of a mixing length scale based turbulent viscosity and $\tilde{\nu}$. This function causes a reduction of the production.
term for excessive strains. The Spalart-Allmaras model has found widespread use in industrial flows because of its ease of implementation and robustness, and further versions have been proposed to account for wall roughness [19] and compressibility effects [20]. The reader is advised to visit the NASA Spalart Allmaras website that has extensive details of all the published SA model versions (http://turbmodels.larc.nasa.gov/spalart.html).

2.1.3 Two-equation models

While one-equation models bring improvement over algebraic models by incorporating a single transport equation for the velocity scale, two-equation models extend this one step further and also developed a transport equation for a length scale, to remove the need for one to be empirically defined. A range of models in this category were proposed, the most popular of which by far are the $k-\varepsilon$ model [21] and the $k-\omega$ model [15]. Both of these approaches base their velocity scale on the turbulent kinetic energy and solve an additional transport equation for the length scale.

Dimensional analysis indicates that the turbulent viscosity $\nu_t$ is the product of a turbulent velocity scale and a turbulent length scale. The approach of Launder et al. [21] is to use the turbulent energy dissipation rate $\varepsilon$, which scales to the length scale as $k^{3/2}/\varepsilon$. Whereas the approach of Wilcox is to use the specific dissipation rate ($\varepsilon/k$), which results in the length scale being proportional to $k^{1/2}/\omega$ [15]. Which leads to the turbulent viscosity having the form for the $k-\varepsilon$ model:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (2.35)$$

and for the $k-\omega$ model,

$$\nu_t = \frac{k}{\omega} \quad (2.36)$$

The model constant $C_\mu = 0.09$, which is taken from an empirically observation in local equilibrium flows (where $P_k = \varepsilon$). For the $k-\varepsilon$ model a transport equation is solved for $\varepsilon$, which is of the form:
\[
\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{P_k}{k} \varepsilon - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]
\]  
(2.37)

The constant \( C_{\varepsilon 1} \approx 1.44 \) is calibrated from homogeneous shear flow, also the value \( C_{\varepsilon 2} \approx 1.92 \) is calibrated using decaying homogeneous isotropic turbulence. The value of \( \sigma_\varepsilon \) is calculated via the following expression, which results from considering model behaviour under idealised conditions (steady, local equilibrium, zero pressure gradient flow):

\[
\sigma_\varepsilon = \frac{\kappa^2}{\sqrt{C_\mu (C_{\varepsilon 2} - C_{\varepsilon 1})}}
\]  
(2.38)

The exact values of these constants depends on the required outcome of the model in certain flow regimes, increasing or decreasing \( C_{\varepsilon 2} \) may improve the performance for one flow but yield inferior results for another.

While the transport equation for \( k \) was derived exactly from the Navier-Stokes equations with some modelling applied to some of the terms, the equation for \( \varepsilon \) is considered to be almost entirely empirical [12]. While it is possible to derive an exact equation for \( \varepsilon \) it has many terms which are different obtain experimentally or even numerically (such as higher order correlations of fluctuating velocity gradients) and so validation and/or model development is not feasible in a simple RANS framework.

The \( k - \varepsilon \) model is one of the most widely used models in academia and especially industry. Its use in many commercial CFD codes over the past 40 years has made it popular due largely to it’s well documented and well understood performance (good and bad), as well as its robustness and relative ease of implementation. The \( k - \varepsilon \) so far described is a high-Reynolds number model that is not suitable for integration down to the wall, the methods to deal with this are discussed later.

The major alternative to the use of \( \varepsilon \) as the length-scale determining variable is the \( k - \omega \) model of Wilcox [15]. Like the \( k - \varepsilon \) model a transport equation is solved for \( k \) and now also \( \omega \):

\[
\frac{Dk}{Dt} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]
\]  
(2.39)

\[
\frac{D\omega}{Dt} = \frac{\omega}{k} P_k - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]
\]  
(2.40)

40
The dissipation and length scales are calculated by:

\[ \varepsilon = \beta^* \omega k \]  \hspace{1cm} (2.41)
\[ l = \frac{k^{1/2}}{\omega} \]  \hspace{1cm} (2.42)

Thus leading to the turbulent viscosity \( \nu_t \) being defined as:

\[ \nu_t = \frac{k}{\omega} \]  \hspace{1cm} (2.43)

Where \( \beta^* = 0.09, \alpha = 0.55, \beta = 0.075, \sigma_k = 2 \) and \( \sigma_\omega = 2 \).

A well known advantage of the \( k - \omega \) model over the \( k - \varepsilon \) model is observed in areas of low turbulence, for which the standard \( k - \varepsilon \) is prone to numerical problems since the term \( \varepsilon^2/k \) tends to infinity as \( k \) tends to zero (as \( \varepsilon \) does not tend to zero at the wall). The \( k - \omega \) model does not require a damping function and can be solved down to the wall.

One disadvantage of the \( k - \omega \) model is its sensitivity to free-stream values of \( \omega \), outside of the shear layer. As reported in Menter [22] the value of the turbulent viscosity can change by up to 100% depending on the value of the free-stream value of \( \omega \); a feature which is undesirable in any model.

A major contribution was made by Menter [22] to develop a zonal model that combined the best features of both the \( k - \varepsilon \) and \( k - \omega \) models. The Baseline (BSL) \( k - \omega \) model [22] specifically addressed the free-stream dependancy of the original \( k - \omega \) by combining the high-Reynolds number \( k - \varepsilon \) model described in Equation 2.37 with the original \( k - \omega \) model described in Equation 2.40. The \( k - \omega \) model is active in the near-wall region, where it has shown superior performance over the \( k - \varepsilon \) model, and then gradually changes to the free-stream independent \( k - \varepsilon \) model in the outer wake region.

In order that only one set of transport equations is solved, the \( k - \varepsilon \) is transformed into the \( k - \omega \) model and the resulting cross-diffusion term that emerges from this operation is multiplied by a blending function \( (F_1) \) to ensure that the model returns to the \( k - \omega \) model in the near-wall regions. It is also used to compute the different model coefficients of each model. The BSL equations are:
\[
\frac{Dk}{Dt} = P_k - C_k k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.44)
\]

\[
\frac{D\omega}{Dt} = \alpha \frac{P_k}{\nu_t} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_t} \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \left( 1 - F_1 \right) \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.45)
\]

The blending function \( F_1 \) that combines the model constants and blends the \( k - \varepsilon \) and \( k - \omega \) models is defined as:

\[
F_1 = \tanh(\arg_1^4) \quad (2.46)
\]

\[
\arg_1 = \min \left[ \max \left( \frac{\sqrt{k}}{C_{\mu \omega y}}, \frac{500 \nu}{y^2 \omega} \right), \frac{4 \rho k}{\sigma_{\omega} CD_{k \omega} y^2} \right] \quad (2.47)
\]

\[
CD_{k \omega} = \max \left( 2 \rho \frac{1}{\sigma_{\omega} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right) \quad (2.48)
\]

Where \( y \) is the distance to the closest no-slip wall.

The values of the model constants \( \alpha, \beta, \sigma_k, \) and \( \sigma_\omega \) from Equations \( 2.57 \) & \( 2.58 \) are combined using the blending function \( F_1 \). Which for \( \alpha \) is:

\[
\alpha = F_1 \alpha_1 + (1 - F_1) \alpha_2 \quad (2.49)
\]

\[
\beta = F_1 \beta_1 + (1 - F_1) \beta_2 \quad (2.50)
\]

Where \( \alpha_1 \) corresponds to the value of \( \alpha \) in the \( k - \omega \) mode and \( \alpha_2 \) corresponds to the value of \( \alpha \) in the \( k - \varepsilon \) mode. Table \ref{table:2.2} shows the model constants for both modes.

A further modification proposed by Menter \cite{22}, the Shear Stress Transport (SST) model, includes a redefinition of the turbulent viscosity in order to limit the growth of the turbulent shear stress in adverse pressure gradients, so that the shear stress is proportional to the turbulent kinetic energy, \( k \). A blending function \( F_2 \) (which is similar to form to the \( F_1 \) function) ensures that in the flow region outside the boundary layer, turbulent viscosity returns to the standard definition of \( \nu_t = k/\omega \).
\[ \nu_t = \frac{a_1 k_{\text{max} \left(a_1 \omega, SF_2 \right)}}{\max \left(a_1 \omega, SF_2 \right)}, \quad a_1 = 0.31 \] (2.51)

\[ F_2 = \tanh(\text{arg}_2^2) \] (2.52)

\[ \text{arg}_2 = \max \left( \frac{2 \sqrt{k}}{C_{\mu} \omega y}, \frac{500 \nu}{y^2 \omega} \right) \] (2.53)

The final modification by Menter [22] is to the production term of the turbulent kinetic energy \( (P_k) \) which is limited to ensure that in stagnation regions where the strain rate goes to zero that the turbulence isn’t over-predicted (as the production term with a standard eddy viscosity becomes quadratic in order). The limiter is of the form:

\[ P_k = \min(P_k, 10 C_\mu k \omega) \] (2.54)

\[ P_k = \nu_t S^2 \] (2.55)

\[ S = \sqrt{2 S_{ij} S_{ij}} \] (2.56)

Thus the final form of the equations for the SST model are as defined below:

\[
\frac{Dk}{Dt} = \min(P_k, 10 C_\mu k \omega) - C_\mu k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right]
\] (2.57)

\[
\frac{D\omega}{Dt} = \alpha \frac{P_k}{\nu_t} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \sigma_\omega \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \frac{1 - F_1}{\sigma_\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\] (2.58)

<table>
<thead>
<tr>
<th>( k - \omega ) mode (1)</th>
<th>( k - \varepsilon ) mode (2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_k )</td>
<td>0.85</td>
</tr>
<tr>
<td>( \sigma_\omega )</td>
<td>0.5</td>
</tr>
<tr>
<td>( \beta )</td>
<td>0.0750</td>
</tr>
<tr>
<td>( C_\mu )</td>
<td>0.09</td>
</tr>
<tr>
<td>( \kappa )</td>
<td>0.41</td>
</tr>
<tr>
<td>( \alpha = \beta / C_\mu - \sigma_\omega \kappa^2 / \sqrt{C_\mu} )</td>
<td>0.553</td>
</tr>
</tbody>
</table>

Table 2.2: Model coefficients for the \( k - \omega \) SST model
Non-Linear Eddy Viscosity Models

While linear eddy viscosity models have become useful tools in the prediction of turbulent flows, they are based upon several fundamental assumptions that limit their applicability to more complex flows [23]. Firstly all linear eddy viscosity models use the Boussinesq approximation such that the Reynolds stress anisotropy is directly proportional the mean rate of strain: \( a_{ij} = -(2\nu_t/k)S_{ij} \). While this may be acceptable for simple 2D shear flows, for more complex flows such as three-dimensional flows or ones with strong curvature, this assumption is not valid. Another disadvantage of standard linear eddy viscosity models that was addressed by a number of models [22, 24, 25] is that as a consequence of the linear eddy viscosity approximation, the turbulent kinetic energy production term becomes proportional to the square of the strain rate. This leads to an over-prediction of the level of turbulence at points where the strain rate invariant is high, and requires either limiters or a reformulation of this term to overcome this problem.

An alternative approach to modelling the Reynolds stresses based solely on the strain rate via the Boussinesq approximation is to express the stress anisotropy tensor \( a_{ij} \) in a non-linear fashion with respect to the strain and vorticity tensors. This approach enables that some of the disadvantages of linear eddy viscosity models to be overcome.

Non-linear eddy viscosity models are based on the concept that \( a_{ij} \) can be expressed as a combination of non-linear polynomials of the mean velocity gradients. Pope [26] used the Cayley-Hamilton theorem to demonstrate that if stress anisotropy depends only strain and vorticity tensors then it can be expressed in ten tensorial groups:

\[
a_{ij} = \beta_1 S_{ij} + \beta_2 (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) + \beta_3 (S_{ik} S_{kj} - \frac{1}{3} S_{kl} S_{kl} \delta_{ij}) \\
+ \beta_4 (\Omega_{ik} S_{kj} - \frac{1}{3} \Omega_{ik} \Omega_{kl} \delta_{ij}) + \beta_5 (\Omega_{il} S_{lm} S_{mj} - S_{il} S_{lm} \Omega_{mj}) \\
+ \beta_6 (\Omega_{il} \Omega_{lm} S_{mj} - S_{il} \Omega_{lm} \Omega_{mj} - \frac{2}{3} S_{lm} \Omega_{mn} \Omega_{nl} \delta_{ij})
\] (2.59)

where the coefficients \( \beta_{1,6} \) are typically functions of \( k \) and \( \varepsilon \), with transport equations solved for \( k \) and \( \varepsilon \) in a similar fashion to the standard \( k - \varepsilon \) model.

Several models have been developed based upon this approach, most notably by Craft et al [27] and Yoshizawa et al., [28] and also a non-linear version of the
v^2 - f model [29], however further descriptions of these models is beyond the scope of this work.

### 2.1.4 Reynolds Stress Models

All eddy viscosity models - whether linear or non-linear - employ algebraic relationships between the Reynolds stresses and other variables. An alternative approach to using the Boussinesq assumption to model the Reynolds Stresses is to solve a transport equation for each component of the Reynolds Stress tensor, as shown in Equation (2.60).

\[
\frac{D\bar{u}_i\bar{u}_j}{Dt} = P_{ij} + \phi_{ij} - \varepsilon_{ij} + D_{ij}
\]  

(2.60)

Where the production term is defined exactly as

\[
P_{ij} = -\bar{u}_i\bar{u}_k \frac{\partial U_j}{\partial x_k} - \bar{u}_j\bar{u}_k \frac{\partial U_i}{\partial x_k}.
\]  

(2.61)

The dissipation is the rate at which the kinetic energy from the larger eddies is transferred to the smaller eddies. It is defined by,

\[
\varepsilon_{ij} = 2\nu \frac{\partial u_i^r}{\partial x_k} \frac{\partial u_j^r}{\partial x_k}.
\]  

(2.62)

The pressure-strain can be thought of the redistribution of the turbulent energy by the fluctuating velocity field [12];

\[
\phi_{ij} = -\frac{p'}{\rho} \left( \frac{\partial u_i^r}{\partial x_j} + \frac{\partial u_j^r}{\partial x_i} \right).
\]  

(2.63)

Diffusion (turbulent, viscous and pressure fluctuations) is,

\[
D_{ij} = \frac{\partial}{\partial x_k} \left( \begin{array}{c}
\nu \frac{\partial u_i^r}{\partial x_k} - \bar{u}_i\bar{u}_j \frac{\partial U_j}{\partial x_k} - \frac{p'}{\rho} \frac{\partial u_i^r}{\partial x_k} - \frac{p'}{\rho} \frac{\partial u_j^r}{\partial x_k} \\
D_{ij}^v - D_{ij}^\rho
\end{array} \right).
\]  

(2.64)

**Dissipation**

For high Reynolds numbers the dissipation is assumed to be isotropic (far...
from the near-wall region) and as such the following relationship is used:

\[ \varepsilon_{ij} = \frac{2}{3} \varepsilon \delta_{ij} \]  

(2.65)

The transport equation generally employed for \( \varepsilon \) is similar to that commonly used in two-equation models. Hanjalic and Launder [30] proposed a transport equation for \( \varepsilon \) of the form,

\[ \frac{D \varepsilon}{Dt} = C_{\varepsilon 1} \frac{P_k \varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left( \frac{C_{\varepsilon}}{\varepsilon} \overline{u_i u'_j} \frac{\partial \varepsilon}{\partial x_j} \right) . \]

(2.66)

This transport equation does not include low Reynolds effects and as such will not be accurate for these types of flows. Also the assumption of isotropy is not valid in the near-wall region, several approaches have been developed to compensate for this and they will be discussed later.

**Pressure-Strain**

The pressure-strain term is the main focus of modelling within the Reynolds Stress Transport approach. Its effect is to redistribute the energy between the Reynolds stresses. The pressure-strain term is typically split into three components:

\[ \phi_{ij} = \phi_{ij1} + \phi_{ij2} + \phi_w, \]

(2.67)

Where,

- \( \phi_{ij1} \) is the term depending only on the turbulence (the slow term)
- \( \phi_{ij2} \) is the term accounting for the effect of mean strain (the rapid term)
- \( \phi_w \) accounts for the presence of the wall

The most commonly used models are those proposed by Launder et al. [31] (the LRR model), and that of Speziale et al. [32] (the SSG model). Both of which can written in the following general form:

\[ \pi_{ij} = -(C_1 \varepsilon + C_1^* P_k) a_{ij} + C_2 \varepsilon (a_{ik} a_{kj} - \frac{1}{3} \delta_{ij} A_2) \]

(2.68)

\[ (C_3 - C_3^* \sqrt{A_2}) k S_{ij} + C_4 k (a_{ik} S_{jk} + a_{jk} S_{ik} - \frac{2}{3} a_{im} S_{lm} \delta_{ij}) \]

(2.69)

\[ + C_5 k (a_{ik} \Omega_{jk} + a_{jk} \Omega_{ik}) \]

(2.70)
The LRR model can be achieved by setting $C_1^*$ and $C_2$ to zero. The model coefficients are shown below:

<table>
<thead>
<tr>
<th></th>
<th>$C_1$</th>
<th>$C_1^*$</th>
<th>$C_2$</th>
<th>$C_3$</th>
<th>$C_3^*$</th>
<th>$C_4$</th>
<th>$C_5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>LRR</td>
<td>1.8</td>
<td>0</td>
<td>0</td>
<td>0.8</td>
<td>0.873</td>
<td>-0.65</td>
<td></td>
</tr>
<tr>
<td>SSG</td>
<td>1.7</td>
<td>0.9</td>
<td>1.05</td>
<td>0.8</td>
<td>0.65</td>
<td>0.625</td>
<td>0.2</td>
</tr>
</tbody>
</table>

There are more complicated models which include higher order terms, however these are rarely used in an industrial situation due to the added complexity and difficulty encountered during implementation and convergence of such models in industrial CFD codes.

**Diffusion**

The viscous diffusion term $\nu \frac{\partial^2 u'_i u'_j}{\partial x^2_k}$ is only significant close to the wall. Hence when using a high Reynolds number RSM, this quantity is typically ignored. Equally, the pressure-diffusion terms $\frac{\partial}{\partial x_k} \left( \rho u'_i \delta_{jk} + \rho u'_j \delta_{ik} \right)$ are also only relevant for low Reynolds number cases. As such it is the triple moment term which is modelled. The most popular model is that of the generalised gradient diffusion hypothesis of Daly et al. [33]

$$d_{ij} = -\frac{\partial}{\partial x_k} \left( C_s \frac{k}{\varepsilon} u'_{ik} \frac{\partial u'_i u'_j}{\partial x_l} \right)$$  \hspace{1cm} (2.71)

By solving a transport equation for each of the Reynolds stresses the RSM approach provides potential improvements over the eddy viscosity approach in terms of accuracy (although EVM’s can perform well on flows that they have been calibrated for). However, due to the difficulties in their implementation, and the additional cost to computing resources, these models are still not used as widely as two-equation models in industrial computations.

**2.1.5 Near-wall modelling**

The nature of turbulent flow in the vicinity of a wall is highly complicated. If a model was originally delivered with free-stream isotropic turbulence conditions in mind then substantial modifications must be incorporated to obtain the correct behaviour as the wall is approached. An alternative approach is to derive a
model with wall effects taken into account from the start. The presence of a wall produces several effects [12]:

- Low-Reynolds number - As the wall is approached the local turbulent Reynolds number \( Re_t = k^2/(\varepsilon \nu) \) tends to zero.

- Wall blocking - The presence of a solid wall effects the pressure field up to an integral length from the wall [12].

- High shear rate - The highest shear rate \( \partial U/\partial y \) occurs near the wall.

- Two-component limit - In the vicinity of the wall \( \overline{v'}v' \) scales by \( y^4 \) whereas \( \overline{u'}u' \) and \( \overline{w'}w' \) scale as \( y^2 \).

The algebraic models discussed earlier are designed to be directly integrated down to the wall. The Van Driest damping function and also the use of a two-layer mixing layer approach make direct integration to the wall possible and make some account for the changing flow physics as a wall is approached. However the two-equation \( k – \varepsilon \) model, given in Equations 2.20 & 2.37 is not suitable for direct integration to the wall. Firstly there is the numerical problem that the term \( C_\varepsilon \varepsilon^2/k \) becomes a singularity at the wall as \( k \) tends to zero and \( \varepsilon \) is finite [23]. Secondly no explicit modification is made to constants which are calibrated away from a wall. Thirdly the turbulent viscosity is observed to be damped closer to the wall as viscous effects become more important. There are two main methods to deal with the near-wall flow, these are the ‘wall function’ approach and the ‘low-Reynolds number model’ approach. These will both be addressed in turn.

### Wall function approach

One approach to deal with the near-wall effects is to assume the flow near the wall is close to local equilibrium and acts like a fully developed turbulent boundary layer. Using this assumption the flow and its boundary conditions can be described by wall functions which aim to match known features of a zero pressure gradient boundary layer. Models which use this approach are typically referred to as ‘high-Reynolds number models’ (i.e the standard \( k – \varepsilon \) model) that have not been derived for integration to the wall.

The implicit assumption is that viscous effects are confined to a thin layer near the wall and thus can be modelled separately than the flow away from the wall.
The advantage of this approach is that the near-wall mesh can be significantly coarsened and thus the computational requirements are reduced, especially for complex geometries. The derivation for a standard wall function is based on a steady two-dimensional zero pressure gradient boundary layer. A simplified transport equation for the momentum can be derived to give:

$$\frac{\partial}{\partial y} \left( \mu \frac{\partial U}{\partial y} - \rho u'v' \right) = 0 \quad (2.72)$$

The can be integrated with boundary conditions at the wall of \( y = 0 \) and \( \rho u'v' = 0 \), to give:

$$\mu \frac{\partial U}{\partial y} - \rho u'v' = \tau_w \quad (2.73)$$

In the viscous sublayer \( (y^+ < 5) \), which is the thin region adjacent to the wall, the Reynolds stresses are assumed to be very small and thus the wall shear stress becomes: \( \tau_w = \mu (\partial U / \partial y) \). In this region the non-dimensional velocity \( U^+ = U/u_\tau = y^+ \), where \( y^+ \) is the non-dimensional wall distance: \( y^+ = yu_\tau/\nu \), where \( u_\tau = \sqrt{\tau_w/\rho} \).

In the fully turbulent region \( (y^+ > 30) \), the viscous stresses become negligible and thus Equation 2.73 becomes \( \rho u'v' = \tau_w \). The velocity derivative can be formed by using the mixing length hypothesis, where \( l = \kappa y \), to give:

$$\tau_w = \rho l^2 \left( \frac{\partial U}{\partial y} \right)^2 \quad (2.74)$$

$$\frac{\partial U}{\partial y} = \frac{u_\tau}{\kappa y} \quad (2.75)$$

which can be integrated to give:

$$U^+ = \frac{1}{\kappa} \ln y^+ + C \quad (2.76)$$

Where \( \kappa = 0.41 \) and \( C = 5.0 \).

Although there is physical justification for this approach in steady simple 2D flows, when applied to non-equilibrium flows or those subjected to separation
these justifications begin to breakdown.

**Low-Reynolds number approach**

An alternative approach to wall-functions is to adapt the transport equation for the model to include terms that can represent both the near-wall and remote flow. There have been many modifications to account for the wall, in fact the original paper of Launder & Jones [21] was a Low-Reynolds version that was suitable for integration to the wall without the use of wall functions. The form of this model is as follows:

\[
\frac{Dk}{Dt} = P_k + \varepsilon + d_k \tag{2.77}
\]

\[
\frac{D\tilde{\varepsilon}}{Dt} = C_{\varepsilon 1} f_1 P_k \tilde{\varepsilon} - C_{\varepsilon 2} f_2 \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \tilde{\varepsilon}}{\partial x_j} \right] + E \tag{2.78}
\]

Where,

\[
\tilde{\varepsilon} = \varepsilon - D \tag{2.79}
\]

\[
D = 2\nu \left( \frac{\partial \sqrt{k}}{\partial x_j} \right) \tag{2.80}
\]

\[
E = 2\nu u_t \left( \frac{\partial^2 U}{\partial y^2} \right) \tag{2.81}
\]

\[
f_1 = 1 \tag{2.82}
\]

\[
f_2 = 1 - 0.3 \exp(-Re_t^2) \tag{2.83}
\]

The turbulent viscosity is modified to included a damping function \(f_\mu\):

\[
\nu_t = f_\mu C_{\mu} \frac{k^2}{\varepsilon} \tag{2.84}
\]

Where the damping function \(f_\mu\) has the form:

\[
f_\mu = \exp \left( \frac{-2.5}{1 + Re_t/50} \right) \tag{2.85}
\]

The same problems also exist for Reynolds Stress models such as the SSG and LRR models that have been derived for use outside of the near-wall region. There are several Low-Reynolds number adaptions of these models such as the
Low-Reynolds number SSG model of Chen et al. [34]. This model has two damping functions that are active in the dissipation term $\varepsilon_{ij}$ and the pressure-strain term $\phi_{ij}$. However these are also ad-hoc modifications that lack generality and compensate for a model that does not explicitly account for the wall effects listed previously.

**Elliptic relaxation**

An alternative method to using empirically derived damping functions is the elliptic relaxation approach pioneered by Durbin [25]. This approach was initially developed to reproduce the correct wall effects in a Reynolds Stress model context, however it was also adapted for use in an eddy viscosity model. Firstly the underlining concept will be discussed with regards to the pressure-strain modelling within a RSM, and then the eddy viscosity model adaption will be discussed which is directly relevant to the $\varphi - f$ DDES model.

Elliptic relaxation is another approach to the modelling of the pressure-strain term in the near-wall region that is asymptotically correct as the wall is approached, i.e that in the vicinity of the wall $\overline{v'v'}$ scales by $y^4$ whereas $\overline{u'u'}$ and $\overline{w'w'}$ scale as $y^2$. It is based on a deeper understanding of the non-homogeneous nature of the flow as the wall is approached and how this relates the modelling of the pressure fluctuations in the pressure-strain term.

The velocity-pressure gradient correlation term from the pressure-strain term can be obtained after some steps from the Poisson equation for the fluctuating pressure to give:

$$\frac{\overline{u'_i}}{\overline{\partial p'(\mathbf{x})}} = \frac{1}{4\pi} \iiint \frac{\overline{u'_i(\mathbf{x}) \partial S(\mathbf{x}')/\partial x_j}}{|\mathbf{x} - \mathbf{x}'|} d^3 \mathbf{x}'$$  \hspace{1cm} (2.86)

where

$$S(\mathbf{x}') = \frac{\partial U_k}{\partial x_j} \frac{\partial u'_i}{\partial x_k}$$  \hspace{1cm} (2.87)

The turbulence correlation function in Equation 2.86 is assumed to be exponential [25], such that:
\[
\bar{u}'(x) (\partial S(x')/\partial x_j) = \bar{u}'(x') (\partial S(x')/\partial x_j) \exp \left(-\left|x - x'\right|/L\right)
\] (2.88)

The kernel in the integral of Equation 2.86 is the Green function for the modified Helmholtz equation. Therefore the pressure strain-rate term is the solution of the following equation:

\[
\phi_{ij} - L^2 \nabla^2 \phi_{ij} = \phi_{ij}^h
\] (2.89)

Thus Equation 2.89 acts to let \( \phi_{ij} \) relax from the value of its source term (its homogenous value through \( \phi_{ij}^h \)) to its wall value over the length scale \( L \). This therefore negates the need for a near-wall pressure strain-rate term, \( \phi_{ij}^w \).

Durbin uses a modified term for the redistribution tensor in order to model the asymptotic behaviour correctly (i.e. by making it vanish at no-slip boundaries) [35].

\[
\varphi_{ij} = \phi_{ij} + \left( \varepsilon_{ij} - \frac{\bar{u}_i \bar{u}_j}{k} \varepsilon \right)
\] (2.90)

After using an intermediate variable \( f_{ij} = \phi_{ij}/k \), the elliptic equation is of the form:

\[
f_{ij} - L^2 \nabla^2 f_{ij} = \frac{\varphi_{ij}^h}{k}
\] (2.91)

The homogenous pressure strain term \( \varphi_{ij}^h \) is modelled in Durbin’s original model [1] by Rotta’s return to isotropy model for the slow part and the isotropisation of production model for the rapid part (although the SSG and LRR described earlier have been used as alternatives). This homogeneous model is derived to model the pressure strain-term in the regions away from the wall, where \( \nabla^2 \phi_{ij} = 0 \):

\[
\varphi_{ij}^h = \frac{1 - C_1}{kT} \left( \bar{u}_i \bar{u}_j - \frac{2}{3} k \delta_{ij} \right) - \frac{C_2}{k} \left( P_{ij} - \frac{2}{3} P \delta_{ij} \right)
\] (2.92)
Where the length and time scales are given by:

\[ T = \max \left( \frac{k}{\varepsilon}, C_T \sqrt{\frac{\nu}{\varepsilon}} \right) \]  \hspace{1cm} (2.93)

\[ L = C_{L max} \left[ \frac{k^{3/2}}{\varepsilon}, C_\eta \frac{\nu^{3/4}}{\varepsilon^{3/4}} \right] \]  \hspace{1cm} (2.94)

The Kolmogorov scales are used in both the length scale and time scale limiters to prevent the formation of singularities near the wall in the viscous sublayer.

The dissipation is modelled by:

\[ \frac{D\varepsilon}{Dt} = \frac{C_{\varepsilon 1} P_k - C_{\varepsilon 2} \varepsilon}{T} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \]  \hspace{1cm} (2.95)

Where, \( \nu_t = C_\mu kT \) and \( C_{\varepsilon 1} = C_\varepsilon (1 + A_1 P/\varepsilon) \). The models constants are shown in Table 2.3 below:

<table>
<thead>
<tr>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_T )</th>
<th>( C_L )</th>
<th>( C_\eta )</th>
<th>( C_\mu )</th>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
<th>( \sigma_\varepsilon )</th>
<th>( A_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.22</td>
<td>0.6</td>
<td>6.0</td>
<td>0.2</td>
<td>80</td>
<td>0.23</td>
<td>1.44</td>
<td>1.9</td>
<td>1.65</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Table 2.3: Model coefficients for the elliptic relaxation RSM model [1]

Whilst the elliptic relaxation approach is physically more justified than ad-hoc damping function and has been shown to offer improvements on several flows [36], the additional 6 equations required for \( f_{ij} \) mean it is computationally more expensive than standard low-Reynolds number RSM approaches.

\[ f_{22} = -\frac{20 \nu^2 \overline{w'v'}}{\varepsilon^2 y^4} \quad f_{12} = -\frac{20 \nu^2 \overline{u'v'}}{\varepsilon^2 y^4} \quad f_{23} = -\frac{20 \nu^2 \overline{u'w'}}{\varepsilon^2 y^4} \]  \hspace{1cm} (2.96)

\[ f_{11} = -\frac{1}{2} f_{22} \quad f_{33} = -\frac{1}{2} f_{22} \quad f_{13} = -\frac{20 \nu^2 \overline{w'w'}}{\varepsilon^2 y^4} \]

This elliptic relaxation approach has the further disadvantage that it is more numerically stiff due to the boundary conditions for \( f_{ij} \) depending on \( y^4 \). However having a separate boundary condition for each \( f_{ij} \) allows the model to reproduce some of the asymptotic behaviours of each Reynolds stress as it approaches the wall. This feature is not available in standard RSM models that have a local
boundary condition of $\overline{u_i' u_j'} = 0$.

**Elliptic Blending Reynolds Stress Model (EB-RSM)**

The elliptic blending Reynolds Stress model (EB-RSM) \cite{37} is a simplification of the previous elliptic relaxation Reynolds Stress model of Durbin \cite{25}. It replaces the 6 $f_{ij}$ equations with a single equation that is solved for a single coefficient $\alpha$ that has the value 0 at the wall and 1 far away from the wall.

$$L^2 \nabla^2 \alpha - \alpha = -1 \quad (2.97)$$

This coefficient $\alpha$ is then used to blend between near-wall and homogenous models for the pressure strain term $\phi_{ij}$ and the dissipation $\varepsilon_{ij}$.

$$\phi_{ij} = (1 - \alpha^2) \phi_{ij}^w + \alpha^2 \phi_{ij}^h, \quad \text{near wall hom} \quad (2.98)$$

$$\varepsilon_{ij} = (1 - \alpha^2) \frac{u_i' u_j'}{k} \varepsilon + \alpha^2 \frac{2}{3} \varepsilon \delta_{ij}, \quad \text{near wall hom} \quad (2.99)$$

where the homogenous part is modelled by the quasi-linear SSG model \cite{32}

$$\phi_{ij}^h = -(C_1 \varepsilon + C_4 P_k) b_{ij} + C_2 \varepsilon (b_{ik} b_{kj} - \frac{1}{3} \delta_{ij} b_{kl} b_{kl}) + (C_3 - C_3^* \sqrt{b_{kl} b_{kl}}) k S_{ij} + C_4 k (b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{lm} S_{lm} \delta_{ij}) + C_5 k (b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik}), \quad (2.100)$$

where
\[ b_{ij} = \frac{a_{ij}}{2k}, \quad (2.101) \]

\[ a_{ij} = u_i'u'_{j} - \frac{2}{3}k\delta_{ij}, \quad (2.102) \]

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right), \quad (2.103) \]

and the near-wall part is modelled by:

\[ \phi_{ij}^w = -5\varepsilon k \left( \frac{u_i'u'_{j}n_jn_k + u_j'u'_{k}n_i n_k - \frac{1}{2}u_k'u'n_kn_i(n_i n_j + \delta_{ij})}{k} \right), \quad (2.104) \]

where \[ n = \nabla \alpha / \| \nabla \alpha \| \] is a generalised wall-normal vector.

The turbulent diffusion and pressure diffusion terms are approximated by the generalised gradient diffusion hypothesis, GGDH [33], which can be written as:

\[ D_T^{ij} = \frac{\partial}{\partial x_k} \left( C_d \frac{u'_k u'_l T}{\partial x_l} \right) \quad (2.105) \]

Where \[ C_d = 0.21 \] and the time scale, \[ T \], is given by Equation \[ 2.106 \]

\[ T = \max \left( \frac{k}{\varepsilon}, C_T \sqrt{\frac{\nu}{\varepsilon}} \right), \quad (2.106) \]

The model constants have the following values for the EB-RSM model:

<table>
<thead>
<tr>
<th>( C_1 )</th>
<th>( C_1^* )</th>
<th>( C_2 )</th>
<th>( C_3 )</th>
<th>( C_3^* )</th>
<th>( C_4 )</th>
<th>( C_5 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.7</td>
<td>0.9</td>
<td>1.05</td>
<td>0.8</td>
<td>0.65</td>
<td>0.625</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Table 2.4: Model coefficients for the EBRSM model

Where the length scale, \( L \), is defined as:

\[ L = C_L \max \left( \frac{k^{3/2}}{\varepsilon}, C_\eta \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \right). \quad (2.107) \]

The turbulent dissipation is approximated by a low-Reynolds number version of the transport equation for \( \varepsilon \).
\[ \frac{D\varepsilon}{Dt} = C'_{\varepsilon 1} \frac{P - C_{\varepsilon 2} \varepsilon}{T} + \nu \frac{\partial^2 \varepsilon}{\partial x_k \partial x_k} + \frac{\partial}{\partial x_k} \left( C'_{d} \overline{u_k u'_l} T \frac{\partial \varepsilon}{\partial x_l} \right), \]  

(2.108)

where

\[ C'_{\varepsilon 1} = C_{\varepsilon 1} \left( 1 + A_1 (1 - \alpha^2) \sqrt{\frac{k}{u_i u_j n_i n_j}} \right). \]  

(2.109)

The remaining model constants are:

<table>
<thead>
<tr>
<th>( C_T )</th>
<th>( C_L )</th>
<th>( C_\eta )</th>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
<th>( C'_{d} )</th>
<th>( A_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.0</td>
<td>0.161</td>
<td>80.0</td>
<td>1.44</td>
<td>1.83</td>
<td>0.21</td>
<td>0.03</td>
</tr>
</tbody>
</table>

Table 2.5: Further model coefficients for the EBRSM model

The elliptic blending model can be seen as a simplified version of the original elliptic relaxation method of Durbin, but one that is more numerically robust and computationally less expensive.

\( \overline{v^2} - f \) model

The elliptic relaxation RSM model can be simplified for use in an eddy viscosity model, as shown by Durbin [25] in the \( k - \varepsilon - \overline{v^2} \) model (more commonly referred to as the \( \overline{v^2} - f \) model). In this model the standard \( k - \varepsilon \) model is solved alongside a third equation for the scalar \( \overline{v^2} \) which for a fully developed channel can be thought of as representing the wall-normal fluctuations. Durbin showed that it is the wall-normal fluctuations rather than the turbulent kinetic energy \( k \) that is the most appropriate velocity scale to use to damp \( \nu_t \) near the wall.

A simplified version of the equations for \( \overline{v^2} \) and \( f_{22} \) from the full RSM model is solved alongside the standard \( k \) and \( \varepsilon \) equations:
\[ \frac{Dk}{Dt} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right] \]  \hspace{1cm} (2.110)

\[ \frac{D\varepsilon}{Dt} = \frac{C_{e1} P_k - C_{e2} \varepsilon}{T} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] \] \hspace{1cm} (2.111)

\[ \frac{Dv^2}{Dt} = k f - \nu^2 \frac{\varepsilon}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial v^2}{\partial x_j} \right] \] \hspace{1cm} (2.112)

\[ L^2 \frac{\partial^2 f}{\partial x_j^2} - f = \frac{1}{T} \left( C_1 - 1 \right) \left[ \frac{v^2}{k} - \frac{2}{3} \right] - C_2 \frac{P_k}{k} \] \hspace{1cm} (2.113)

The boundary condition for \( f \) is the same as for the full RSM \( f_{22} \), which ensures the correct asymptotic behaviour \( v^2 = O(y^4) \):

\[ f = -\frac{20v^2v^2v^2}{\varepsilon^2y^4} \] \hspace{1cm} (2.114)

The turbulent viscosity is adapted to incorporate these changes, such that the form is now:

\[ \nu_t = C_\mu \nu^2 T \] \hspace{1cm} (2.115)

Over the past 15 years there have many versions of the \( v^2 - f \) model that have sought to improve its numerical robustness as well as its predictive ability. It is beyond the scope to discuss these developments, however the reader is advised to read an excellent summary in the doctoral thesis of Billard [38]. The version of the \( v^2 - f \) model that has been used throughout the course of this work will now be presented.

\( \varphi - f \) model

The \( \varphi - f \) model [39] is a numerically robust version of the ‘Code Friendly’ \( v^2 - f \) model developed by Lien & Durbin [40]. In this version, \( v^2 \) is replaced by \( \varphi = \frac{v^2}{k} \), which results in a much more robust model that converges more easily and allows the use of larger time steps compared to the model of Durbin. The model uses the same transport equation for the turbulent kinetic energy \( k \), as the high Reynolds number \( k - \varepsilon \) model [21], but uses a slightly modified equation
for the turbulent dissipation $\varepsilon$, that incorporates a modified $C_{\varepsilon_1}$ term to allow a different value based upon $\varphi$ close to the sublayer. A transport equation is also solved for $\varphi$ and the elliptic operator $f$.

\[
\frac{Dk}{Dt} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \tag{2.116}
\]

\[
\frac{D\varepsilon}{Dt} = \frac{C_{\varepsilon_1} P_k - C_{\varepsilon_2} \varepsilon}{T} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \tag{2.117}
\]

\[
\frac{D\varphi}{Dt} = f - P_k \frac{\varphi}{k} + 2 \left( \frac{\nu_t}{k} \right) \frac{\partial \varphi}{\partial x_j} \frac{\partial k}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu_t}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j} \right] \tag{2.118}
\]

\[
L^2 \frac{\partial^2 f}{\partial x_j^2} - f = \frac{1}{T} (C_1 - 1) \left[ \varphi - \frac{2}{3} \right] - C_2 \frac{P_k}{k} - 2 \left( \frac{\nu}{k} \right) \frac{\partial \varphi}{\partial x_j} \frac{\partial k}{\partial x_j} - \nu \frac{\partial^2 \varphi}{\partial x_j^2} \tag{2.119}
\]

The turbulent viscosity can be defined as:

\[
\nu_t = C_\mu \varphi kT \tag{2.120}
\]

where the time scale, $T$, and the length scale, $L$, are bounded by the Kolmogorov scales in order to avoid a singularity occurring in Equation (2.119) and also for the turbulent viscosity (Equation 2.120):

\[
T = \max \left( \frac{k}{\varepsilon}, C_T \sqrt{\frac{\nu}{\varepsilon}} \right) \tag{2.121}
\]

\[
L = C_L \max \left[ \frac{k^{3/2}}{\varepsilon}, C_\eta \frac{\nu^{3/4}}{\varepsilon^{1/4}} \right] \tag{2.122}
\]

The term $C_{\varepsilon_1}$ in Equation (2.117) is changed from the original value of 1.44 used in the standard $k - \varepsilon$ model to:

\[
C_{\varepsilon_1} = 1.4 \left( 1 + 0.05 \sqrt{\frac{1}{\varphi}} \right) \tag{2.123}
\]

The model constants for the $\varphi - f$ model are shown in Table 2.6 below:
The $\varphi - f$ model offers a more physically justified near-wall modelling strategy than those models such as the SST and SA models that use empirically derived damping functions to match the near-wall conditions. This model has been shown to give improved results over the standard $k-\varepsilon$ and SST models for an asymmetric plane diffuser and periodic hills [36], and it has also shown promising results within a novel hybrid RANS-LES approach [41]. It was for these reasons that it was selected to be used as an alternative RANS model in the DDES approach that will discussed later in detail.

### Table 2.6: Model coefficients for the $\varphi - f$ model

<table>
<thead>
<tr>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$C_L$</th>
<th>$C_u$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>1.3</td>
<td>1.9</td>
<td>1.4</td>
<td>0.3</td>
<td>0.3</td>
<td>0.22</td>
<td>100</td>
</tr>
</tbody>
</table>

The $\varphi - f$ model offers a more physically justified near-wall modelling strategy than those models such as the SST and SA models that use empirically derived damping functions to match the near-wall conditions. This model has been shown to give improved results over the standard $k-\varepsilon$ and SST models for an asymmetric plane diffuser and periodic hills [36], and it has also shown promising results within a novel hybrid RANS-LES approach [41]. It was for these reasons that it was selected to be used as an alternative RANS model in the DDES approach that will discussed later in detail.
2.2 Large Eddy Simulation (LES)

An alternative approach to solving the instantaneous Navier-Stokes equations is Large Eddy Simulation (LES), in which a filtering operation is conducted which separates the smallest turbulent scales from the largest. This is based upon the assumption that the large scales in a turbulent flow are those containing the most energy, are anisotropic and are dependant on the flow geometry and thus should be resolved. The small scales can be thought as being more isotropic, dissipative and not as dependant on the flow geometry, and as such can be modelled. This means the mesh can be coarser and thus allows a significant saving on mesh resolution compared to DNS.

The process of scale separation is achieved by applying a filter to the velocity field, where the instantaneous velocity field is split into a resolved and residual part:

\[ u(x, t) = \hat{U}(x, t) + u'(x, t) \] (2.124)

This filtering procedure can be achieved with different types of filters but in practice the maximum grid size is commonly used as the filter which is why the modelled component is typically called the ‘sub-grid’ component.

Applying this filter to the momentum equations leads to:

\[ \frac{\partial \hat{U}_i}{\partial t} + \frac{\partial \hat{U}_i \hat{U}_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \hat{p}}{\partial x_i} + \frac{\partial \hat{\tau}_{ij}}{\partial x_i} \] (2.125)

It is important to note that because the product of the filtered velocities is not the same as the filter of the product of the velocities:

\[ \hat{U}_i \hat{U}_j \neq \hat{U}_i \hat{U}_j \] (2.126)

then a modelling approximation must be introduced:

\[ \hat{U}_i \hat{U}_j = \hat{U}_i \hat{U}_j + \hat{u}_i \hat{u}_j \] (2.127)

Applying this to Equation 2.125 gives:
\[
\frac{\partial \hat{U}_i}{\partial t} + \frac{\partial \hat{U}_i \hat{U}_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \hat{p}}{\partial x_i} + \frac{\partial \hat{\tau}_{ij}}{\partial x_i} - \hat{u}_i' \hat{u}_j' \quad (2.128)
\]

To close the equations, a model is required for the sub-grid scale stress, \(\tau^s_{ij}\), using the filtered(resolved) velocities. A simple model based on a similar concept to the Boussinesq approximation was introduced by Smagorinsky [10]:

\[
- \hat{u}_i' \hat{u}_j' = 2 \nu_{SGS} \hat{S}_{ij} - \frac{2}{3} k_{SGS} \delta_{ij}, \quad (2.129)
\]

The sub-grid scale viscosity, \(\nu_{SGS}\) is modelled in a similar way to the mixing length model, by assuming it is proportional to the length scale and filtered strain. The length scale is calculated using the filter width \(\Delta\) and \(k_{SGS}\) is the sub-grid scale energy.

\[
\nu_{SGS} = (C_s \Delta)^2 \hat{S}^* \quad (2.130)
\]

Where \(C_s\) is the Smagorinsky constant, \(\hat{S}^* = \sqrt{2 \hat{S}_{ij} \hat{S}_{ij}}\), and the filter width is \(\Delta = 2 (\Delta_x \Delta_y \Delta_z)^{1/3}\). Lilly [42] assumed an inertial range specturm and deduced a value of \(C_s = 0.17\), however Deadroff [43] found that in the presence of a wall the value should be reduced and typically values of \(C_s = 0.065\) are used.

Germano [44] proposed an alternative method of calculating \(C_s\) which dynamically computes the coefficient by defining a test filter whose width is larger than the grid filter. Other sub-grid models have been developed, and the reader is referred to the work of Pope [12] for a fuller explanation.

Although LES offers a reduction in mesh resolution compared to DNS, where the entire range of turbulent scales are resolved, it is still necessary to ensure that the mesh is of sufficient quality. There have been several studies to establish best-practice guidelines for mesh design and resolution for LES. These have typically focussed on classical flows such as channel flows and boundary layers, and thus can not be directly translated to more complex geometries. However they do offer useful insight into the design of an appropriate mesh for LES.

A frequently used measure is the stream-wise, span-wise and wall-normal cell sizes in wall units. There has been some variation within different publications for the exact values however a common guide is \(y^+ < 2\), \(\Delta x^+ = 50 - 150\),
As these guidelines are typically based on simple flows with high-order numerics, some allowances have to be made when applied to different codes and flows.

A measure of the suitability of the grid to resolve the structures in the interior of the flow is the ratio $\Delta/\eta$, where $\Delta$ is the filter width and $\eta$ is an estimate of the Kolmogorov length scale ($= (\nu^3/\varepsilon)^{1/4}$). The dissipation rate can be taken from the modelled dissipation equation, $\varepsilon$ if there is a lack of a turbulence energy-budget (in a similar fashion to Šarić et al. [46]). As reported in Fröhlich et al. [47] the ideal value for this ratio for a LES is $= \Delta/\eta < 12$, in order that a suitable amount of the dissipation range of the spectrum is resolved.

A further indicator for the resolution of the mesh is the ratio of the modelled turbulent viscosity to the molecular viscosity $\nu_t/\nu$. This gives an indication of the ratio of the modelled and resolved contributions to the dissipation [47]. It is difficult to assign one number as a guideline, however $\nu_t/\nu < 1$ assures that the sub-grid scale model provides more of the dissipation than that associated with the resolved turbulence [47].

Another important consideration is the use of an accurate numerical scheme that ensures little or no additional numerical dissipation. The assumption in the filtering operation is that the sub-grid scale model accounts for all of the dissipation and thus in order to be fully accurate, no additional dissipation should arise from the numerical scheme. This typically means the use of a central differencing scheme (CDS) rather than low-order upwinding which is typically used for RANS computations.

This is not an exhaustive list of the requirements for a proper LES, for a more comprehensive review the reader is advised to refer to Sagaut et al. [48].
2.3 Hybrid RANS-LES

Introduction

The choice between RANS and LES methods is dependant on many factors, such as the accuracy required or the physical quantities that are of interest. For flows that are largely steady and for which only mean quantities are of interest (such as an airfoil at a low angle of attack), RANS modelling is often a suitable and cheap choice. Indeed in many cases where ‘engineering accuracy’ is only required (i.e within 10% of an integral force, e.g drag force) or just to capture general trends, then standard linear eddy viscosity models are often deemed adequate, and much simplify the process of mesh creation and lower the computational resources required. Many large aerospace and automotive companies still use steady linear RANS models as the backbone of their CFD process precisely because of this.

However for complex flows that contain regions of large separation or strong anisotropy, and where information about more than mean quantities are of interest (such as aero-acoustics analysis) then traditional RANS techniques often fail. For many cases their inability to capture the large scale unsteadiness and a tendency to under-predict the shear-stress in the separated layer (which leads to a longer separation length) mean they are unsuitable for these types of flows [6]. Even more advanced Reynolds Stress models that are able to account for the anisotropy in the flow are often unable to provide an good enough description of the unsteady flow field for highly separated flows [49].

Whilst LES models can in general provide a much better alternative to RANS modelling for unsteady flows, they do so at a much higher cost, so much higher that for high-Reynolds numbers flow these costs are too great for general purpose calculations. In order to satisfy the mesh requirements that were outlined earlier in this work, often unfeasibly large computational resources are required [5]. As the Reynolds number increases, the size of the smallest eddies decrease and therefore so must the grid and time step to account for this. For many engineering problems in the aerospace and automobile industry the computational resources are simply too great.

However for many flows, the domain itself can be split up into different regions which have different properties in terms of the complexity of the flow physics and the areas that are of interest to the end user. This approach is common
to even RANS computations in industry and academia which typically results in greater mesh density in regions that contain the most ‘complex’ physics or that are influential on the final quantity that is being measured. Examples of this are airfoils at high angles of attack and cylinders where the area of greatest interest is the separation region behind the trailing edge/cylinder. From a RANS perspective it would be wasteful and pointless to have the same mesh density in front of the cylinder as to the rear of the cylinder. Although it is important to have a suitable grid density to obtain the correct boundary layer and transition and separation points (depending on the Reynolds number) one would expect a greater mesh density behind the cylinder where the shear layer is formed and vortices are shed. Similarly for an airfoil while the flow is attached and the pressure gradient is favourable on the front portion of the airfoil the grid density is coarser than towards the trailing edge where the flow starts to separate.

With this analogy in mind it can be argued that rather than performing a RANS simulation over the whole domain and suffering a possible loss in accuracy at the trailing edge of a airfoil, or performing a LES over the whole domain and suffering the extra computational cost of a much finer grid, a hybrid approach could be taken. This would solve a RANS model in regions where the flow is attached, more simple to model and thus requires a coarser grid and then switch to a LES simulation in the regions where the RANS model would perform poorly such as the separation region behind the body. Even though the derivation of the Navier-Stokes equations for RANS and LES formulations are different, the end result is of the same form when an eddy viscosity is used to model the Reynolds or sub-grid scale stresses respectively [50]. As the CFD code has no knowledge of this derivation, this commonality makes hybrid RANS-LES approaches easy to implement into existing codes.

One important difference between hybrid RANS-LES approaches is how to specify the RANS and LES regions. One could either explicitly state that a certain portion of the flow is RANS and other is LES (a zonal approach), or let a quantity of the flow or mesh decide automatically and seamlessly where a RANS or LES approach should be used (non-zonal). The advantages of a non-zonal approach is its universality and that no prior knowledge of the flow (although as these methods are often based on the grid resolution directly or indirectly, some knowledge is still required). They also require far less user input and as such are more attractive to industry. One disadvantage however is that as they
are often based on some sensing parameter; such as the grid density or some measure of the unsteadiness of the flow. If this is incorrectly predicted, then the resultant activation of RANS or LES content can be incorrect. On the other hand, one advantage of a zonal approach is that the user has complete control of the activation of RANS and LES content, which suits simulations where the flow is predictable and known. One of the disadvantages however is the interface between the zones and the inlet conditions that must be applied. How to sustain or disperse resolved content in the LES and RANS regions is an important area of research.

**Detached-Eddy Simulation (DES)**

Arguably one of the first methods to pioneer this hybrid approach was Detached-Eddy Simulation (DES) \[51\]. This approach is based on splitting the domain into RANS and LES regions based on the grid resolution. DES can be seen as a three-dimensional, unsteady model based on an underlying ‘off-the-shelf’ RANS model. It seamlessly acts as a sub-grid scale model in regions where the grid is fine enough to support a LES, and as a RANS model in areas where it is not. The main advantage of this approach is that it is conceptually easy to understand and mathematically simple because it only requires the modification of a single term. In DES, the RANS turbulent length scale is adapted to Equation 2.131 which effectively changes the turbulent length scale to one based on the grid resolution. For the original SA-DES model this results in every instance of the turbulent length scale being substituted for the new definition, as shown in Equation 2.133. Although DES was originally used with the Spalart-Allmaras RANS model, it can conceptually be applied to any RANS model \[52\].

\[
L_{DES} = \min(L_{RANS}, L_{LES}) \quad (2.131)
\]
\[
L_{LES} = C_{DES} \Delta \quad (2.132)
\]

Where \(L_{RANS}\), \(L_{LES}\) and \(L_{DES}\) represent the RANS, LES and combined DES turbulent length scales respectively. \(\Delta\) is the LES filter width which is typically taken as the cell volume for unstructured codes, although other expressions such as the maximum cell size are sometimes used \[53\]. \(C_{DES}\) is an empirical parameter
that needs to be tuned to match the correct level of dissipation, typically $C_{DES} = 0.60$ but this constant is highly dependant on the numerics of the code and the underlying RANS model, and thus must be tuned for any new formulations.

\[
\frac{D\bar{\nu}}{Dt} = c_b 1 \tilde{\nu} - c_w f_w \left( \frac{\tilde{\nu}}{L_{DES}} \right)^2 + \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( (\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right] \quad (2.133)
\]

The original intention of DES [51] is to enable the boundary layers and attached steady flow to be modelled in RANS mode, i.e $L_{RANS} < L_{LES}$, and then for the separated unsteady flow away from the boundary layer to be resolved in LES mode, i.e $L_{LES} < L_{RANS}$. As the LES length scale is based on the grid resolution, this requires the mesh to be suitably designed to have a finer cells in the proposed LES region and coarser cells in the RANS regions.

**DES improvements and modifications**

Since the inception of DES, several groups found that a potential error was possible when the near-wall grid refinement was too great. This meant that the model turned to its LES mode within the attached boundary layer which lowered the turbulent viscosity and for extreme cases triggered too early separation. This was termed grid-induced separation (GIS) [54] and was caused by the modelled turbulence level dropping but no resolved content was available to replace it (also often referred to as Modelled-Stress Depletion(MSD)).

In order to stop this from occurring, regardless of the mesh resolution in the boundary layer, a boundary layer ‘shield’ was integrated into the DES length scale equation to enforce the RANS mode in the attached boundary layers. This was originally achieved for the SST-DES model using the SST $F_1$ and $F_2$ limiters [55] but was later generalised for any RANS model, which became known as Delayed Detached-Eddy Simulation (DDES) [7] as shown below:

\[
f_d = 1 - \tanh \left( |8r_d|^3 \right), \quad (2.134)
\]

where
\[ r_d = \frac{\nu_t + \nu}{\sqrt{U_{ij}U_{ij}\kappa^2y^2}}, \]  

(2.135)

and, \( \kappa \) is the Karman constant, and \( y \) is the distance to the wall. \( r_d \) equals 1 in the boundary layer and gradually reduces to 0 towards the edge of the boundary layer. The function \( f_d \) is designed to be 1 in the LES region where \( r_d \ll 1 \), and 0 elsewhere.

This modification means that \( L_{DDES} \) is now redefined to:

\[ L_{DDES} = L_{RANS} - f_d \max(0, L_{RANS} - C_{DDES}\Delta) \]  

(2.136)

This new DDES formulation was evaluated in the DESider [6] and ATAAC [56] projects and has now made the original DES version obsolete.

A further modification made through the work of the FLOMANIA [49] and DESider projects was to redefine the length scale \( L_{DDES} \) to counter-act the activation of the SA damping terms when the turbulent viscosity became too low [7]. The SA RANS model contains several low-Reynolds number terms that were designed to be activated in the near-wall region. The model uses the turbulent viscosity ratio \( \nu_t/\nu \) to ‘sense’ the proximity to the wall, so that a low value of this ratio would activate these damping terms which further decrease the turbulent viscosity ratio in the presence of a wall. Unfortunately an unexpected consequence of this was that when the SA-DES or SA-DDES models were in LES mode, in some situations, the turbulent viscosity ratio became so low that it triggered these low-Reynolds number damping terms and excessively lowered the turbulent viscosity ratio.

To solve this the DES length scale was altered to include a correction term \( \psi \) that compensates for this erroneous activation, as seen in Equation 2.138 (the full derivation can be found in [7]). The SST-DES model however does not require this modification.

\[ L_{DDES} = L_{RANS} - f_d \max(0, L_{RANS} - C_{DDES}\Delta) \]  

(2.137)

\[ \psi = \sqrt{\min\left(100; \frac{1}{f_v^1} - \frac{f_v^2 c_{h1}}{f_w f_v^1 \kappa^2 c_{w1}}\right)} \]  

(2.138)
The grey area problem

One problem common to all DES and DDES formulations is the so-called ‘grey area’ problem. This is effectively the development region between the RANS and LES zones where the transition between modelled and resolved turbulence takes place. It is most clearly identified for a turbulent boundary layer that separates to form a free shear layer \[^{57}\]. In this example, the attached boundary layer is treated in RANS mode and therefore the resolved turbulence is very low compared to the modelled contribution. As the flow separates and forms a separated shear layer, the model switches to its LES mode, thus the modelled turbulence drops as the turbulent length scale is reduced ($L_{LES} < L_{RANS}$). However as there is no resolved turbulence upstream of the LES zone, there is a ‘grey area’ where the resolved turbulence has to form, and at this point the turbulence is neither fully modelled nor fully resolved. The extent and length of this region is not only case-sensitive but also sensitive to the underlying RANS model and the numerics of the code (i.e. the level of additional numerical dissipation).

Introducing synthetic turbulence at the LES interface can speed up this transition from modelled to resolved turbulence, but its strength and position can have a wide ranging effect on the solution. Thus some models have re-calibrated the $f_d$ function of DDES to limit the the extent of the RANS region in the separated shear layer \[^{58}\] and others to re-define the filter width \[^{59}\] to achieve a similar affect.

The grey area problem remains a challenge for hybrid RANS-LES models and limits the use of standard DDES in weakly separated flows where the transition from modelled to resolved turbulence content is often seen to be at its slowest \[^{57}\].

Partially Averaged Navier-Stokes (PANS)

Another approach to hybrid RANS-LES modelling is to ‘partially average’ the Navier-Stokes equations which corresponds to a filtering operation for a portion of the fluctuating scales \[^{60}\]. Two approaches that are based upon this concept are Partially Integrated Transport Modelling (PITM) \[^{61}\] and the Partially Averaged Navier-Stokes (PANS) model \[^{62}\]. The purpose of these approaches is to provide a model that can smoothly simulate turbulent flows using RANS to DNS in...
a single approach. Although the derivation of both approaches take different paths, the end result is a set of transport equations that hold the same form as the underlying RANS equations, but with different model coefficients. This is particularly attractive because it makes the approach easy to implement and couple with different underlying RANS models.

By adjusting a constant (defined as the ratio of the modelled to resolved energy, \(f_k\)) from \(f_k = 0\) to \(f_k = 1\) the model moves from a DNS approach where the entire energy spectrum is resolved (and the eddy viscosity tends to zero) to a fully RANS approach when \(f_k = 1\) (and the whole spectrum is modelled with an eddy viscosity). This is illustrated in Equation 2.139 for a standard \(k - \varepsilon\) PANS model.

\[
\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} P_k \frac{\varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \nabla \cdot \left[ \left( \nu + \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] \tag{2.139}
\]

\[
C_{\varepsilon 2} = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} (C_{\varepsilon 2} - C_{\varepsilon 1}) \tag{2.140}
\]

where \(f_\varepsilon = 1\), \(C_{\varepsilon 1} = 1.5\), \(C_{\varepsilon 2} = 1.9\) and \(\sigma_{\varepsilon u} = \sigma_\varepsilon \frac{f_k^2}{f_\varepsilon}\), where \(\sigma_\varepsilon = 1.4\).

As \(f_k\) decreases from one to zero, the destruction term in the dissipation equation also decreases which means the modelled turbulent dissipation increases and the turbulent viscosity reduces. The effect of this is to smoothly control the amount of turbulence that is resolved.

These two approaches have been extensively studied over the past decade and extended with more advanced near-wall EVM models \[60\], including an elliptic relaxation based PANS model \[63\] and an EB-RSM based PITM model \[64\] that have both shown promising results. More recently the PANS approach has also been used within an embedded LES approach, where the PANS model is used in both the RANS region (where \(f_k = 1\)) and in the LES region where it was set to \(f_k = 0.4\) \[65\].

**Scale-Adaptive-Simulation (SAS)**

An alternative seamless or non-zonal approach is Scale-Adaptive-Simulation (SAS) \[66\]. The principle is similar to DES, which is to reduce the turbulent viscosity in regions of unsteady flow to allow this region to be resolved and not damped by the turbulence viscosity. Whereas DES alters the dissipation term in the turbulent kinetic energy equation, the SAS method is based upon an extra
production term in the turbulent dissipation equation that boosts the production of dissipation and therefore lowers the turbulent viscosity in unsteady regions.

This term is based on the ratio of the modelled turbulent length scale to the von Karman length scale Equation 2.141. The von Karman length scale is the ratio of a first velocity derivate to a second velocity derivative. As the second derivate can be seen as the change in the first derivative over a given time, for a steady flow this second derivate is small. However for an unsteady flow where the first derivative would be changing over time, then the second derivate increases, thereby decreasing the ratio, as the second derivate is the denominator. This has the effect of increasing the SAS term (as the von karman length scale is the dominator), which increases the dissipation and in turn also decreases the turbulent viscosity (as the dissipation is the denominator). The destruction term in the turbulent kinetic energy equation also increases for the case of the SST model implementation.

This means that the SAS modification results in the modelled component of the turbulence becoming lower when the flow is detected to be unsteady (via the von karman length scale) and therefore more of the flow can be resolved in a LES fashion.

The most widely used implementation of the SAS term is the SST-SAS model, although in principle it can be included in any RANS model.

The SST-SAS model [66] is based on the standard SST model [67]. The SAS source term is added to the turbulent length-scale determining equation for $\omega$.

This term, $Q_{SAS}$, shown in Equation 2.141 enables the turbulent dissipation to be adapted to ensure that the eddy viscosity is reduced in regions of highly unsteady and/or separated flow, in a similar way to DES [7].

The advantage of using this approach over DES is that there is no explicit link to the grid size, and is therefore not prone to grid-induced separation or other grid-related issues that affect DES.

$$Q_{SAS} = \max \left[ \rho_\kappa S^2 \left( \frac{L}{L_{\nu K}} \right)^2 - C \cdot \frac{2\rho k}{\sigma_\phi} \max \left( \frac{|\nabla \omega|^2}{\omega^2}, \frac{|\nabla k|^2}{k^2} \right), 0 \right] \quad (2.141)$$

Where the model constants are:

$$\zeta_2 = 3.51 \ , \ \sigma_\phi = \frac{2}{3} \ , \ C = 2 \quad (2.142)$$
There are two turbulent length scales used in the SAS model, these are the length scale of the modelled turbulence, $L$, which is the standard length scale used in the majority of eddy-viscosity based models, and also an additional von Karman length scale, $L_{vK}$. The von Karman length scale used in the model is a 3D version of the original 2D boundary layer definition and sensitizes the model to the changing turbulence length scales.

$$L = \frac{\sqrt{k}}{c_{\mu}/4 \cdot \omega} \quad (2.143)$$

$$L_{vK} = \frac{\kappa S}{|\nabla^2 \mathbf{U}|} \quad (2.144)$$

Where the magnitude of the velocity Laplacian is:

$$|\nabla^2 \mathbf{U}| = \sqrt{(\nabla^2 U)^2 + (\nabla^2 V)^2 + (\nabla^2 W)^2} \quad , \quad \nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2} \quad (2.145)$$

$S$ is the scalar invariant,

$$S = \sqrt{2S_{ij}S_{ij}} \quad \kappa = 0.41$$

A limiter is required for the von Karman Length scale, $L_{vK}$, to ensure that the eddy viscosity, $\nu_t$, does not drop below, $L_{vK}^{\text{min}}$, which is representative of the range of values that a LES would achieve.

$$L_{vK} = \text{max} \left( \frac{\kappa S}{|\nabla^2 \mathbf{U}|}, L_{vK}^{\text{min}} \right) \quad (2.146)$$

$$L_{vK}^{\text{min}} = C_S \cdot \sqrt{\frac{\zeta_2 \kappa}{(\beta/c_{\mu}) - \alpha}} \cdot \Delta \quad (2.147)$$

The scale-adaptive-simulation formulation is not restricted to the SST model and can be coupled with a Reynolds stress transport model \cite{68} amongst others. It has applied to a wide variety of flows with varied success \cite{50, 69, 70}, which
has often been dependant on the nature of the flow separation in question (i.e geometry or pressure induced).

**Wall-modelled LES (WMLES)**

A further modification or extension of DDES is its application to wall-modelled LES computations. For these simulations the RANS model is used to model a thinner near-wall region, which is smaller than the boundary layer. One such model is Improved Delayed Detached-Eddy simulation (IDDES) [71] which is based on the original DDES model but with a new definition for the filter width to include the wall-normal distance and also a new ‘branch’ of DDES for WMLES.

The filter width $\Delta$ is redefined as shown in Equation 2.148 to include the wall-distance $d_w$. The purpose of this is to reduce the filter width near the wall which in turn lowers the turbulent viscosity and resolves more of the boundary layer. This also helps to address the ‘grey area’ problem by speeding up the transition to resolved turbulence.

$$
\Delta = \min\left(\max\left[C_w d_w, C_w h_{\text{max}}, h_{wn}\right], h_{\text{max}}\right)
$$

Where $C_w = 0.15$ is an empirical constant and $h_{wn}$ is the grid step in the wall-normal direction. This is a fundamental shift from the DDES filter width and introduces some amount of empiricism due to the definition of the wall distance (which can be complicated to compute for complex geometries) and the constant $C_w$.

The turbulence length scale is replaced in a similar fashion to DDES and is of the form:

$$
L_{IDDES} = \tilde{f}_d(1 + f_c)L_{RANS} + (1 - \tilde{f}_d)L_{LES}
$$

Where $L_{LES}$ and $L_{RANS}$ represent the LES and RANS length scales respectively. The blending function $\tilde{f}_d$ decides between the standard DDES blending function $f_d$ and the IDDES WMLES function $f_B$, as shown below:
\[ f_d = \max[(1 - f_{dt}), f_B] \] (2.150)

Where \( f_{dt} \) is the DDES blending function \( f_{dt} = 1 - \tanh[(8r_{dt})^3] \). The blending function \( f_B \) is of the form:

\[ f_B = \min[2\exp(-9\alpha^2), 1.0] \] (2.151)

Where \( \alpha = 0.25 - d_w/h_{\text{max}} \), which ensures that the model rapidly switches from RANS to LES model within the wall distance \( 0.5h_{\text{max}} < d_w < h_{\text{max}} \) [71].

The other blending function in Equation [2.149] is \( f_e \) which is termed the elevating function. The purpose of this is to prevent the excessive reduction of the RANS Reynolds stresses in the vicinity of the interface between the RANS and LES regions [71]. It is of the form:

\[ f_e = \max[(f_{e1} - 1), 0]\Psi f_{e2} \] (2.152)

Where the function \( f_{e1} \) is defined as:

\[ f_{e1}(d_w/h_{\text{max}}) = \begin{cases} 
2\exp(-11.09\alpha^2) & \text{if } \alpha \geq 0 \\
2\exp(-9.0\alpha^2) & \text{if } \alpha < 0
\end{cases} \] (2.153)

This function is dependant on the grid resolution through \( \alpha \) and as such needs some form of tuning. The function \( f_{e2} \) is defined as:

\[ f_{e2} = 1.0 - \max[f_t, f_l] \] (2.154)

This function controls the intensity of the increase of the RANS component. The functions \( f_t \) and \( f_l \) are defined:

\[ f_t = \tanh[(c_t^2r_{al})^3] \] (2.155)

\[ f_l = \tanh[(c_l^2r_{al})^{10}] \] (2.156)
Where $f_t$ and $f_l$ represent the turbulent and laminar version of the DDES $r_d$ function, and are defined as:

\[ r_{dt} = \frac{\nu_t}{\sqrt{U_{i,j} U_{i,j} k^2 y^2}} \]  
\[ r_{dl} = \frac{\nu}{\sqrt{U_{i,j} U_{i,j} k^2 y^2}} \]  

$c_t$ and $c_l$ are model constants that need to be tuned depending on the underlying RANS model (in a similar fashion to DDES), such that $f_{e2}$ is zero when $r_{dt}$ or $r_{dl}$ are close to 1.0 [71].

Thus the purpose of IDDES is to switch between standard DDES and a wall-modelled LES version depending on the near-wall grid resolution and the state of the inflow turbulence. The large amount of model constants and blending functions however make this model highly tuned to well designed near-wall meshes and a particular set of flows. This model lacks some of the simplicity that has made DES and DDES one of the most preferred hybrid RANS-LES models, especially in commercial CFD codes.

**Zonal methods**

Zonal hybrid RANS-LES methods split the domain into specific RANS and LES regions either explicitly or based on a non-dimensional unit such as the wall distance. The primary advantage of this method is the ability to decide before the simulation which areas will be treated in each mode regardless of the grid resolution. This method is well suited to flows that are largely known and which might be difficult to setup for seamless methods that are based on the grid resolution (e.g. automobiles or helicopters). Typically at the interfaces between the RANS and LES zones synthetic turbulence is introduced to trigger the formation of resolved turbulence at the LES interface. One zonal approach based upon DES is the zonal Detached-Eddy Simulation (ZDES) method [59][72][74]. This was developed as an alternative way to cure the grey area and GIS problem, by manually specifying the RANS and LES zones. This method is based on three modes that are manually specified by the user onto different domains of the simulation as shown below:
\[ d_{ZDES} = \begin{cases} 
  d_w & \text{if } \text{ides} = 0 \\
  \tilde{d}_{DES}^I & \text{if } \text{ides} = 1 \text{ and } \text{imode} = 1 \\
  \tilde{d}_{DES}^{II} & \text{if } \text{ides} = 1 \text{ and } \text{imode} = 2 \\
  \tilde{d}_{DES}^{III} & \text{if } \text{ides} = 1 \text{ and } \text{imode} = 3 
\end{cases} \] (2.159)

Where \( d_w \) is the wall-distance term which represents the RANS turbulent length scale for the SA RANS model. \( \tilde{d}_{DES} \) represents the DES length scales for each mode.

The ZDES length scale which enters the transport equation for \( \tilde{\nu} \) (or potentially \( k \) if this method was applied to a two-equation model) is:

\[ \tilde{d}_{ZDES} = (1 - \text{ides})L_{RANS} + \text{ides}L_{LES} \] (2.160)

Where \( L_{RANS} = d_w \) (for the SA underlying RANS model) and \( L_{LES} = \tilde{d}_{DES}^{I\text{or}II\text{or}III} \), in which \( I, II, III \) represent the three modes of ZDES.

The first mode is same as the standard DES model:

\[ \tilde{d}_{DES}^I = \min(L_{RANS}, C_{DES}\tilde{\Delta}_{DES}^I) \] (2.161)

The second mode is of the same form as DDES:

\[ \tilde{d}_{DES}^{II} = L_{RANS} - f_{\text{dmax}}(0, L_{RANS} - C_{DES}\tilde{\Delta}_{DES}^{II}) \] (2.162)

The third mode is a wall-modelled LES function where the first mode is explicitly used below a certain wall-distance value:

\[ \tilde{d}_{DES}^{III} = \begin{cases} 
  d_w & \text{if } d_w < d_w^{\text{interface}} \\
  \tilde{d}_{DES}^I & \text{otherwise} 
\end{cases} \] (2.163)

The purpose of these three modes is to switch the version of DES depending on the flow type. In [59], the following flow types are characterised:

- Category I - Location of the separation point is fixed by the geometry. This is where mode 1 can be used because the transition from RANS to LES
occurs quickly without any grey area.

- Category II - Location of the separation point is unknown a priori. This is suited to mode 2, where the transition from RANS to LES is crucial to the development of the initial shear layer. In this mode the DDES shielding function is used to prevent grid-induced separation (GIS).

- Category III - Wall-modelled LES. Mode 3 allows the user to specify the position of the interface between the RANS and LES zones, such as in a channel flow or on a more complex geometry.

For a complex flow, several different regions can be used in different areas and the transition from one to another is an area which was addressed in [59], but still presents a challenge due to the need to produce synthetic turbulence at the interface for mode 3, but then damp these out for a specified RANS zone further ahead.

The filter width $\tilde{\Delta}_{DES}$ is set to the cell volume $\Delta_{vol}$ for mode 1. But for mode 2, the filter width includes the $f_d$ function of DDES to select between the using the maximum grid dimension $\Delta_{max}$ or the cell volume $\Delta_{vol}$. This aim of this is to use the maximum grid dimension as the filter width within the boundary layer through the $f_d$ shielding function and then the cell volume at the edge of the boundary layer to ensure a quicker drop of the turbulent viscosity and therefore a more rapid switch to LES mode.

The ZDES method has been extensively tested on a wide range of academic cases and well as complex industrial flows. This method is well suited to industrial flows that are typically well known (such as aerospace or automobile vehicles that have long design cycles), and allows a very precise allocation of the RANS and LES zones.

Two-Velocity field hybrid RANS-LES model

A particularly novel hybrid RANS-LES formulation is the two-velocity field hybrid RANS-LES model [41]. In this model instead of using a single velocity field to couple both RANS and LES zones, two velocity fields; the fluctuating velocity and running time-averaged are defined and modelled by a separate eddy viscosity [75]. The principle for this method is taken from the work of Schumann [76], in which the residual sub-grid scale stress tensor is split into two: a locally isotropic part and an in homogenous part as shown in Equation 2.164.
\[ \tau_{ij}^r - \frac{2}{3} \tau_{kk} \delta_{ij} = -2 \nu_r f_b (S_{ij} - \langle S_{ij} \rangle) - 2(1 - f_b) \nu_a (\overline{S}_{ij}) \]

Where \( \langle \cdot \rangle \) represents an ensemble average of the filtered equations. The turbulent viscosities \( \nu_r \) and \( \nu_a \) are based upon the fluctuating and mean strains respectively \[75\]. A blending function \( f_b = \tanh(1.5L_t/\Delta) \) is used to avoid double counting of the stresses \[41\].

The Smagorinsky sub-grid scale model is used to model the isotropic turbulent viscosity \( \nu_r \), and the \( \varphi - f \) RANS model is used to compute the inhomogeneous turbulent viscosity \( \nu_a \).

\[ \begin{align*}
\nu_r &= (C_s \Delta)^2 \sqrt{2(\overline{S}_{ij} - \langle S_{ij} \rangle)(\overline{S}_{ij} - \langle S_{ij} \rangle)} \\
\nu_a &= C_\mu \varphi k T \\
T &= \max(\frac{k}{\varepsilon}, C_T \sqrt{\frac{\nu}{\varepsilon}})
\end{align*} \] (2.165-2.167)

The model has shown good results for the 3D Diffuser \[11\], the flow around a trailing edge \[75\] and others during the ATAAC project \[56\].

There are many more hybrid RANS-LES methods that seek to achieve the same objectives as the methods described above. As the scope of this research has been DES related, a more thorough review of other models was not included. However the reader is referred to the work of Fröhlich et al. \[53\] and Sagaut et al. \[77\] for a more comprehensive review.

**Numerics and mesh resolution considerations**

In terms of mesh resolution and numerics, the LES mode of any hybrid RANS-LES method should adhere to the same guidelines as LES, i.e high-order numerical schemes with little numerical dissipation, a time step small enough to ensure \( CFL < 1 \) and if possible cubic cells which meet the requirements of the checks detailed previously, such as the ratio \( \Delta/\eta \), or the ratio \( \nu_t/\nu \).

In a hybrid model the RANS mode can be treated with coarser higher aspect ratio cells in the near-wall region compared to LES, and upwind numerical
schemes rather than pure central differencing schemes are typically deemed satisfactory. For seamless and DES related methods, where the grid resolution is explicitly linked to the choice of RANS or LES zones, the mesh design is also extremely important and influential on the end result. Too coarse mesh resolution in the LES region can lead to an underestimation of the turbulent shear stress which can lead to an inaccurate prediction of the separation region. The choice of numerical scheme is also important for seamless methods, as the numerics must match each zone (i.e CDS for LES and upwinding for RANS). This issue will be discussed further in Chapters 3 and 4.

2.4 Closure

In this chapter a review of different turbulence modelling approaches has been presented. It has been shown that there exists a wide range of turbulence modelling options that each seek to provide closure to the Navier-Stokes equations. The relative merits of different eddy viscosity and Reynolds stress models has been discussed as well as alternative eddy viscosity models that seek to correct some of the failures of the standard models. Special attention has been placed upon elliptic relaxation models and the way in which these attempt to provide improved near-wall asymptotic behaviour. The concept of hybrid RANS-LES modelling has also been explained, in particular the way in which these approaches offer more efficient near-wall modelling for high-Reynolds number flows (compared to a wall-resolved LES) and how they also provide more advanced near-wall modelling compared to wall-functions for LES.
Chapter 3

Numerical solver

3.1 Introduction

All calculations were performed using the open-source code Code_Saturne which was developed by EDF Energy. It is a 3D code based on a finite-volume numerical method capable of solving steady or transient, laminar or turbulent flows, on structured or unstructured grids. A range of RANS turbulence models is available, including high and low-Reynolds number second moment closure and eddy-viscosity based models. All other hybrid RANS-LES models referred to in this work are implemented by the author unless otherwise stated. The following section gives an overview of the numerical methods used in Code_Saturne as well as some validation of the turbulence models used on a fully developed channel flow.

3.2 Finite volume method

In Cartesian coordinates the general transport equation for a variable \( \phi \) can be expressed as:

\[
\frac{\partial \rho \phi}{\partial t} + \frac{\partial \rho u_j \phi}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) + S_\phi
\]

(3.1)

Where terms \( a, b, c \) and \( d \) of Equation 3.1 represent the unsteady, convection, diffusion and source terms respectively. \( u_j \) represents the velocity, \( \Gamma \) the diffusion coefficient and \( S_\phi \) the source term.
In the finite volume method, the fluid domain is divided into a finite number of control volumes, with the conservation equations applied to each of these. In order to use this method to solve any general transport equation, the starting point is to use its integral form:

\[ \int_{\Omega} \frac{\partial \rho \phi}{\partial t} d\Omega + \int_{V} \frac{\partial}{\partial x_j} (\rho u_j \phi) d\Omega = \int_{\Omega} \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) d\Omega + \int_{\Omega} S_{\phi} d\Omega \quad (3.2) \]

Using Gauss's theorem, the volume integrals of the convection and diffusion terms are transformed into surface integrals, which leads to:

\[ \int_{\Omega} \frac{\partial \rho \phi}{\partial t} d\Omega + \int_{S} \rho u_j n_j \phi dA = \int_{S} \Gamma \frac{\partial \phi}{\partial x_j} n_j dA + \int_{\Omega} S_{\phi} d\Omega \quad (3.3) \]

The integrals of the source term and unsteady term are then approximated using cell centre values, \( S_{I,\phi_I} \) and the cell volume \( \Omega_I \):

\[ \underbrace{\int_{\Omega_I} \frac{\partial \rho \phi_I}{\partial t} d\Omega}_{a} + \underbrace{\int_{S} \rho u_j n_j \phi dA}_{b} = \int_{S} \Gamma \frac{\partial \phi}{\partial x_j} n_j dA + \underbrace{S_{\phi,I} \Omega_I}_{d} \quad (3.4) \]

Terms \( b \) and \( c \) of Equation (3.4) require specific discretisation methods, the choice of which greatly affects the overall accuracy of the solution. These methods will be explained in the following sections.

### 3.3 Spatial Discretisation

The surface integrals are summed over each face of each cell for every control volume. These surface integrals are approximated by the mid-point rule, which is the product of the integrand at the cell-centre and the face area. Code_Saturne uses a co-located arrangement for all the variables. These variables are stored at the cell-centre and as such interpolation is required to obtain their values at the face centre. Figure 3.1 shows a generic representation of a face \( F \) between two adjacent cells \( \Omega_I \) and \( \Omega_J \). Points \( I \) and \( J \) are the centre of mass for each of the cells, and points \( I' \) and \( J' \) are the projections of these points onto the vector normal to the face \( F \).
3.3.1 Convection term

For the convection term (term \(b\)), there are several options for the user to obtain face-centred values.

\[
\int_{S_{ij}} \rho u_j \phi n_j dA \approx (\rho u_j \phi n_j A)_{ij}
\] (3.5)

There are two main discretisation schemes in Code_Saturne to approximate the face value of \(\phi\). These are a first and second order upwind differencing scheme (UDS and SOLU) and a second order central differencing scheme (CDS). It is possible to blend between these two schemes using a blended differencing scheme (BDS). There is also a slope test based on the product of the gradients at the cell centres to dynamically switch from CDS or SOLU to UDS to eliminate unwanted oscillations. A hybrid numerical scheme has also been implemented by the author to switch between CDS and SOLU depending on a blending function.
Upwind differencing scheme (UDS)

This scheme takes one of the neighbouring cell centre values as the face value depending on the flow direction, as shown in Equation 3.6 (see also Figure 3.2):

\[
\phi_{IJ} = \begin{cases} 
\phi_I & \text{if } (u_i n_i) \geq 0 \\
\phi_J & \text{if } (u_i n_i) < 0 
\end{cases}
\] (3.6)

This scheme is first-order accurate and is stable and bounded, although it can introduce additional numerical diffusion. If the grid is coarse this additional numerical diffusion can exert a large influence on the quality of the solution.

Second order linear upwind scheme (SOLU)

This scheme is a second order accurate scheme building upon the first-order upwind scheme:

\[
\phi_{IJ} = \begin{cases} 
\phi_I + I' F(\nabla \phi)_I & \text{if } (u_i n_i) \geq 0 \\
\phi_J + J' F(\nabla \phi)_J & \text{if } (u_i n_i) < 0 
\end{cases}
\] (3.7)
Where $I'F$ and $J'F$ are the distances between the face centre and the cell centre. The gradients are calculated explicitly from the solution at the previous time step.

**Central differencing scheme (CDS)**

This scheme uses a simple linear interpolation of $\phi$ between the two nearest cell centres.

$$\phi_{IJ} = \phi_J \lambda_{IJ} + \phi_I (1 - \lambda_{IJ})$$  \hspace{1cm} (3.8)

Where the linear interpolation, $\lambda_{IJ} = FJ'/I'J' (= 1/2$ for regular grids). It is second-order accurate on uniform grids and exhibits low numerical dissipation. However when the cell Peclet number is high, it can suffer from oscillatory solutions [79].

**Slope test**

To avoid numerical oscillations that can occur from the use of CDS, *Code_Saturne* uses a slope test to check for non-monotonicity and locally switch from CDS (or SOLU) to UDS to eliminate unwanted oscillations. The test is based on the calculation of an upwind gradient:

$$G_I = \phi^{SOLU}_{IJ},$$  \hspace{1cm} (3.9)

Where $\phi^{SOLU}_{IJ}$ is a second-order upwind interpolation for the variable being subjected to the test.

The test checks for local non-monotonicity, by checking if the dot product between the gradient calculation in neighbouring cells, $I$ and $J$, is positive. If this product is negative, then the numerical scheme is locally switched from CDS or SOLU to fully UDS.

$$G_I \cdot G_j < 0$$  \hspace{1cm} (3.10)

This is useful for locally fixing some areas of the domain where poor mesh quality may cause the computation to become unstable or produce spurious oscillations.
Blended differencing scheme (BDS)

This scheme allows the end-user to prescribe a linear blend of UDS with CDS or SOLU uniformly over the whole domain, according to the following formulation:

\[ \phi_{BDS} = \phi_{CDS \text{ or } SOLU} + (1 - \zeta) \phi_{UDS} \]  

(3.11)

Where the subscripts represent the scheme used and \(\zeta\) is set to between zero and one depending on the ratio of CDS/SOLU to UDS required.

Flux reconstruction

All of the schemes mentioned previously are implemented in Code_Saturne with a correction for grid non-orthogonality. This is especially important for unstructured grids where the face centre does not lie in the midpoint between the cell centres. If the gradient of \(\phi\) is known at the cell centre \(I\), then the value of \(\phi\) at point \(I'\) is approximated as:

\[ \phi_{I'} = \phi_{I} + \frac{1}{2} II' \left( \left( \frac{\partial \phi}{\partial x_j} \right)_I + \left( \frac{\partial \phi}{\partial x_j} \right)_J \right) \]  

(3.12)

Which means that for non-orthogonal grids, Equations 3.6, 3.8 and 3.11 use \(\phi_{I'}\) or \(\phi_{J'}\) instead of \(\phi_I\) and \(\phi_J\).

Hybrid numerical scheme (HNS)

Undertaking a DDES calculation poses a problem to the end-user when deciding upon the correct numerical convection scheme. An explicit LES requires a low dissipative numerical scheme, as the sub-grid scale model provides the dissipation of the small turbulent scales. 2nd order or higher order central differencing schemes (CDS) are frequently used to facilitate this requirement as upwind differencing schemes (UDS) typically add too much numerical dissipation.

Applying a CDS scheme to a RANS model can lead to excessive numerical oscillations when the grid is too coarse or the cell Péclet number is large. To overcome this, UDS schemes or schemes based on a blend of CDS and UDS are frequently used.

In a seamless hybrid RANS-LES approach such as DDES, there should also be a seamless switch between the numerical schemes to fulfil the numerical requirements of each mode. The hybrid numerical scheme of Travin et al. [52] is
one such approach. It allows a switch between an upwind based scheme and a
central differencing scheme depending on a blending function. This hybrid nu-
merical scheme found widespread use with the standard version of DES, and was
automatically adopted for use with the newer DDES formulation. However, since
the switching between LES and RANS in DDES is based on the blending function
\( f_d \), the numerical scheme should also be blended using the same function, and not
by means of a separate function, developed originally for standard DES. Thus,
in the current work the blending function \( f_d \) to control the numerical scheme as
follows:

\[
\phi_f = \phi_{f,SOLU} \quad \text{if} \quad L_{RANS} < L_{LES} \\
\phi_f = (1 - f_d)\phi_{f,SOLU} + f_d\phi_{f,CDS} \quad \text{if} \quad L_{RANS} > L_{LES}
\]

Where SOLU and CDS represent a second-order upwind based scheme and a
centred scheme, respectively. Further details of this scheme will be presented in
Chapter 4.

3.3.2 Diffusion term

The discretisation of the diffusion (term \( c \) of Equation 3.4) can be approxi-
mated by:

\[
\int_S \Gamma \frac{\partial \phi}{\partial x_j} n_j dA \approx \Gamma \frac{\partial \phi}{\partial x_j} n_j A
\]

(3.15)

with a second order linear approximation for the gradient at the face centre:

\[
\Gamma \frac{\partial \phi}{\partial x_j} n_j A = \Gamma \frac{\phi_{J'} - \phi_I}{IJ} n_j A
\]

(3.16)

Where \( \phi_{I'} \) and \( \phi_{J'} \) are calculated using equation 3.12.

3.3.3 Gradient reconstruction

There are several methods of calculating the gradient \( \nabla \phi_f \) at the cell centre in
\textit{CodeSaturne}. The method used throughout this study is the iterative method
of Muzaferija et al. [80]. Assuming that \( \phi_f \) is known, then an expression for the
gradient is given using divergence theorem:

$$\int_{\Omega} \nabla \phi d\Omega = \int_{S} \phi \, n_j \, dA$$

(3.17)

Which can be reformulated in terms of the value of $\phi$ at the cell faces to give:

$$\Omega_I (\nabla \phi)_I = \phi_{IJ} n_{ij} A_{IJ}$$

(3.18)

This gradient reconstruction procedure requires the value of $\phi_{IJ}$ to calculate the gradient, however the flux reconstruction procedure requires the gradient to calculate $\phi_{IJ}$. Therefore Equation 3.18 is solved in an iterative manner until the values of the gradient $(\nabla \phi)_I$ and $\phi_{IJ}$ have converged.

### 3.4 Boundary conditions

**Inlet**

For the inlet boundary condition the values at the boundary face for the momentum and turbulence model equations are specified as Dirichlet conditions by the user. A zero-pressure gradient Neumann condition is set for the pressure. The correct specification of turbulence model parameters, $k$ and $\varepsilon$ are (for some applications) very important and can have a strong effect on the results. The issue of inlet sensitivity has been examined and discussed in depth by Spalart et al [81] for the case of external aerodynamic flows.

**Outlet**

For the outlet boundary condition a homogeneous Neumann condition is applied to all the momentum and turbulence equations so that the normal gradient to the outlet plane is zero. For the pressure a homogeneous Dirichlet value is set at one face.

**Walls**

At walls the pressure and mass flow rate are set with homogeneous Neumann conditions so that the gradients normal to the wall are set to be zero. A homogeneous Dirichlet condition is applied to the tangential velocity. The values of
Scalars may be prescribed by either Dirichlet or Neumann conditions dependent on the choice of the user.

Symmetry

For a symmetry boundary, the same conditions are applied as for walls except for the tangential velocity, which is a homogeneous Neumann condition.

3.5 Turbulence models

The turbulence models that have been used within this study will be presented as implemented into Code_Saturne. Three turbulence models form the main body of this work; they are the \( k - \omega \) SST model, the Spalart-Allmaras (SA) model and the \( \varphi - f \) model. These three models are then used as the underlying RANS model for three DDES models; the SST-DDES model, the SA-DDES model and the \( \varphi - f \) DDES model (to be presented in Chapter 4).

3.5.1 RANS models

Spalart-Allmaras model

The SA model version used throughout this work is the same as presented in Equation 2.25 but without the additional \( f_{\ell 2} \) tripping term. Thus the transport equation for the SA \( \tilde{\nu} \) term is displayed below with the same coefficients as Table 2.1:

\[
\frac{D\tilde{\nu}}{Dt} = c_{b1} \tilde{S} \tilde{\nu} - c_{w1} f_w \left( \frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( \nu + \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_j} \right] + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \quad (3.19)
\]

\( k - \omega \) SST model

The form of the SST model is as shown previously in Equation 2.58, where the definition of the turbulent viscosity is shown in Equation 2.51.

\( \varphi - f \) model

The form of the \( \varphi - f \) model is also as shown previously in Equations 2.116 to 2.119, where the definition of the turbulent viscosity is shown in Equation 2.51.
3.5.2 DDES models

SA-DDES model

The SA-DDES model is based on Equation 3.19 with the wall distance term $d$, being replaced by the DDES length scale $L_{DDES}$ as described in Spalart et al. [7]. Such that Equation 3.20 becomes:

$$\frac{D\tilde{\nu}}{Dt} = c_{b1}\tilde{S} - c_{w1}f_w \left( \frac{\tilde{\nu}}{L_{DDES}} \right)^2 + \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( \nu + \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_j} \right] + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i}$$ (3.20)

Although there is some debate (private communication with Dr C. Mockett) in the DDES community about whether all incidences of the wall distance should be replaced with the DDES length scale, in this work the approach as originally described in [7] has been taken. Equations 2.29 and 2.33 are modified such that they read:

$$\tilde{S} = \Omega + \frac{\tilde{\nu}}{\kappa^2 L_{DDES}^2} f_{v2}$$ (3.21)

$$r = \frac{\tilde{\nu}}{\tilde{S} \kappa^2 L_{DDES}^2}$$ (3.22)

Where the DDES length scale is defined as:

$$L_{DDES} = d - f_d \max (0, d - C_{DDES} \Psi \Delta)$$ (3.23)

The Low-Reynolds number correction term $\Psi$ as described in Spalart et al. [7] is used to return the DDES model to the classical Smagorinsky sub grid scale model in LES mode. In some incidences where the turbulent viscosity drops to too low a value, the wall sensing terms of the SA model ($f_{v1}, f_{v2}$) switches on and reduces the turbulent viscosity to an even more unrealistic value. This correction term stops that. Its form is:
\[ \Psi = \sqrt{\min \left( 100; \frac{1}{f_{v1}} - \frac{f_{v2}c_{b1}}{f_w f_{v1} K^2 c_{w1}} \right)} \]  

(3.24)

The value for \( C_{DDES} \) has been calibrated on the Decaying Isotropic Turbulence test case, which gives rise to \( C_{DDES} = 0.65 \) as shown in Figure 3.3. Further details of the Decaying Isotropic Turbulence test case and validation will be given in Chapter 4.

For the \( k - \omega \) SST model, the length scale determining variable is \( \omega \); as such the destruction term for the turbulent kinetic energy \( k \) (Equation 2.57), is modified to include the DDES length scale;

\[
\frac{Dk}{Dt} = \min(P_k, 10 C_{\mu} k \omega) - \frac{k^{3/2}}{L_{DDES}} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right]
\]

(3.25)

Where

\[
L_{DDES} = \frac{\sqrt{k} \beta^* \omega}{f_{d \max}} - f_{d \max} \left( 0, \frac{\sqrt{k} \beta^* \omega}{C_{DDES} \Delta} \right)
\]

(3.26)

The value of \( C_{DDES} \) has also been calibrated on the DIT case to give a value of \( C_{DDES} = 0.65 \) as shown in Figure 4.2(a).
3.5.3 Validation - Fully developed channel flow at Re=395

Fully developed channel flow is a common test case used for the validation of RANS turbulence models. There are comprehensive sets of DNS data available at a range of different Reynolds numbers which enables a detailed investigation into the flow statistics produced by the turbulence model. This test case’s suitability stems from a simple simple set of boundary conditions and an unambiguous source of turbulence generation, arising from the presence of the walls. As such this flow is commonly used to fine tune the predicted turbulent Reynolds stress budgets obtained close to a wall, where such values become anisotropic. The DNS data used for this investigation is from Moser et al. [82], and this data forms a benchmark against which several RANS models are tested.

Computational grid and solution domain

The solution domain (Dimensions: $L_x \times L_y \times L_z = 0.1 \times 1 \times 0.1$) was meshed with a structured grid of 197 cells. It was refined near the wall to ensure the grid was suitable for low Reynolds number models to perform correctly and the non-dimensional wall distance for the nodes closest to the wall were $y^+ < 1$.

Boundary conditions

All cases were computed for a Reynolds number of Re = 395 based on the friction velocity. A periodic boundary condition was set in the stream-wise direction and symmetry was imposed at half the channel height. A constant pressure drop was imposed to force the flow in the stream-wise direction. A dimensionless
time step of 0.01 (based on the channel half height and free-stream velocity) was used, which provided a CFL number of less than one for all cases.

Results

The results for the SST, $\varphi-f$ and SA models are in line with previous studies for these models [36, 39]. The most notable differences are in the predictions for the turbulent kinetic energy (Figure 3.5(b)), where the $\varphi-f$ model provides the best agreement with the DNS data (results for the SA model are excluded as the turbulent kinetic energy is not directly used). The dissipation is over-estimated by the SST model (Figure 3.6(a)), yet as the turbulent kinetic energy is also under-predicted, the resultant turbulent viscosity is similar to the $\varphi-f$ (Figure 3.7(a)). The main purpose of this case was to ensure that the models had been implemented correctly and validated against a well know case. These results show that this is the case and gives confidence to move to more complex cases.

![Figure 3.5: (a) Mean stream-wise velocity and (b) Mean turbulent kinetic energy for fully developed channel flow at Re=395](image)

Figure 3.5: [a] Mean stream-wise velocity and [b] Mean turbulent kinetic energy for fully developed channel flow at Re=395

3.6 Closure

This chapter has introduced the main numerical methods used within Code_Saturne. The discretisation schemes for the convective and diffusive terms have been presented, as well as the boundary conditions used within this work. A novel hybrid numerical scheme based upon the DDES blending function to switch between a
Figure 3.6: (a) Mean turbulent dissipation and (b) Mean wall-normal Reynolds Stress component for fully developed channel flow at Re=395

CDS and upwinding scheme has also been presented. The exact formulations of the turbulence models used within this work have also been outlined as well as a brief discussion concerning their performance on a fully developed turbulent channel flow.
Figure 3.7: (a) Mean turbulent viscosity and (b) mean $\varphi$ function for fully developed channel flow at Re=395
Chapter 4

Development of the $\varphi - f$ DDES model

4.1 Introduction

The original formulation of DES [51] was based upon the Spalart-Allmaras RANS model [16]; this was chosen because it offered the most convenient length scale to turn a RANS model into a SGS model [83], and it is also the RANS model co-created by one of the principal creators of DES.

Having a DES based only upon the Spalart-Allmaras model was deemed a flaw in Travin et al. [52], as it was concluded that it is useful and necessary to have an underlying RANS model that can predict separation, or near-wall conditions, where the SA model fails so to do. In Travin et al. [52] the SST-DES model was presented; it was calibrated using Decaying Isotropic Turbulence [84] and evaluated on several test cases including the backward facing step, circular cylinder and NACA0012 airfoil. For the cases where the separation was due to the geometry (backward facing step), massive stall (NACA0012 airfoil at 45 AOA) or laminar separation (circular cylinder with laminar separation) there was little difference between the SA-DES and SST-DES models. It was also stated that the authors would expect to see a tangible difference between the underlying RANS models for turbulent boundary-layer separation [52]. During the FLOMANIA project [49] several new DES formulations based upon different RANS models were investigated [85]; the Strain-Adaptive Linear Spalart-Allmaras model [18], the linear local realizable (LLR) $k - \omega$ Model [86], and the compact explicit algebraic stress
model (CEASM) [57]. These DDES formulations were investigated more thoroughly in the doctoral thesis of Mockett [57]. The main conclusions matched that of Travin et al. [52], in that for massively-separated flows with geometry-induced separation, there was little sensitivity to the underlying RANS model, however for flows with turbulent separation from smooth surfaces the sensitivity was high [57].

An investigation was conducted during this period into the effect of alternative length scale substitutions within a DDES [88], which concluded that once the strongly varying levels of dissipation of the alternative SGS models were calibrated by the simulation of Decaying Isotropic Turbulence (DIT), their performance in a practical test case was comparable [88]. It was also noted that although several formulations did not adhere to a Smagorinsky-like form under equilibrium conditions, they were deemed acceptable formulations of DDES. A Reynolds Stress model variant of DES was initiated during the course of FLOMANIA, however no results or model formulations were described or shown [49]; this was most likely due to issues with stability and the challenge of deriving a suitable formulation. The conclusion of the FLOMANIA project was that in cases that are subjected to massive flow separation, DES shows a ‘significantly reduced dependency’ on the background RANS model compared to URANS. However in cases where the boundary layer physics are important, the underlying RANS model plays an important role.

During the DESider project [6] (the follow up project to FLOMANIA) many partners investigated different DDES formulations. The use of the CEASM was carried forth, but no second moment closure models were investigated nor were any elliptic relaxation based models. These various DES formulations were tested on a range of test cases, including external and internal flows. The conclusions of the DESider project with regards to background model choice were:

- There was only marginal sensitivity of DES to the background RANS model where there was massive separation due to the geometry but noticeable sensitivity when the flow was subject to pressure-induced turbulent separation.
- When hybrid RANS-LES schemes are employed ‘large eddies’ frequently pass into the RANS layer and the underlying RANS model is therefore expected to account for severe non-equilibrium effects. Very few RANS models implemented in a DES formulation during the project would be expected to represent this.
Several more complex test cases such as the Ahmed car body [89] have shown a strong sensitivity to the underlying RANS model within a DDES approach [6]. As the RANS model is responsible for the separation point prediction and the state of the upstream flow, any differences prior to this point have a large effect on the resulting separated LES region.

From over a decade of research on Detached-Eddy Simulation, the choice of the underlying RANS model within a DES simulation is still important. It has been shown that for the formulations tested, there are fewer differences in the presented results than between alternative URANS models in highly separated cases. However there are also cases where, due to the complex flow physics, the underlying RANS model does influence the result and as such the choice of this model is important with DES.

It is also the case that the current DES formulations for most part have been limited to two-equation RANS models, based upon the eddy-viscosity hypothesis. Just as more complex URANS models (such as second moment closure models) have been developed to represent better complex flow physics, incorporating these into a DES has the potential to improve the prediction of near wall flow structures. This is therefore the area that this research will focus on.

4.2 The $\phi - f$ DDES model

The $\phi - f$ model was chosen as the underlying RANS model for several reasons. Firstly, this model in its original RANS form has shown good performance on a range of test cases, including both internal and external flows compared to standard linear eddy viscosity models such as the SST and SA [36, 39, 90–92]. By including a separate transport equation for the wall-normal fluctuations, the $\phi - f$ model is better able to represent the near-wall boundary layer and turbulent shear stress than models such as the SST and SA models, that are based on more ad-hoc damping functions to model the near-wall region. As the RANS model within a DDES approach is used to model the attached boundary layers, having a model that is better able to model this region is an advantage.

This model also has already been used successfully within a hybrid RANS-LES approach, the two-velocity field hybrid RANS-LES method [75] where it has been shown to be an accurate underlying model giving good separation point prediction. Since the first publication of the work contained within this thesis,
there has been a $\overline{\nu^2} - f$ based DES model \[93\] that has shown promising results for a circular cylinder at Re=3900.

Therefore for these reasons it was decided to develop a new formulation of DDES based upon the $\varphi - f$ model.

### 4.2.1 Derivation

The $\varphi - f$ DDES model is based on the original DDES formulation \[7\], in which the standard DDES modifications are made to the length scale determining equation by changing the length scale to $L_{DDES}$:

$$L_{DDES} = L_{RANS} - f_d \max(0, L_{RANS} - C_{DDES}\Delta)$$  \hspace{1cm} (4.1)

To recall, the blending function for DDES is:

$$f_d = 1 - \tanh \left( [8r_d]^3 \right),$$  \hspace{1cm} (4.2)

where

$$r_d = \frac{\nu_t + \nu}{\sqrt{U_{ij}U_{ij}\kappa^2y^2}},$$  \hspace{1cm} (4.3)

$k$ is the Karman constant and $y$ is the distance to the wall. $r_d$ equals 1 in the boundary layer and gradually reduces to 0 towards the edge of the boundary layer. The function $f_d$ is designed to be 1 in the LES region where $r_d \ll 1$, and 0 elsewhere.

For the $\varphi - f$ model, the standard DDES modification is made to the turbulent kinetic energy equation; thus Equation 2.116 becomes:

$$\frac{Dk}{Dt} = P_k - \varphi \frac{k^{3/2}}{L_{DDES}} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \hspace{1cm} \text{Modified}$$  \hspace{1cm} (4.4)

In the current formulation, the DDES length scale modification appears only in the turbulent kinetic energy equation and not in Equation 2.119 for $\varphi$. This was
chosen to be consistent with the principle of the original DES formulation. The study of [88] investigated the differing results that may be obtained when using an alternative length scale substitution (through $\nu_t$, which also is comparable to using $\varphi$), however in the scope of the present study, it was decided to establish a baseline version of the $\varphi-f$ DDES model before investigating alternative substitutions.

The length scale was replaced by $\varphi k^{3/2}/\varepsilon$ rather than $k^{3/2}/\varepsilon$ as the former gives a more representative approximation of the near-wall length scale, which then stays consistent from RANS to LES mode.

One of the primary verification tests for a DDES formulation is its simplification under local equilibrium conditions. The turbulent viscosity should return to a sub-grid scale Smagorinsky-like form when using the DDES length scale $L_{DDES} = \Psi C_{DDES} \Delta$, i.e $\nu_t = (C\Delta)^2 S$, where $C$ is a constant and $\Psi$ is the correction function. Under local equilibrium conditions (where production, $P_k = \nu_t S^2$, is equal to dissipation) the $k$ equation (in LES mode) and $\varepsilon$ equation become:

$$\nu_t S^2 = \frac{\varphi k^{3/2}}{\Psi \varphi f C_{DDES} \Delta}$$

(4.5)

$$\nu_t S^2 = \frac{C_{\varepsilon^2}}{C_{\varepsilon^1}} \varepsilon$$

(4.6)

From which it is straightforward to show that the Smagorinsky form of the $\varphi-f$ sub-grid scale model is given by:

$$\nu_t = A_{\varphi f} (\Psi_{\varphi f} C_{DDES} \Delta)^2 S, \quad A_{\varphi f} = \left( \frac{C_{\varepsilon^2}}{C_{\varepsilon^1}} \right)^{3/2} \left( \frac{1}{\varphi} \right)^{1/2}$$

(4.7)

The correction term, $\Psi_{\varphi f}$ should be of the form $A_{\varphi f} \Psi_{\varphi f}^2 = \text{const}$. Unlike the SST-DDES model $A_{\varphi f}$ is not a constant and is dependant on $\varphi$ both directly and via the $C_{\varepsilon^1}$ parameter (Equation 2.123). This effectively results in a dynamically $\varphi$-dependant LES (Figure 4.1). The $\varphi$-dependance is removed by setting:

$$\Psi_{\varphi f} = \left( \frac{C_{\varepsilon^1}}{C_{\varepsilon^2}} \right)^{3/4} (\varphi)^{1/4}$$

(4.8)

This cancels out the terms in front of $C_{DDES} \Delta$ and returns the model to the
standard LES mode for DDES.

4.2.2 Calibration

As with the $C_s$ constant from the Smagorinsky LES sub-grid scale model, the constant $C_{DDES}$ must be calibrated to achieve the correct level of dissipation. It was found in both the DESider [6] and ATAAC [56] projects, as well as many other publications [57] that the value of this constant is dependant on not only the implementation but also the numerical scheme employed. Thus it is important to validate the choice of this constant before moving to any further validation cases. The use of decaying isotropic turbulence (DIT) is used as a simple test case to calibrate this constant.

Decaying Isotropic Turbulence

The natural evolution of homogeneous decaying isotropic turbulence (DIT) provides a fundamental test case in assessing the numerical performance of the LES branch of DDES. This test cases allows the following to be evaluated:

- The capability of the turbulence model to predict the energy cascade and to resolve turbulent structures.
- Calibration of the $C_{DDES}$ parameter in DDES.
- The performance of the $\Psi$ correction term.

![Figure 4.1: Demonstration of the functionality of the $\Psi$ correction term for the DIT case using a $64^3$ grid at $t = 2s$ and $C_{DDES} = 0.60$](image)

Figure 4.1: Demonstration of the functionality of the $\Psi$ correction term for the DIT case using a $64^3$ grid at $t = 2s$ and $C_{DDES} = 0.60$
• Assessment of the solver’s level of numerical dissipation. e.g verification of the use of a centred convection scheme against an upwind scheme in the LES branch of DDES.

DIT is therefore seen as an essential verification and validation test case for any DDES formulation. The solution domain of the DIT calculation is a cube with side length $2\pi$. The solution domain is meshed with two regular grids consisting of $32^3$ and $64^3$ cubic and equidistant cells. Periodic boundary conditions are imposed throughout. The initial velocity field is set with a suitable instance of isotropic turbulence by the use of an inverse Fourier transform using the experimental data of Comte-Bellot & Corrsin [84]. In order to obtain the initial values for other variables such as the pressure and turbulence quantities ($k$ & $\varepsilon$ etc), a frozen velocity field simulation was conducted, where the velocity field is frozen and the other dependant variables are calculated; once converged, this was used as initial conditions for the unsteady decay of turbulence simulation. To ensure that the LES mode of DDES is used, the length scale is set to that of a LES:

$$L_{DDES} = L_{LES} = \Psi C_{DDES} \Delta. \quad (4.9)$$

The temporal discretization is 2\textsuperscript{nd} order, and the hybrid numerical scheme is used to discretise spatially the convective momentum terms, while a 1st order upwind scheme is applied to the turbulent quantities. All results are presented for a non-dimensional time of $t = 2.0$ and are compared to the experimental data of Commte-Bellot & Corrsin [84].

Results

For both the SST-DDES and $\phi - f$ DDES models, three different values for $C_{DDES}$ were investigated using two grids of $32^3$ and $64^3$ in order to account for the effect of mesh refinement. The results for three values of $C_{DDES}$ using the SST-DDES model (Figure 4.2(a)) show that increasing the value of $C_{DDES}$ corresponds to increased high-wave number damping, and can be seen as adding more numerical dissipation. Although DIT is a much simplified test case, a value of $C_{DDES} = 0.65$ seems optimal considering both grids. This is in agreement with most other implementations summarised in the DESider project [6].

For the $\phi - f$ DDES model shown in Figure 4.2(b), the value of $C_{DDES}$ varies in a similar fashion to the SST-DDES formulation and shows the same sensitivity.
to the mesh refinement. A value of $C_{DDES} = 0.60$ was seen to be optimal although it is not possible to satisfy both grids with the same value of $C_{DDES}$ and any value between 0.55 and 0.65 could be justified. This emphasizes that the calibration of the $C_{DDES}$ must be conducted for each DDES formulation.

Both models show a weaker dependency on $C_{DDES}$ as the grid is refined. This can be explained by the fact that as $\Delta$ becomes smaller, the modelled turbulence should become smaller and there will be an increasing proportion of resolved turbulence.

![Energy spectra versus wavenumber ($\kappa$) for the velocity field, using varying values of $C_{DDES}$ for the 32$^3$ and 64$^3$ grids](image)

**Figure 4.2:** Energy spectra versus wavenumber ($\kappa$) for the velocity field, using varying values of $C_{DDES}$ for the 32$^3$ and 64$^3$ grids (a) SST DDES (b) $\varphi - f$ DDES

### 4.2.3 Numerical scheme sensitivity study

As DDES switches from a RANS mode to a LES-like mode it is important to investigate the effect of the numerical scheme on the LES branch of DDES. The simplest test case to evaluate the effect of the numerical scheme is again decaying isotropic turbulence (DIT).

For this investigation, several numerical schemes for the convective terms are used including the hybrid numerical scheme developed in this work. For the DIT test case all results are presented for a non-dimensional time of $t = 2.0$ and are again compared to the experimental data of [84]. A summary of the numerical schemes evaluated is given in Table 4.1.

Figures 4.3(a) & 4.3(b) show the results for the DIT case using the SST-DDES and $\varphi - f$ DDES models. The effect of different numerical schemes is a substantial shift in the prediction of the energy spectrum. Anything other than
Table 4.1: Summary of convective schemes applied to the momentum equations

<table>
<thead>
<tr>
<th>Convective schemes</th>
</tr>
</thead>
<tbody>
<tr>
<td>First order upwind scheme (UDS)</td>
</tr>
<tr>
<td>Second order linear upwind scheme (SOLU)</td>
</tr>
<tr>
<td>Centred difference scheme (CDS)</td>
</tr>
<tr>
<td>Hybrid numerical scheme (Hybrid)</td>
</tr>
</tbody>
</table>

A fully CDS scheme results in additional damping at high wave numbers and an over-prediction of the energy at lower wave numbers (corresponding to the energy-containing range).

The hybrid numerical scheme introduced in Chapter 3 performs as expected, employing a fully CDS scheme (as its in LES mode) and therefore matching the spectrum predicted by the CDS scheme applied without the hybrid numerical scheme. There is a considerable difference between the predictions of the 2nd order SOLU scheme and CDS, with the SOLU scheme grossly over-predicting the energy at the lower wave numbers. Therefore, in order to capture correctly the energy decay, a low-dissipative scheme such as the CDS scheme must be used to avoid any additional high-wave number damping.

Figures 4.4(a) & 4.4(b) show the results for the mean span-wise averaged skin-friction coefficient results for the 2D wall-mounted hump (studied in more detail in Chapter 7) using the SST DDES and $\varphi - f$ DDES models. For both models, using the second-order upwind scheme (SOLU) (commonly used for RANS models) produces a strongly delayed separation region that is common to both DDES models. While the length of the recirculation zone is similar to the experiment, the delay means that the separated shear layer is slow to form and strongly suggests that an upwind based scheme should not be used for a DDES. The upwind based scheme makes the grey area problem much greater and highlights the need for low-dissipation schemes in the initial separated shear layer region. This agrees with the results from the DIT case where the SOLU scheme produced excessive numerical dissipation. While using a CDS scheme gives much improved results compared to the SOLU scheme, applying it across the whole domain in both RANS and LES modes resulted in some numerical instabilities, such as oscillations that grew over time. In such cases it became necessary to reduce the time step and increase under-relaxation.
The hybrid numerical scheme behaves much like the CDS scheme in terms of accuracy but without the numerical instabilities. As it is directly based on the DDES blending function, the choice of SOLU or CDS is explicitly linked to the \( f_d \) function. As the recirculation region behind the hump is in LES mode (apart from the boundary layer) then it is guaranteed that the numerical scheme will be CDS provided that the LES and RANS zone are correctly meshed, as this ultimately decides the accuracy of the simulation.

Figures 4.5(a) and 4.5(b) show the mean span-wise averaged skin-friction coefficient results for the 2D periodic hills (studied in detail in Chapter 5) using the SST DDES and \( \phi - f \) DDES models. There is a sensitivity to the numerical scheme for both DDES formulation but it is far less pronounced than for the 2D hump. This can be attributed to the finer mesh used for the 2D hills (there is also little differences between the DDES formulation themselves), and the lower Reynolds number which suggests that the need for low-dissipative schemes increases for coarser meshes and higher Reynolds numbers.

It can therefore be seen that a proper DDES computation requires a low-dissipative numerical scheme in the separated regions to correctly capture the level of dissipation. This is especially important in the separated shear layer where a scheme that is too dissipative causes the resolved turbulence to take much longer to form and thus exaggerates the modelled-stress-depletion (MSD) problem. For these reasons the hybrid numerical scheme presented in Chapter 3 and shown in these results will be used in each of the test cases presented in the following chapters.

4.2.4 Computational expense

Table 4.2 shows the average time per iteration for each of the test cases using the DDES formulations. Even though the \( \phi - f \) DDES model contains a further two transport equations compared to the SST DDES model, it is never more than 10% more computationally expensive. Moreover, no numerical difficulties were encountered with the \( \phi - f \) DDES model and as such there was no need for smaller timesteps or additional numerical dissipation.
Table 4.2: Average time per iteration (in seconds) for each test case using the three DDES formulations

<table>
<thead>
<tr>
<th>Test cases</th>
<th>CPU Cores</th>
<th>SA DDES</th>
<th>SST DDES</th>
<th>ϕ – f DDES</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D periodic hills</td>
<td>48</td>
<td>2.25</td>
<td>2.3</td>
<td>2.5</td>
</tr>
<tr>
<td>NACA0021</td>
<td>12</td>
<td>1.7</td>
<td>1.9</td>
<td>2.1</td>
</tr>
<tr>
<td>2D wall-mounted hump</td>
<td>96</td>
<td>2.1</td>
<td>2.2</td>
<td>2.4</td>
</tr>
<tr>
<td>Ahmed Car body</td>
<td>120</td>
<td>3.7</td>
<td>3.8</td>
<td>4.2</td>
</tr>
</tbody>
</table>

Figure 4.3: Energy spectra versus wavenumber (κ) for the velocity field, with different numerical schemes using the $64^3$ grid (a) SST DDES (b) ϕ – f DDES

4.3 Closure

In this chapter the rationale for developing a hybrid RANS-LES model based upon elliptic relaxation has been presented. The specific development of the ϕ – f DDES model has been outlined together with its derivation and calibration using decaying isotropic turbulence. The sensitivity of the ϕ – f DDES and SST-DDES models to the numerical scheme has been demonstrated on a range of test cases. It has been shown that the use of a low dissipative convective scheme is crucial to obtaining accurate DDES results. Finally the computational expense of the ϕ – f DDES has been shown to be on average no more than 10% of the SST-DDES model.
Figure 4.4: Mean span-wise averaged skin friction coefficient ($C_f$) using different numerical schemes for the 2D-wall mounted hump (a) SST DDES (b) $\varphi - f$ DDES

Figure 4.5: Mean span-wise averaged skin friction coefficient ($C_f$) using different numerical schemes for the 2D periodic hills for the (a) SST DDES (b) $\varphi - f$ DDES
Chapter 5

2D Periodic hills

5.1 Introduction

The flow over a periodic arrangement of hills whose shape is defined in Fröhlich et al. [47] is investigated. The 2D computational domain is shown in Figure 5.1 where \( h \) is the hill height and \( L_h = 1.93h, L_x=9h, L_y=3.06h \) and the span-wise domain is \( L_z=4.5h \). Two Reynolds numbers were investigated; \( Re = 10590 \) and \( Re=37000 \) based on the hill height and the bulk velocity at the crest of the hill. The case has been investigated extensively because of the strong flow separation observed after the hill and the challenge it presents to turbulence modelling strategies in spite of the simplicity of the geometry. This test case was studied in the 9th and 10th ERCOFTAC workshops [94, 95] which established the performance of a range of RANS models. More recently the performance of several hybrid RANS-LES models was investigated by Šarić et al. [96]. Three refined LES studies have also been conducted and which are used as reference data for this study [47, 97] and a detailed analysis of this geometry for a range of Reynolds numbers, where DNS and LES computations were compared against two sets of experimental data [98].

This flow exhibits highly unsteady features that occur over a range of frequencies [47]. The entire flow is dominated by large scale turbulent structures around the point of separation and which has been shown to give rise to a low frequency oscillation of the separation point. These features render the accurate simulation of this flow beyond the capabilities of the majority of RANS models. Eddy viscosity and second moment closure RANS models fail in general to predict the correct level of turbulent shear stress and overall turbulence in the separated
shear layer, which results in an over-predicted recirculation region and thus a delayed reattachment point [6]. The results from three RANS models are shown in Figure 5.2 (SST, SA and $\varphi - f$), which demonstrates that while providing a reasonable qualitative prediction, none is able fully to capture the correct separation and reattachment points. The strong stream-wise curvature effects and the proximity of the wall make this a challenging and thus suitable test case to assess the performance of hybrid RANS-LES approaches that provide an alternative to RANS modelling.

5.2 Computational grid and boundary conditions

The mesh used is structured and contains 1.56 million cells (160×160×60). This mesh was set as the mandatory mesh for the European ATAAC project (Advanced Turbulence Simulation for Aerodynamic Application Challenges) [56] and is suitable refined near to the wall to ensure that $y^+ < 1$. Periodicity is set in the stream-wise and span-wise directions and a no-slip wall condition is applied to the top and bottom walls. The time step is set to $\Delta t U_b / h = 0.005$ for each simulation to ensure a CFL number of less than one. Each simulation was run for a total of 200 convective units; time-averaging began after the initial 50 units.

Figure 5.1: Case setup for the 2D periodic hills
5.3 Results

5.3.1 $Re=10590$

Mesh and reference data analysis

In order to assess the suitability of the mesh, Best Practice Guidelines (BPG) are used; spatial dimensions of the wall-adjacent cells, the ratio of the filter width to the Kolmogorov length scale and the ratio of modelled turbulent viscosity to molecular viscosity. For the spatial dimensions of the wall-adjacent cells; the suggested values for a wall-resolved LES (according to [45]) are: $y^+ < 2$, $\Delta x^+ = 50 - 150$, $\Delta z^+ = 15 - 40$. It can be seen from Figure 5.3 that the values on this mesh are well within the recommended range in each cell direction. This mesh is therefore sufficiently fine enough for a DDES computation where the near-wall cells are treated in RANS mode and indeed could be coarser than the requirements set out in Piomelli et al. [45]. Although it would be of interest to see the performance of the DDES models on a grid more suited to DDES (i.e. coarser in the near-wall than one designed for wall-resolved LES), the primary purpose of this test case is to validate the implementation of each of the DDES formulations and therefore the refinement of this grid is not as important.

A useful measure of the ability of the grid to resolve structures in the interior of the flow is the ratio $\Delta/\eta$, where $\Delta$ is the filter width and $\eta$ is an estimate of the Kolmogorov length scale ($= (\nu^3/\varepsilon)^{1/4}$). The dissipation rate is taken from the modelled dissipation equation, $\varepsilon$ because of the lack of a turbulence energy-budget (following work by Šarić et al. [46]). As reported in Fröhlich et al. [47] the ideal value for this ratio for a LES is $\Delta/\eta < 12$, in order that a suitable amount of the dissipation range of the spectrum is resolved. Figure 5.4 shows that at selected profiles along the course of the domain, the value for $\Delta/\eta$ is close to this ideal range. This illustrates that the grid is suitable for when the LES mode of DDES is active.

A final indicator for the resolution of the mesh is the ratio of the modelled turbulent viscosity to the molecular viscosity. This gives an indication of the ratio of the modelled and resolved contributions to the dissipation [47]. The range of values shown in Figure 5.5 suggests that the contribution of the RANS modelled turbulent viscosity is relatively low, suggesting that most of the flow is resolved. This is reflected in the RANS to LES length scale ratio shown in Figure 5.11.
where the majority of the flow is in LES model.

The results from the DDES simulations are compared to three LES simulations (labelled as LES. TL \[99\], LES. MFR \[100\] and LES. B \[98\]), and also a set of experimental data \[101\]. The LES computations of Temmerman et al. \[99\] and Mellen et al. \[100\] were run on the same mesh \((196 \times 128 \times 186)\), but with two different LES sub-grid scale models; the wall-adapted local eddy-viscosity (WALE) sub-grid scale model \[99\] and the Dynamic Smagorinsky model (DSM) \[100\]. The third and most recent LES computation by Breuer et al. \[98\] was performed on a much finer grid of approximately 13 million cells where the upper wall was also well resolved, and this was performed using the DSM sub-grid scale model.

From Figures 5.6 & 5.7 it can be seen that for the mean stream-wise velocity and turbulent shear stress all LES results produce largely similar results with only slight differences for the maximum shear stress values. While the skin friction coefficient results (Figure 5.8) do show a difference in the prediction of the reattachment point, this is not reflected in the rest of the flow. As the LES of Breuer et al. is substantially finer than the other studies, it is this LES data that will be used to compare against the DDES models.

Unfortunately the skin friction coefficient is not calculated in the experimental results, although the reattachment point was measured (Table 5.1). There is a noticeable difference between the reattachment point predicted by the experimental data and the LES data, which is partly explained by the difficulty in obtaining a mean measurement using PIV and LDA \[101\]. However, the local maximum of the stream-wise velocity near the wall at \(x/h = 0.05\) is higher for the experimental data (Figure 5.6(a)), resulting in greater momentum in the separation region which appears to make the flow reattach earlier than the LES data (the near-wall velocity is higher at \(x/h = 3\) and \(x/h = 6\)).

**DDES results: Introduction**

For the majority of the flow, all DDES formulations predict the correct level of turbulent shear-stress and therefore a similar velocity profile to that of the reference LES data \[98\]. There is little variation between the three DDES variants apart from in the region \(0 < x/h < 1\) (Figure 5.9). Since all the models are in ‘LES’ mode for the majority of the flow (except the boundary layer) this behaviour is expected. A closer analysis at several profiles along the domain will now follow.
DDES results: $x/h = 0.5$

At the first profile position, $x/h = 0.5$, there is a considerable amount of turbulence generation within the boundary layer at the crest of the hill. There is a strong peak of the stream-wise Reynolds stress $u' u'$ (Figure 5.19(a)) in this region due to the acceleration up the windward slope of the hill, which causes a high level of shear strain [47]. Comparing the relative size of each Reynolds stress (via Figures 5.19 and 5.20), it is clear that there is a large level of anisotropy in this region (the ratio $u' u'/v' v'$ reaches an order of 20 in this area). This anisotropy is well predicted by the DDES models, with $u' u'$ and $w' w'$ (Figures 5.19(a) & 5.21(a)) matching closely the reference LES data with little to no difference between each DDES model. The stream-wise velocity also has excellent agreement with the reference LES data (Figure 5.16(a)). There is however an over-prediction of the transverse velocity (Figure 5.17(a)) by all the models (especially the SA DDES model) and also the $v' v'$ stress is slightly under-predicted (Figure 5.20(a)).

The over-prediction of the transverse velocity will be influential in the prediction of the reattachment point as this effectively sets the angle of the streamlines from the crest of the hill.

The shear stress reaches a local minima in the boundary layer in keeping with the sharp stream-wise velocity gradient, which then drops to almost zero as the stream-wise velocity (Figure 5.16(a)) reaches its local maximum. This is in line with the eddy viscosity concept, as noted in Fröhlich et al. [47]. Outside of the boundary layer the same isn’t true. The velocity gradient changes sign and reaches a local minima at $y/h = 1.6$, however the shear stress does not change sign and instead reaches its maximum at this point. This implies stress-transport effects, that together with the high levels of anisotropy, goes some way to explain why linear eddy viscosity RANS models fail for this type of flow. The near-wall peak is slightly under-predicted by each of the DDES models but then apart from a slight over-prediction at $y/h = 1.5$ for the SA DDES model, each DDES model predicts the same shear stress as the reference LES data.

With such a refined grid the DDES formulations are effectively acting as a wall-modelled LES; the RANS region is limited to a thin near-wall region and the rest of the flow is resolved on a LES quality mesh. It is therefore not surprising that each formulation shows close agreement with the LES data.
DDES results: $x/h = 3.0$

At the second profile position, $x/h = 3.0$, the flow below $y/h = 0.5$ is subject to a recirculation region where the stream-wise and transverse velocity components exhibit a negative flow pattern (Figures 5.16(b) & 5.17(b)). The transverse velocity is predicted correctly by each of the DDES models but the strength of the flow reversal is slightly under-predicted by the SST DDES and to a lesser extent also the $\varphi - f$ DDES and SA DDES models (5.16(b)). Above the separated shear layer the stream-wise velocity increases until the upper wall boundary layer. The shear stress, shown in Figure 5.18(b), is over-predicted by the SA-DDES model but still matches the LES reference for the rest of the profile. The $\varphi - f$ DDES models provides the most accurate prediction for the shear stress at this point in the flow. The level of anisotropy is much lower in this region with the ratio $\overline{u'u'}/\overline{v'v'}$ being of the order of 1.3 [47]. The $\varphi - f$ DDES model predicts the results will more closely match the LES data for each of the Reynolds stresses (Figures 5.19(b), 5.20(b) and 5.21(b)), with the SST DDES and SA DDES both over-predicting these particularly in the near-wall regions, where the local maximum is too large.

The improved prediction of the near-wall region (which is covered by the RANS mode of DDES) by the $\varphi - f$ DDES model can be attributed to the improved near-wall modelling of the $\varphi - f$ model. The inclusion of a separate transport equation for $\varphi$ and $f$ allows the model to more accurately account for the anisotropy in the near-wall region compared to the ad-hoc damping functions used in the SST and SA models.

DDES results: $x/h = 6.0$

The next profile position, $x/h = 6.0$ is the post-reattachment region which is characterised by a developing boundary layer and the wake from the upstream separated shear layer. The SST-DDES model predicts the correct stream-wise velocity outside of the boundary layer but within the boundary layer it over-predicts its strength (Figure 5.16(c)) which was also seen at $x/h = 3$, this is linked to the over-prediction of the span-wise Reynolds stress upstream of this point (Figure 5.21(c)), resulting in a higher level of turbulent mixing and a shorter reattachment point (Figure 5.9). For the SA and $\varphi - f$ DDES models they show closer agreement with the LES data than the SST DDES model. There is a noticeable difference between the DDES models for the transverse velocity
(Figure 5.17(c)), however the relative magnitude of the velocity in this direction is very small compared to the stream-wise component and thus its effect should be minimal on the overall flow.

In the separated shear layer region the stream-wise and span-wise stresses are under-predicted by all the DDES models (Figures 5.19(c) & 5.21(c)). As with all these profiles positions the difference between the DDES models and LES results are small and could be easily explained by a slightly too diffusive numerical scheme or the coarseness of the grid.

**DDES results: \( x/h = 8.0 \)**

The final profile position, \( x/h = 8.0 \) is where the flow is subjected to strong acceleration on the windward face of the hill. This is reflected in the stream-wise velocity (Figure 5.16(d)), where a greater magnitude is observed near the wall. Even though this is the region of the grid where the resolution is coarsest (Figure 5.3), all the DDES models predict the mean velocity and stresses well, matching the reference LES data closely. The stream-wise stress is slightly under-predicted (Figure 5.19(d)), which helps to explain why it is also under-predicted at \( x/h = 0.05 \). Similarly to \( x/h = 0.05 \), very high levels of anisotropy are observed close to the wall, where the span-wise stress is considerably larger than both \( u'u' \) and \( v'v' \) (Figure 5.21(d)). This reason for this anisotropy is not clear but was attributed to the action of splatting in [47], and also confirmed in [101].

**DDES results: Conclusions**

Overall the DDES models have performed well on this test case, predicting the correct level of turbulence in most of the flow and matching the LES reference data closely. The DDES models were performed on a grid of 1.5 million cells (160x160x60), whereas the LES computations range from 4.6 to 13 million cells, which suggests in this case that hybrid RANS-LES methods can offer an attractive alternative to LES. It could be argued that even 1.5 million cells is more than fine enough for a DDES especially in the near-wall regions where a RANS model is applied and thus does not need as fine a grid as wall-resolved LES. The high levels of anisotropy and deviations from the eddy viscosity concept also illustrate why standard linear eddy viscosity RANS models perform so poorly and necessitate the need for more advanced modelling strategies. Coarser meshes will
be evaluated in future work but as the main purpose of these calculations were to validate the DDES formulations, other meshes were not used in this thesis.

5.3.2 Re=37000

Mesh and reference data analysis

The same geometry and mesh was also used to simulate flow at \( Re = 37000 \) to analyse the effect of the Reynolds number and to test the effectiveness of DDES where the grid is coarser (relative to this higher Reynolds number).

There is no LES data at this higher Reynolds number with which to compare. However a computation was completed by Manhart et al. [102] in which several simulations were conducted using hybrid RANS-LES and LES models on a range of meshes. Using the same mesh as described earlier, Manhart et al. [102] obtained very similar results to those described in this section, which serves as further validation of our calculations.

For the same profile positions introduced for the lower Reynolds number test case, the \( x^+, y^+ \) and \( z^+ \) values are noticeably larger at this Reynolds number (Figure 5.22), approximately three times that found at \( Re = 10590 \). In order to achieve the same resolution as the lower Reynolds number case, a mesh approximately three times the cell count would be required, which would mean a mesh closer to 5 million cells. It is informative to see how the DDES models behave on a coarser grid which would normally be required for a wall-resolved LES. DDES was ultimately conceived to be used in situations where a wall-resolved LES would be too costly in terms of CPU time (resulting from a finer mesh).

In a similar fashion to \( x^+ \), the ratio \( \Delta / \eta \) is much higher at \( Re = 37000 \) (Figure 5.23). The maximum value has increased from approximately 16 to 45, which is in line with the Reynolds number increasing by a similar ratio. Clearly this will have some influence of the resolution of the smallest scales and from these figures alone, it can be hypothesised that the amount of resolved turbulence (and thus the accuracy of the DDES) will be reduced. The ratio \( \nu_t / \nu \) has similarly increased by the same factor (approximately 3, as seen in Figure 5.24), thus inferring a greater part of the turbulence is coming from modelled turbulence (i.e. the sub-grid scale model, which in this case is the underlying RANS model).
**DDES results: Introduction**

Unfortunately the experimental data [101] does not include the skin friction coefficient along the bottom wall, but the reattachment point is given in Table 5.2. It was shown for $Re = 10590$ that the experimental reattachment was somewhat different to both the LES computations, so one might expect the experimental reattachment point also to have a certain amount of uncertainty involved, considering the challenge of correctly identifying the mean reattachment point in a very unsteady flow. Nevertheless, the trend of a shorter recirculation zone is observed by the experiment and DDES models (Figure 5.25), approximately 10% shorter. Rapp et al. [101] observe that this shortening of the recirculation region cannot be explained by increased turbulent mixing, as the turbulent Reynolds stresses are smaller at this higher Reynolds number (Figures 5.29, 5.30 and 5.31). The authors of [101] note that the streamlines were observed to be pointing downwards above the crest at $Re = 37000$ but pointing upwards at $Re = 10590$, which would then impact on the resulting recirculation and separated shear layer [101].

**DDES results: $x/h = 0.05$**

At the first profile position, $x/h = 0.05$, the same near-wall peak of the stream-wise velocity is observed (Figure 5.26(a)), but at $Re=37000$ it is stronger than at $Re=10590$ and the transverse velocity is smaller (Figure 5.27(a)), suggesting a flatter velocity profile which as explained earlier would help to produce a shorter recirculation zone. The DDES models do not match the experimental data as well as for $Re=10590$, with all the models over-predicting the transverse velocity and under-predicting the near-wall peak of the stream-wise velocity, thus showing flow features of a lower Reynolds number flow, most likely due to the coarser grid. The simulations of Manhart et al. [102] also failed to capture the Reynolds number sensitivity of this flow and under-predicted the velocity over the rest of the hill and the subsequent recirculation zone.

The levels of turbulence in general are over-predicted by all the DDES models, with all profiles of Reynolds stresses and shear stress exhibiting over-predicted values (Figure 5.28(a), 5.29(a), 5.30(a) and 5.31(a)). There is little or no difference between the DDES models (as the RANS region is still confined to a small near-wall region), which backs up the idea that the grid is causing the differences between computation and experiment. It is worth noting however that the experimental data at this higher Reynolds number is not as accurate in terms of its
spacial resolution, so the experimental results are also likely to contain a larger amount of error (as was also suggested at $Re=10590$ when compared to a very fine LES computation). Compared to the lower Reynolds number, at $Re=37000$ the near-wall peak of the stream-wise and span-wise Reynolds stresses are smaller.

**DDES results: $x/h = 3.0$**

At $x/h = 3.0$ the flow is still within the recirculation zone however the transverse velocity has a greater downwards component (Figure 5.27(b)) and the magnitude of the flow reversal appears to be weaker (Figure 5.26(b)) suggesting the flow has greater momentum here (originating from the higher velocity at the crest of the hill). The Reynolds stresses and turbulent shear stress are smaller at this higher Reynolds number, which is in line with the reduced values at $x/h = 0.05$. Once again the DDES models all over-predict the Reynolds stresses and shear stress at this profile position (Figures 5.28(b), 5.29(b), 5.30(b) and 5.31(b)). Similarly to $Re=10590$, the SST-DDES over-predicts the near-wall peaks of the stream-wise and span-wise stresses, which appear not to be related to the mesh, but rather to the near-wall modelling of the underlying SST-RANS model. The most notable deviation from the experimental data is for the wall-normal Reynolds stress (Figure 5.30(b)), for which values, all DDES models perform an over-prediction of nearly 50%. While SST-DDES and $\varphi - f$ DDES models predict the reattachment point, this is likely fortuitous due to inaccuracies in the determination of the experimental reattachment point; since it seems clear from the profiles of the mean velocity and turbulent Reynolds Stresses that the DDES results over-predict the recirculation zone.

**DDES results: $x/h = 6.0$**

By $x/h = 6.0$ the flow has already reattached and the profiles of the Reynolds stresses appear a similar to $Re=10590$ case, though roughly 20% smaller in magnitude. All DDES models over-predict the shear stress and Reynolds stresses (Figures 5.28(c), 5.29(c), 5.30(c) and 5.31(c)), just as reported in Manhart et al. [102]. The main differences between the DDES models is in the near-wall region where the RANS mode is in operation. As the majority of the flow is in LES mode because of the fineness of the grid, the differences between the DDES models outside of this near-wall is expected to be low (just as for the $Re=10590$ case).
DDES results: Conclusions

In summary, while at $Re = 10590$ the agreement of the DDES models to the LES studies and experimental data is good, for $Re = 37000$ the Reynolds number sensitivity observed in the experiment is not reproduced with LES [102] or the DDES models from this study. However as both the DDES models and LES computations of Manhart et al. [102] agree using reasonably fine grids (1.5 million in this study to 4 million cells in Manhart et al. [102]), it would be informative to run a computation using a mesh of similar resolution to that of Breuer et al. [98] which for $Re=37000$ would likely be close to 50 million cells.

Figure 5.2: Mean skin friction coefficient ($C_f$) for the SST, $\varphi - f$ and Spalart-Allmaras RANS models on the 2D periodic hills for (a) $Re = 10590$ and (b) $Re = 37000$

Table 5.1: Reattachment points for the 2D periodic hill ($x/h$)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>10590</td>
<td>4.21</td>
<td>4.72</td>
<td>4.56</td>
<td>4.69</td>
</tr>
<tr>
<td>37000</td>
<td>3.76</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
</tbody>
</table>

Table 5.2: Reattachment points for the 2D periodic hill ($x/h$)

<table>
<thead>
<tr>
<th>Re</th>
<th>Exp [101]</th>
<th>LES. B [98]</th>
<th>SST DDES</th>
<th>$\varphi - f$ DDES</th>
<th>SA DDES</th>
</tr>
</thead>
<tbody>
<tr>
<td>10590</td>
<td>4.21</td>
<td>4.69</td>
<td>4.20</td>
<td>4.42</td>
<td>4.56</td>
</tr>
<tr>
<td>37000</td>
<td>3.76</td>
<td>n/a</td>
<td>3.83</td>
<td>3.82</td>
<td>4.05</td>
</tr>
</tbody>
</table>
Figure 5.3: Wall-adjacent cell size along the bottom wall of the 2D periodic hills at Re=10590

Figure 5.4: Ratio of filter width to Kolmorogov scale: $\Delta/\eta$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590
Figure 5.5: Mean turbulent viscosity ratio: $\nu_t/\nu$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

Figure 5.6: Mean span-wise averaged stream-wise velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 2.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590 using the reference LES and experimental data
Figure 5.7: Mean span-wise averaged turbulent shear stress profiles at [a] \( x/h = 0.05 \) [b] \( x/h = 2.0 \) [c] \( x/h = 6.0 \) [d] \( x/h = 8.0 \) for the 2D periodic hills at \( Re = 10590 \) using the reference LES and experimental data.

Figure 5.8: Mean span-wise averaged skin friction coefficient \( (C_f) \) for the reference LES and experimental data on the 2D periodic hills at \( Re = 10590 \)
Figure 5.9: Mean span-wise averaged skin friction coefficient ($C_f$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D periodic hills at $Re = 10590$. 
Figure 5.10: Mean span-wise averaged pressure coefficient ($C_p$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D periodic hills at $Re = 10590$

Figure 5.11: The location of the DDES zones (0=RANS 1=LES) and the blending function $f_d$ for the SST DDES model
Figure 5.12: Instantaneous velocity shots using the \( \varphi - f \) DDES model which highlights the alternating separation and reattachment across the flow at \( \text{Re}=10590 \)

Figure 5.13: Mean streamlines for the SST DDES model for the 2D periodic hills at \( \text{Re}=10590 \)

Figure 5.14: Mean streamlines for the \( \varphi - f \) DDES model for the 2D periodic hills at \( \text{Re}=10590 \)
Figure 5.15: Mean streamlines for the SA DDES model for the 2D periodic hills at Re=10590

Figure 5.16: Mean span-wise averaged stream-wise velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

Figure 5.17: Mean span-wise averaged transverse velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590
Figure 5.18: Mean span-wise averaged turbulent shear stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=10590$

Figure 5.19: Mean span-wise averaged turbulent $\overline{u'v'}$ Reynolds’s stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=10590$
Figure 5.20: Mean span-wise averaged turbulent $v'v'$ Reynolds stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590

Figure 5.21: Mean span-wise averaged turbulent $w'w'$ Reynolds stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=10590
Figure 5.22: Wall-adjacent cell size along the bottom wall of the 2D periodic hills at Re=37000

Figure 5.23: Ratio of filter width to Kolmogorov scale: $\Delta/\eta$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at Re=37000
Figure 5.24: Mean turbulent viscosity ratio: $\nu_t/\nu$ at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$

Figure 5.25: Mean span-wise averaged skin friction coefficient ($C_f$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D periodic hills at $Re = 37000$
Figure 5.26: Mean span-wise averaged stream-wise velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$.

Figure 5.27: Mean span-wise averaged transverse velocity profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$. 

128
Figure 5.28: Mean span-wise averaged turbulent shear stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$

Figure 5.29: Mean span-wise averaged turbulent $u'u'$ Reynolds’s stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$
Figure 5.30: Mean span-wise averaged turbulent $v'v'$ Reynolds stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$

Figure 5.31: Mean span-wise averaged turbulent $w'w'$ Reynolds stress profiles at (a) $x/h = 0.05$ (b) $x/h = 3.0$ (c) $x/h = 6.0$ (d) $x/h = 8.0$ for the 2D periodic hills at $Re=37000$
Chapter 6

NACA0021 at 60° incidence

6.1 Introduction

The flow over a symmetric NACA0021 airfoil at 60° incidence for a Reynolds number of $Re_c = 2.7 \times 10^5$ (based on the chord length $c$ and the free-stream velocity $U_\infty$) is investigated. The flow at this angle of attack is post-stall and exhibits highly unsteady flow features; a flow DES was originally designed for. The first real application of DES was for a similar airfoil (NACA0012) at a similar high angle of attack [103], and the success of this simulation helped to spread the use of DES to a wider audience [104]. This test case has since become one of the most widely used test cases to evaluate DES variants and improvements [52, 85, 88, 104–108]. This test case was examined experimentally by Swalwell et al. [109] who collected the most extensive set of experimental data for this airfoil to date. This included mean flow data such as mean span-wise averaged pressure, lift and drag coefficients, as well as time histories of the lift and drag coefficient and their respective spectra. This allows for a much more in-depth analysis of hybrid RANS-LES methods than using purely mean force loads which was previously only available for such an airfoil.

This test case was extensively examined during the DESider [6] project, where a range of DDES and SAS models were investigated. This project examined the sensitivity to the time step, span size of the domain, grid refinement and underlying RANS model. The main conclusions from this project were that DES based methods gave superior mean flow predictions compared to URANS models, but that these DES results were strongly dependant on the time sample taken (at least 300-400 convective time units are required) and the span-wise extent of
the domain. There was consistent trend of the agreement with the experimental integral forces (lift and drag) becoming worse for a wider span-wise extent, but the PSD spectra becoming more in line with the experimental values. This led to Garbaruk et al. [104] concluding that the experimental data was a possible source of error (up to 12% variance was observed for the integral forces compared to similar experimental results).

Nevertheless this case still remains an excellent test case to validate new DDES formulations because of the large amount of previous computations to compare against.

6.2 Computational grid and boundary conditions

The O-type mesh (140×100) (Figure 6.1(b) & 6.1(a)) was supplied by New Technologies and Services (NTS) as part of the DESider project. In this study a span-wise domain of 1c was selected, resulting in a mesh of just over half a million cells (140×100×34). The free-stream turbulence intensity was set to the same level as the experiment; 0.6% and a turbulent viscosity ratio of \( \nu_t/\nu = 10 \) was used at the inlet. A non-dimensional time step of \( \Delta tU_0/c = 0.0025 \) (based on the airfoil chord and reference velocity) was used for all calculations which resulted in a CFL of less than 1 in the areas of resolved flow. Each simulation was run for a total of approximately 750 convective transit times (= \( tU_0/c \)). Averaging commenced after the initial condition transients had died away (typically 250 transit times). The far field domain is split into appropriate arcs for the inlet and outlet, and the airfoil itself is treated as a no-slip wall. The domain extends a span wise distance of one chord length and periodicity is applied in this direction.

\[ \text{Figure 6.1: Mesh and domain for the NACA0021 airfoil} \]
6.3 Results

6.3.1 Mesh resolution

The current mesh size and span-wise domain were selected to reduce the computational requirements that occur due to the long time samples required. It was concluded in the DESider project [6] that this grid resolution was sufficient to capture the unsteady flow field (a 19.2 million cell unstructured grid gave very similar results), therefore while an analysis of the ratio \( \Delta/\eta \) and the cell size in wall units can be informative (as was demonstrated in the 2-D periodic hills section), actual mesh refinement checks such as those conducted in the DESider project [6] are more informative in this case. While a wider span-wise extent would have been more appropriate, consistent trends were observed for a wide range of different DDES models from \( 1c - 4c \) and as such some interpolation can be made from the \( 1c \) results of this study. Moreover as the principle purpose of this study is to compare DDES simulations, as long as a consistent span-wise extent is used for each model then the actual value itself is less important.

6.3.2 DDES results

DDES results: Global forces

Table 6.1 shows the global force results from each DDES formulation as well as those provided from the experimental data [109]. The \( \varphi - f \) DDES model matches more closely the experimental data than both the SST DDES and SA DDES results, as well as predicting a more accurate Strouhal number for the main frequency peak (Figure 6.3(b)). Each DDES model over-predicts the Power Spectral-Density (PSD) of the lift coefficient, but this is in line with the results from Garbaruk et al. [104], which relates this to the use of a narrower span-wise extent. The effect of moving to a span-wise extent of \( 4c \) was shown in Garbaruk et al. [104] to consistently lower the frequency across the spectrum which then matched the experimental values more closely.

Figure 6.2 shows the running time-averages for the lift and drag coefficients; it is clear that even after 500 convective units, a stationary mean is not present (however the error is less than 5\%). More importantly a clear trend is visible between the models and further time-averaging would not alter this, although it does highlight the need for long time samples.
Interestingly both the SST DDES and $\phi - f$ DDES models predict a similar $C_L/C_D$ value to the experiment which suggests a general offset in the flow prediction between the two methods rather than a fundamental difference. The SA-DDES model however under-predicts this figure by 20%, suggesting a smaller component of lift-induced drag and a fundamentally different flow field. The improved prediction in pressure distribution (Fig. 6.4(a)) for the $\phi - f$ DDES model is clearly manifested in the global lift coefficient. The SA-DDES model demonstrates close agreement to the experimental data for the pressure distribution, but this is not reflected in the global lift coefficient value. This suggests a larger contribution of the shear-stress to the lift coefficient for the SA DDES; which is observed in the skin-friction coefficient results (Figure 6.4(b)).

**DDES results: Skin-friction coefficient**

The skin-friction coefficient shown in Figure 6.4(b) shows the differing separation and reattachment points for the different models. The main difference is the small attached region on the upper surface $0.03 < x/c < 0.2$, in which both the $\phi - f$ DDES and SST-DDES show clearly attached flow whereas the SA DDES model only just reattaches. This difference here may effect the position of the separated shear layer and therefore the resulting separation region.

**DDES results: Turbulence levels**

The level of resolved turbulent kinetic energy is known to be over-predicted by the SST DDES model [3], which makes the $\phi - f$ DDES models and to a lesser extent the SA DDES predictions more accurate overall (Figure 6.5). The level of modelled turbulence is low for all the formulations showing that each formulation is in ‘LES’ mode (Figure 6.6).

The higher level of resolved turbulence for the SST-DDES model means there will be greater fluid mixing in the recirculation zone. This increased mixing brings more higher velocity fluid into the region and reduces the recirculation zone as seen in Figure 6.7(a). As there is higher velocity in this region compared to that produced by the $\phi - f$ DDES model, this results in a lower pressure on the upper surface (as seen in Figure 6.4(a)) and ultimately higher lift and drag (Table 6.1).

Without experimental data for the turbulent kinetic energy profiles it is not possible to fully explain these differing results, however it is a function of both the upstream flow conditions and the sub-grid scale turbulence model within
DDES. As each DDES model predicts a slightly different flow pattern in the initial separation region, these changes are most likely to be responsible for the differing resolved turbulence levels. The effect of the different $C_{DDES}$ constants in the sub-grid scale model of each DDES formulation may also have an effect on the resolved turbulence levels.

**DDES results: Flow structures**

The mean streamlines around the NACA 0021 airfoil are shown in Figure 6.7 for each of the DDES formulations. There are some differences in the size of the recirculation zones although it is not possible to make any firm conclusions without any experimental streamlines to compare against. Clearly a change in the size and strength of a recirculation would most likely have a strong influence on the lift and drag values.

Figure 6.8 shows an instantaneous view of the vorticity magnitude for each DDES formulations. For both the SST DDES and $\varphi - f$ DDES models the resolution of the turbulence is broadly similar, the structures from the vortex shedding are clear as well as the two shear layers from the leading and trailing edges. However for the SA-DDES model although the separated shear layers are present, the vortex structures are fewer which could explain the lower drag values compared to the size of the lift coefficient. The position of the leading edge shear layer is also different to that of both the SST DDES and $\varphi - f$ DDES models which would also explain the differing lift-to-drag ratio.

**DDES results: Conclusions**

This case has served as a useful validation exercise for the $\varphi - f$ DDES model and has show its performance to be in-line with the results from the many previous studies \[52, 85, 88, 104-108\], and in this case superior to both the SST-DDES and SA-DDES models. This superior performance is related to level of the resolved turbulence and the prediction of the correct position of the separated shear layer, both of which are more accurately predicted by the $\varphi - f$ DDES model based upon previous computational studies.
Table 6.1: Lift and Drag coefficient results from DDES simulations and experimental data for the NACA0021 airfoil

<table>
<thead>
<tr>
<th></th>
<th>Exp</th>
<th>SST DDES</th>
<th>φ − f DDES</th>
<th>SA DDES</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_D$</td>
<td>1.55</td>
<td>1.80</td>
<td>1.59</td>
<td>1.50</td>
</tr>
<tr>
<td>$C_L$</td>
<td>0.93</td>
<td>1.11</td>
<td>0.98</td>
<td>1.11</td>
</tr>
<tr>
<td>$C_L/C_D$</td>
<td>1.66</td>
<td>1.62</td>
<td>1.62</td>
<td>1.35</td>
</tr>
<tr>
<td>$St$</td>
<td>0.20</td>
<td>0.18</td>
<td>0.20</td>
<td>0.20</td>
</tr>
</tbody>
</table>

Figure 6.2: Running time-average of the span-averaged (a) lift coefficient and (b) drag coefficient by the SST DDES, φ − f DDES and SA DDES models for the NACA 0021 airfoil.

Figure 6.3: Power Spectral Density (PSD) of the lift coefficient for the (a) SST DDES model, (b) φ − f DDES model and (c) the SA DDES model for the NACA 0021 airfoil.
Figure 6.4: Mean span-wise averaged \(C_p\) pressure coefficient and \(\varphi - f\) skin-friction coefficient by the SST DDES, \(\varphi - f\) DDES and SA DDES models for the NACA 0021 airfoil. The upper line represents the lower surface of the airfoil for the skin-friction coefficient and vice versa for the lower line.

Figure 6.5: Mean resolved turbulent kinetic energy (TKE) for the SST DDES model, \(\varphi - f\) DDES model and SA DDES model for the NACA0021 airfoil.

Figure 6.6: Mean modelled turbulent kinetic energy (TKE) for the SST DDES model, \(\varphi - f\) DDES model and SA DDES model for the NACA0021 airfoil.
Figure 6.7: Mean velocity streamlines for the (a) SST DDES model, (b) $\varphi - f$ DDES model and the (c) SA DDES model for the NACA0021 airfoil

Figure 6.8: Instantaneous field of Vorticity for the (a) SST DDES model, (b) $\varphi - f$ DDES model and the (c) SA DDES model for the NACA0021 airfoil
Chapter 7

2D wall-mounted hump

7.1 Introduction

The turbulent flow over a wall-mounted, 2-D hump (Figure 7.1(a)) at a high-Reynolds number of $Re_c = 9.36 \times 10^5$ (based on a chord length of $c = 0.42$ m and a free-stream velocity of $U_\infty = 34.6$ m/s) was examined experimentally at the NASA Langley Research Centre [110] as part of the CFDVAL workshop on computational methods and turbulence models validation. This test case has also been studied at the 11th ERCOFTAC workshop on refined turbulence modelling [111] and the 12th ERCOFTAC/IAHR workshop on refined turbulence modelling [112]. It has also been investigated as part of the European ATAAC project (Advanced Turbulence Simulation for Aerodynamic Application Challenges) [56]. A detailed investigation has been carried out by Šarić et al. [46] where RANS, DES and LES models were investigated and more recently a comprehensive LES study was undertaken by Avdis et al. [113]. Both showed that DES and LES methods are more capable of resolving this unsteady flow than standard RANS models.

This flow is characterised by a largely geometry induced separation point ($x = 0.65c$). The flow exhibits a unsteady flow field dominated by large scale structures and a separated shear layer which must be modelled correctly to ensure the size and strength of the recirculation is correct. The work of the previous ERCOFTAC workshops showed that many RANS models under-predicted the level of turbulence in the separated shear layer and thus over-predicted the recirculation zone and reattachment point [112].
7.2 Computational grid and boundary conditions

Two structured meshes were investigated, which were provided by Chalmers University and New Technologies and Services (NTS) for the ATAAC project. The first mesh has a solution domain of $L_x \times L_y \times L_z = 6.14c \times 0.9c \times 0.2c$ which is meshed with 1638400 cells ($400 \times 128 \times 32$). The second mesh (Figure 7.1(b) & 7.2) has a greater concentration of cells close to the wall and has a solution domain of $L_x \times L_y \times L_z = 6.14c \times 0.9c \times 0.4c$ which is meshed with 2934976 cells ($379 \times 121 \times 64$). The second mesh was found to produce the best results for each model (Figure 7.1(b)) and as such all the results presented in this work were conducted using this mesh.

Although the experimental domain started at $-6.39c$ compared to the current $-2.14c$, work by Šarić et al. [46] shows that there is no significant difference
between the choice of the inlet position and thus it was decided to reduce the solution domain to the smaller size. The dimensionless wall distance of the nodes closest to the wall were $y^+ < 1$ for the lower wall and $y^+ = 30$ for the upper wall.

The oncoming flow is characterised by a zero-pressure-gradient turbulent boundary layer, whose thickness $\delta$ is approximately 57% of the hump height measured at the upstream extent of the domain (-2.14$c$), this corresponds to a momentum-thickness-based Reynolds number $Re_\theta = 7200$ [110]. The mean profiles of the velocity and turbulent quantities which were used as boundary conditions at the inlet were taken from a precursor computation. A no-slip boundary condition was applied at the bottom wall and a slip wall was applied to the top wall. Periodic boundary conditions were used along the span-wise direction. All cases ran using a non-dimensional time step of $\Delta t U_0/c = 0.001$ (based on the hump chord and reference velocity).

### 7.3 Results

#### 7.3.1 Mesh and reference data analysis

To ensure that the grid is capable of resolving enough of the flow when the model is in LES mode, the cell size in wall units for the wall-adjacent cells for each direction is calculated (Figures 7.4(a) & 7.4(b)). For a wall-resolved LES the suggested values are: $y^+ < 2$, $\Delta x^+ = 50 - 150$, $\Delta z^+ = 15 - 40$ [11]. Between $x/c = -2.14$ and $x/c = 0.6$ the flow remains attached and devoid of unsteady flow features which means the near-wall resolution requirements can be relaxed. The large values of $\Delta x^+$ and $\Delta z^+$ in this region (Figure 7.4(a)) are therefore not expected to degrade the accuracy of the flow in this region. However between $x/c = 0.6$ and $x/c = 1.4$ where the flow separates and reattaches (Figure 7.4(b)) the size of the wall-adjacent cells is influential to the accuracy of the simulation. A measure of the suitability of the grid to resolve the structures in the interior of the flow is the ratio $\Delta/\eta$, where $\Delta$ is the filter width and $\eta$ is an estimate of the Kolmogorogov length scale ($= (\nu^3/\varepsilon)^{1/4}$). The dissipation rate is taken from the modelled dissipation equation, $\varepsilon$ because of the lack of a turbulence energy-budget (in a similar fashion to Šarić et al [46]). As reported in Fröhlich et al. [47] the ideal value for this ratio for a LES is $= \Delta/\eta < 12$, in order that a suitable amount of the dissipation range of the spectrum is resolved. Figure 7.5 shows
that at selected profiles along the course of the domain, the value for $\Delta/\eta$ exceeds this ideal range and is around five times the size of the maximum value observed for the 2D periodic hills (Figure 5.4) and the ideal range. The 2D hump is being simulated at a much larger Reynolds numbers to that of the 2D periodic hills ($Re = 10590 \& Re = 37000$ for the 2D hills compared to $Re = 936000$ for the 2D hump), which means that the smallest structures become even smaller for a higher Reynolds number and thus requires more cells to capture them with the same resolution [12].

To achieve the same $\Delta/\eta$ ratio that was observed for the 2D periodic hills, the 2D hump would require approximately five times more cells in this region of the domain ($x/c = 0.6 - 1.4$), which would mean the total cell count increasing from approximately 3 million cells to around 10 million cells (considering the requirements of a well laid out structured mesh). As a DDES simulation is being performed rather than a wall-resolved LES any requirements can be relaxed, however this gives some explanation as to why at higher Reynolds numbers DDES can be an attractive alternative to LES.

A final indicator for the resolution of the mesh (and for DDES the relative activation of the LES and RANS modes) is the ratio of the modelled turbulent viscosity to the molecular viscosity. This gives an indication of the ratio of the modelled and resolved contributions to the dissipation [47]. The range of values shown in Figure 7.6 suggest that again the resolution of the grid is worse than that of the 2D periodic hills. The flow from $x/c = -2.14$ to $x/c = 0.6$ is steady and the boundary layer ($y/c = 0$ to $y/c = 0.08$) is treated in RANS mode ($L_{RANS} < L_{LES}$), which explains the large values of $\nu_t/\nu$. Between $x/c = 0.60$ and $x/c = 1.4$ the flow is largely in LES mode and thus the ratio $\nu_t/\nu$ reaches a maximum of 40. This value shows that the modelled components still contributes to the total turbulence level, which is due to the grid resolution being too coarse and also the history effect of the modelled turbulence from the upstream flow.

7.3.2 DDES Results

DDES results: Introduction

Figure 7.7(a) highlights the differences between the three DDES models in the separation region of the flow ($x/c = 0.6 - 1.4$). From this figure, the SA DDES model appears to offer the best prediction of the flow, with the correct
strength and length of the recirculation zone. The SST DDES model under-
predicts the strength of the recirculation and thus over-predicts the reattachment
point. The $\varphi - f$ DDES model captures the correct strength of the recirculation
region although its formation is slightly delayed compared to the experiment and
thus over-predicts the reattachment point.

**DDES results: Global flow structures and recirculation region**

The close agreement of the SA DDES model with the skin friction coefficient is
not reflected in the pressure coefficient (Figure 7.7(b)). Here the SST DDES and
$\varphi - f$ DDES models are closer to the experimental values, with the $\varphi - f$ DDES
showing a slightly better prediction than the SST DDES, which is in line with
the skin friction coefficient results discussed previously. The SA DDES model
over-predicts the pressure recovery at $x/c \approx 0.65$ and as a result moves towards
a favourable pressure gradient earlier than the other two models (which is not in
line with the experimental results). This move to a earlier favourable pressure
gradient helps to reattach the flow which may explain the good agreement with
the skin friction coefficient.

The region up to and including the immediate area around the separation is
key in determining the downstream flow. Even though the separation point is
largely dictated by the geometry, the strength and length of the recirculation is
influenced by the growth of instabilities around the separation point, and also
the level of turbulence that is present before and after this point.

Figures 7.8 and 7.9 show the skin friction and pressure coefficients for the
whole domain. There is no difference in the pressure coefficient up until the
separation point ($x/c = 0.65$), but there are noticeable differences in the skin
friction in the boundary layer leading up to the base of the hump ($x/c = -2.14 -
0$). This shows that the choice of the underlying RANS model in each DDES
formulation has an effect on the state of the flow as it separates over the hump.
Unfortunately there is no experimental data for the skin-friction coefficient before
the hump so it is not possible to conclude which model best predicts the flow in
this region.

Figures 7.13, 7.14 and 7.15 show the iso-Q contours of the flow between
$x/c = 0.6$ and $x/c = 1.5$ for the SST DDES, $\varphi - f$ DDES and SA DDES mod-
els respectively. These plots give some idea as to the state of the flow and the
separated shear layer. It can be seen from these figures that the $\varphi - f$ develops
finer structures over a shorter distance than the SST DDES model, which agrees with the skin-friction coefficient plots which show the SST DDES not capturing the strength of the recirculation in the initial separated shear layer. Ideally a DDES model will transform instantly from a RANS simulation to a LES simulation when the flow separates but in reality this does not occur and a certain time or distance is required for instabilities to form which trigger the development of LES content. Apart from the initial development of the flow, both the SST and $\varphi - f$ models show broadly similar flow structures in terms of size and shape. For the SA-DDES model the flow looks quite different with larger structures and higher velocities in the initial separated shear layer region. This agrees with the higher velocities observed in Figures 7.17(b) to 7.17(d), which are higher than the experimental values (and also those predicted by the SST and $\varphi - f$ DDES models).

For DDES, another factor in the treatment of the separated shear layer is the $f_d$ function of the DDES formulation which controls the selection of RANS and LES modes in the boundary layer. It was observed that each DDES formulation gave almost the same distribution (as shown in Figure 7.16), so although there might be small changes arising from a slight variation in the distribution of this function, it is not the primary factor in the differing results between the models.

DDES results: $x/c = 0.65 - 0.66$

At the point of separation ($x/c = 0.65$ and $x/c = 0.66$), the SA DDES model shows the best agreement for the stream-wise velocity (Figure 7.17(a) and to a lesser extent, Figure 7.17(b)) but over-predicts the transverse velocity by almost 50% compared to the experimental results (Figures 7.18(a) and 7.18(b)). The SST DDES model both under-predicts the stream-wise and transverse velocities at the crest of the hump, which is directly linked to the performance of the underlying RANS model prior to this point. In contrast to both the SST DDES and SA DDES models, the $\varphi - f$ DDES model captures close to the correct transverse velocity, but slightly under-predicts the stream-wise velocity compared to the SA DDES model. The turbulent shear stress, shown in Figures 7.19(a) and 7.19(b) is slightly over-predicted by the SA DDES model, and under-predicted by the SST DDES model. The $\varphi - f$ DDES model shows good agreement with the experiment for both profile points. As the turbulent shear stress at these points is mainly due to modelled turbulence (i.e a small resolved turbulence component)
the differences in the models can be attributed to the upstream flow which comes from the RANS mode of each DDES model. A similar observation can be made for the other Reynolds stress components shown in Figures 7.20(a) [7.20(b)] [7.20(c)] and [7.21(b)] where although none of the models match the experimental results, the same differences seen between the models for the turbulent shear stress are observed for the other components of the Reynolds Stress tensor. The implication of these results is firstly that the state of the shear layer at the point of separation is different for each DDES variant, which can be attributed to the performance of the underlying RANS model up to this point. Secondly and more importantly the flow will also separate with a different velocity magnitude for each model, that results in a different recirculation region. The SA DDES model has a much stronger downwards motion which may explain why the recirculation is shorter than both the $\varphi - f$ DDES and the SST DDES models. As the SST DDES has a higher transverse velocity and therefore a more upwards motion at $x/c = 0.65$, this may explain why the SST DDES has the longest recirculation length. These differences between the recirculation length for each model is visible in the mean streamlines plots shown in Figures 7.10, 7.11 and 7.12.

**DDES results: $x/c = 0.80 - 0.90$**

In the middle of the recirculation zone ($x/c = 0.80$ and $x/c = 0.90$), the SA DDES model over-predicts the stream-wise velocity in the area of the separated shear layer ($y/c = 0.08 - 0.12$) (Figures 7.17(c) and 7.17(d)) and also under-predicts the transverse velocity (Figures 7.18(c) and 7.18(d)). These figures show that the flow still has a greater downwards component compared to the experiment and other DDES models which results in greater momentum in the recirculation zone. The SST DDES shows a similar trend as to that seen at $x/c = 0.65$ and $x/c = 0.66$, in that it under-predicts the stream-wise velocity but over predicts the transverse velocity, which again explains the longer recirculation length. The $\varphi - f$ DDES is in between both the SST DDES and SA DDES models and therefore seems to match the magnitude of the velocity more closely, however it does not capture it correctly. The SA DDES over-predicts the turbulent shear stress by almost 50% at $x/c = 0.9$ (as shown in Figure 7.19(d)), and this is also true for the other components of the Reynolds stress tensor (Figures 7.20(d) and 7.21(d)). The effect of a larger turbulent shear stress value is to increase turbulence and therefore increase turbulent mixing which will result
in a shorter recirculation length. The SST DDES under-predicts the shear stress by close to 50% at \( x/c = 0.8 \) (Figure 7.19(c)), which helps to explain why the recirculation zone is over-predicted. The \( \varphi - f \) DDES comes close to matching the experimental data at \( x/c = 0.8 \) and \( x/c = 0.9 \) for each of the components of the Reynolds stress tensor, and shows a marked improvement over the SST DDES model in this respect. The \( \varphi - f \) model is better able to capture the near-wall Reynolds stress levels due to an additional transport equation for the wall-normal fluctuations which provides a better approximate of the anisotropy of the flow.

**DDES results: \( x/c = 1.0 - 1.1 \)**

In the region towards the point of reattachment (\( x/c = 1.0 \) and \( x/c = 1.1 \)) the same trend that has been observed throughout the flow is again seen; the SA DDES slightly over-predicting the stream-wise velocity and the SST DDES (and to a lesser extent) the \( \varphi - f \) DDES under-predicting the stream-wise velocity. The SA DDES captures the Reynolds stresses (Figures 7.24, 7.25 and 7.26) more accurately than the SST DDES and \( \varphi - f \) DDES models, although it does over-predict each component at \( x/c = 1.0 \), which was also seen previously at \( x/c = 0.8 \) and \( x/c = 0.9 \).

**DDES results: \( x/c = 1.2 - 1.3 \)**

In the post attachment region (\( x/c = 1.2 \) and \( x/c = 1.3 \)), as the SA DDES model reattaches at the correct location it shows very good agreement with the experimental data for both the stream-wise and transverse velocity (Figure 7.22(c), 7.22(d), 7.23(c) and 7.23(d)). In contrast to this both the SST DDES and \( \varphi - f \) DDES reattach too late and as a result the velocity profiles are over-predicted both in the stream-wise and transverse directions. The same pattern is true for the Reynolds stresses (Figures 7.24, 7.25 and 7.26), with the SA DDES showing good agreement with the experiment for each component. Both the SST DDES and \( \varphi - f \) DDES models over-predict the Reynolds stresses at these profile positions as the flow has not fully reattached and thus the level of turbulence is too great. For the \( \varphi - f \) DDES (and to a lesser extent, the SST DDES) the whole recirculation zone has been shifted in the stream-wise direction because of the delay in the formation of resolved turbulence in the separated shear layer. If each profile position for the Reynolds stresses and velocities were shifted backwards
they would agree well with the experimental data, thus it can be argued that the \( \varphi - f \) DDES captures the physics of the recirculation and reattachment process correctly, but the transition from RANS to LES content at the separation point affects the ability to fully capture the initial separated shear layer.

**DDES results: Conclusions**

This case demonstrates that even for a flow whose separation is largely geometry induced, the modelling of the flow upstream of the separation point (which is modelled in RANS mode) can have a significant affect on the size and strength of the recirculation zone. Capturing the correct state of the shear layer as it separates is crucial for the resulting flow. It is difficult to conclude which of the three variants performed best as the model that predicts the correct reattachment point (the SA DDES model) achieves this by incorrectly predicting the flow direction at separation which results in the correct recirculation length (for the wrong reason). It is the underlying RANS model that has made the major contribution to the differences seen between the three DDES model, which highlights the need for careful consideration of the underlying RANS model when selecting a suitable DDES model. For each of the DDES models there is still some delay in the production of LES content which can be attributed to the delay in the rise of resolved turbulence and a \( f_d \) function that extents into the separated shear layer (thereby delaying the transition to LES mode). Thus although the underlying RANS model has a strong influence on the accuracy of the flow prediction, any improvements beyond this point is also limited by the DDES formulation itself.

![Graph](image)

Figure 7.3: Mean span-wise averaged skin-friction coefficient for both 2D wall-mounted hump meshes using the \( \varphi - f \) DDES model.
Figure 7.4: Wall-adjacent cell size along the bottom wall of the 2D hump
Figure 7.5: Ratio of filter width to Kolmogorov scale: $\Delta/\eta$ at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.9$ (e) $x/c = 1.0$ (f) $x/c = 1.1$ (g) $x/c = 1.2$ (h) $x/c = 1.3$ for the 2D wall-mounted hump
Figure 7.6: Mean turbulent viscosity ratio: \( \nu_t/\nu \) at (a) \( x/c = 0.65 \) (b) \( x/c = 0.66 \) (c) \( x/c = 0.80 \) (d) \( x/c = 0.9 \) (e) \( x/c = 1.0 \) (f) \( x/c = 1.1 \) (g) \( x/c = 1.2 \) (h) \( x/c = 1.3 \) for the 2D wall-mounted hump.

Figure 7.7: Mean span-wise averaged skin friction coefficient (\( C_f \)) (a) and pressure coefficient (\( C_p \)) (b) for the SST-DDES, \( \varphi - f \) DDES and SA-DDES models on the 2D hump.
Figure 7.8: Mean span-wise averaged skin friction coefficient ($C_f$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D hump
Figure 7.9: Mean span-wise averaged pressure coefficient ($C_p$) for the SST-DDES, $\varphi - f$ DDES and SA-DDES models on the 2D hump.

Figure 7.10: Mean streamlines for the SST DDES model (top) and the SST DDES model (bottom) for the 2D wall-mounted hump.

Figure 7.11: Mean streamlines for the $\varphi - f$ DDES model for the 2D wall-mounted hump.
Figure 7.12: Mean streamlines for the SA DDES model for the 2D wall-mounted hump

Figure 7.13: Iso-Q contours coloured by velocity magnitude for the SST DDES model for the 2D wall-mounted hump

Figure 7.14: Iso-Q contours coloured by velocity magnitude for the $\varphi - f$ DDES model for the 2D wall-mounted hump
Figure 7.15: Iso-Q contours coloured by velocity magnitude for the SA DDES model for the 2D wall-mounted hump

Figure 7.16: $f_d$ function for the $\varphi - f$ DDES model on the 2D wall-mounted hump

Figure 7.17: Mean span-wise averaged stream-wise velocity profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.9$ for the 2D wall-mounted hump
Figure 7.18: Mean span-wise averaged transverse velocity profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.9$ for the 2D wall-mounted hump

Figure 7.19: Mean span-wise averaged turbulent shear stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump
Figure 7.20: Mean span-wise averaged turbulent $\overline{u' u'}$ Reynolds’s stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump.

Figure 7.21: Mean span-wise averaged turbulent $\overline{v' v'}$ Reynolds’s stress profiles at (a) $x/c = 0.65$ (b) $x/c = 0.66$ (c) $x/c = 0.80$ (d) $x/c = 0.90$ for the 2D wall-mounted hump.
Figure 7.22: Mean span-wise averaged stream-wise velocity profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump

Figure 7.23: Mean span-wise averaged transverse velocity profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump

Figure 7.24: Mean span-wise averaged turbulent shear stress profiles at (a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump
Figure 7.25: Mean span-wise averaged turbulent $u'\ u'$ Reynold’s stress profiles at
(a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump

Figure 7.26: Mean span-wise averaged turbulent $v'\ v'$ Reynold’s stress profiles at
(a) $x/c = 1.0$ (b) $x/c = 1.1$ (c) $x/c = 1.2$ (d) $x/c = 1.3$ for the 2D wall-mounted hump
Chapter 8

Ahmed car body

8.1 Introduction

Computational fluid dynamics has increasingly become an important design tool for the automotive industry to supplement experimental studies. With a desire to reduce noise emissions and improve fuel efficiency, reliable CFD simulations of the complex turbulent flow around vehicles is becoming an ever more important goal. Validating CFD methods on an actual car geometry however is computationally expensive and any conclusions are likely to be dependant on the individual car geometry. The Ahmed car body [89] represents a generic car geometry with a slanted back and a flat front. While it is a much simplified version of a real car, it nevertheless exhibits many of flow features found in real-life cars such as the complex vortex interactions that occur in its wake.

This geometry was first proposed and studied by Ahmed et al. [89] (Figure 8.1): The wake behind the car body is a result of the interaction between the counter-rotating vortices produced by the slant side edges and the separated flow over the rear of the body. The angle of the slant back section was found to be influential in the structure of the wake and the reattachment point. At 35° the counter-rotating vortices are weaker, which results in the flow being completely separated over the entire slant back of the vehicle. At 25° the counter-rotating vortices are strong enough to help to bring enough momentum into the separation region to reattach the flow half way down the slant back.

More recently this geometry has also been investigated by Lienhart et al. [4], who performed a more detailed experimental study (at a lower Reynolds number), which included LDA measurements of the mean and fluctuating velocities as well
as on-surface oil flows. This was performed for slant back angles of both 25° and 35°, and the results confirmed many of the observations that were made by Ahmed in their initial investigation.

This case has been the focus of several CFD investigations, most notably the 9th and 10th ERCOFTAC workshops on refined turbulence modelling [94, 95], where a range of RANS models were investigated, and also the DFG-CNRS program: LES of Complex Flows [2], where several LES and DES formulations were evaluated. It has also been studied within the FLOMANIA [49] and DESider [6] EU projects, where a range of RANS, LES and hybrid RANS-LES models were investigated.

The general conclusions from the studies involving RANS models [49, 94, 95] were dependant on the slant angle. At 35° (where the separation occurred along the entire slant back) most of the RANS approaches (both simple linear $k-\varepsilon$ and more complex Reynolds stress models) captured the correct level of the turbulent stresses and, as a result, showed good agreement with the experimental results for the separation and reattachment points. However at 25° (where the complex interaction between the counter-rotating vortices and the separated flow results in a shorter separation region) the majority of RANS models, regardless of mesh refinement or wall treatment, failed to predict the flow correctly. In general, they either failed to predict separation completely, or even when they did, they did not predict the correct separation point and thus were unable to capture the correct size of the recirculation region due to an under-prediction of the turbulent stresses [2].

A number of LES studies have more recently been performed on this case, mainly at the more challenging 25° slant angle [2, 6, 114, 116]. These studies were performed with a range of sub-grid scale models and wall-treatments and, while some were more successful than others, many failed to capture fully the correct recirculation region. The high-Reynolds number ($Re = 7.68 \times 10^5$) meant that even with meshes of up to 48 million cells, the resolution requirements were still not ideal for a well resolved LES. One conclusion from Serre et al. [2] was that due to the high cost of a well resolved LES, hybrid RANS-LES methods represent an attractive alternative, even if the mesh generation requires greater care with regards to the RANS-LES zones.

During the DESider project [6], the majority of the turbulence models tested were DES or DDES approaches based upon either the SST or SA underlying
RANS models. Each of these methods were applied to the more challenging 25° case with considerably coarser grids than were used for previously discussed LES computations (3-5 million cells). None of the methods showed good agreement with the experimental data through-out the flow and it was concluded that this case is sensitive to the underlying RANS model. It was also concluded that the flow may be very sensitive to small changes in the computational setup, such as the grid or inlet conditions [6]. The conditions of the flow just as it moves over the slant is crucial to the development of the separated shear layer and the strength of the counter-rotating vortices. It is the underlying RANS model that is largely responsible for the flow up to this point; therefore the choice of the RANS model within the hybrid RANS-LES approach is important.

Other computations have been made with DES [2, 3, 117] (SA-DES and SST-DDES) however none of these were able correctly to match the experimental data. This may be attributed to the mesh resolution, underlying RANS model or the numerics of the CFD code used. As was seen with both the RANS and LES studies, all the DES results reported good agreement with the 35° case, which suggests that the 25° slant angle is the most interesting geometry to investigate further.

8.2 Computational grid and boundary conditions

The car body was mounted on four stilts 50mm above the ground in the experiment to model the effect of the car’s height with wheels (and thus capture the important ground effect). However to aid the mesh generation in the CFD simulations these stilts were not used and thus the body is fixed at 50mm above the floor. Only the 25° slant back angle has been investigated in this study, as it is this angle that produces the most challenging flow features; the strong influence of the counter-rotating vortices on the reattachment point. The body has a length of $L = 1044\,mm$, a height of $H = 288\,mm$ and a width of $W = 389\,mm$. Two meshes were used; firstly, a structured fine mesh of 2.7 million cells (Figure 8.2) and secondly, a finer mesh of 4.4 million cells with greater refinement in the wake and separation locations (Figure 8.3). The flow is at a Reynolds number of $Re = 768,000$ based on the body height $H$ and the free-stream velocity $U_{\infty} = 40\,ms^{-1}$. An inlet condition is imposed 2.1m upstream of the body and an outlet condition is imposed 6m downstream. A no-slip wall condition is imposed on the ground.
floor and car body, with slip walls applied to the wind tunnel walls. The time step is set to $\Delta t U_\infty / L = 5 \times 10^{-4}$ for each simulation which ensures a maximum CFL number of less than one. Each simulation was run for a total of 30 convective transit times ($= T U_\infty / L$); time-averaging began after the initial 10 transit times.

8.3 Results

8.3.1 Mesh resolution

For a complex flow at a high-Reynolds number such as the Ahmed car body, an assessment of the mesh resolution is useful before any discussion of the results. The following analysis relates to the finer mesh of 4.4 million cells, which was found to be most accurate. As discussed in the preceding results sections, one method to assess the quality of the grid is the cell size in wall units, for the wall-adjacent cells in each spatial direction. For a wall-resolved LES the suggested values are: $y^+ < 2$, $\Delta x^+ = 50 - 150$, $\Delta z^+ = 15 - 40$ \[15\] where $x, y, z$ refer to the stream-wise, wall-normal and lateral components respectively. Although these guidelines are not directly applicable for DDES, it is useful to assess their values for this case. Over the slant back of the car body these values are $50 < \Delta x^+ < 200$, $0.1 < \Delta y^+ < 0.5$, $10 < \Delta z^+ < 80$; however both $\Delta x^+$ and $\Delta z^+$ rose to nearly twice this maximum value at $x/H \approx -0.52$ where the flow separates. From these values the near-wall resolution can be judged to be satisfactory for a hybrid RANS-LES method especially in the wall-normal direction but not ideal for both the stream-wise and lateral directions especially at the point of separation where the LES mode of DDES is active.

Another useful measure of the mesh resolution further away from the wall is the ratio $\Delta / \eta$, where $\Delta$ is the filter width and $\eta$ is an estimate of the Kolmogorov length scale $= (\nu^3/\varepsilon)^{1/4}$. The dissipation rate is taken from the modelled dissipation equation, $\varepsilon$ (in a similar fashion to Šarić et al. \[16\]). As reported in Fröhlich et al. \[17\] the ideal value for this ratio for a LES is $\Delta / \eta < 12$, to ensure that a suitable amount of the dissipation range is resolved. Over the slant back of the car body ($-0.52 < x/H < 0$), this ratio reaches as much as $\Delta / \eta = 150$ in the separated shear layer. The relatively coarse nature of this grid means that the refinement observed at the wall is not sustained as the mesh expands away from the wall. Such a high value is not ideal and suggests that the LES mode
will not be fully capable of resolving the flow and additional dissipation from the sub-grid scale model (the underlying RANS model) is expected. A final measure of the suitability of the grid is the ratio of the modelled turbulent viscosity to the molecular viscosity \( \nu_t/\nu \). This gives an indication of the ratio of the modelled and resolved contributions to the dissipation [47]. This ratio reaches up to \( \nu_t/\nu = 150 \) in the initial separation region and reduces to approximately \( \nu_t/\nu = 50 \) further down the back of the body. As expected, the relative coarseness of the grid means at the back of the body the underlying RANS model still has a contribution to the overall turbulence level, and as such is not ideal for the LES mode of each DDES formulation.

With these factors in mind, these guidelines and results are difficult directly to apply to a complex flow and therefore the resolution requirements set out for a wall-resolved LES in simple flows is difficult to achieve in more complex high-Reynolds number flows.

### 8.3.2 DDES results

**DDES results: General flow features**

Figures 8.4, 8.5, 8.6 and 8.7 show the mean streamlines over the rear portion of the car body at the centreline \( y=0 \) for the experimental data, SST-DDES, \( \varphi - f \) DDES and SA DDES models, respectively. Compared to the experimental data in Figure 8.4, both the \( \varphi - f \) DDES and SA DDES models show the flow completely separated over the slant back of the body, which then forms a large separation region behind the main body. A second separation region occurs due to the bottom slant edges, which is present in each of the computations. The \( \varphi - f \) DDES model has a smaller recirculation region than the SA DDES model, which over-predicts the extent of this region compared to the experimental data. The height and thickness of the separation region on the slanted back of the car body for the SA DDES method results in a separation region almost twice as large as that observed with the SST DDES and \( \varphi - f \) DDES models. The flow for both the \( \varphi - f \) DDES and SA DDES models is more similar to that observed experimentally and numerically for the 35\(^\circ\) degree case, where the weaker counter-rotating vortices result in the flow completely separating over the entire slant back surface. The SST DDES model correctly predicts the smaller recirculation region over the slant back, which results in the flow reattaching about three quarters of
the way down the slant back. Whilst this is closer to the experimental data, the recirculation at the back of the body is still larger than the experimental data and even larger than that predicted by the $\varphi - f$ DDES model.

**DDES results: Iso-surfaces**

The iso-surfaces of the Q-criterion for each of the DDES formulations are shown in Figures 8.8 to 8.13. Each DDES formulation shows the same general features, which are the counter-rotating vortices produced by the slant edges on both sides of the car body, and the separated flow region behind the car body. The $\varphi - f$ DDES shows the most resolved content compared to the SST and SA DDES models, however it is the differences in the structure of the counter-rotating vortices that help to explain the differing separation regions. As described earlier, it is the strength of these counter-rotating vortices that help the flow to reattach at $25^\circ$, whilst at $>30^\circ$ they are too weak to bring enough momentum into the separation region to force it to reattach. In Figure 8.13 the SA DDES model shows weaker vortices than both the SST DDES and $\varphi - f$ DDES which may explain why this model shows the largest separation region. Although there are differences in the flow structure between the SST DDES and $\varphi - f$ DDES, the vortices themselves both look relatively strong compared to the SA DDES, so it is not possible to explain the differences between these two models from these iso-surfaces.

**DDES results: Iso-contours**

The differences between the DDES formulations is expressed more clearly through iso-contours (locations shown in Figure 8.14) for the mean velocity velocity at four $yz$ slices at $x/H = 0$, $x/H = 0.27$, $x/H = 0.69$ and $x/H = 1.74$ (Figures 8.15, 8.16, 8.17 and 8.18 respectively). The experimental results clearly show the counter rotating vortices as they dissipate further away from the car body. The re-attached flow on the slant back of the car body is visible in Figure 8.15(a), as well as the main separation region behind the car body in Figure 8.16(a). The SST DDES model largely captures the correct flow features at $x/H = 0$ and $x/H = 0.27$, as shown in Figures 8.15(b) and 8.16(b), the main feature being the reattached flow at the end of the car body. At these locations the $\varphi - f$ DDES model (Figures 8.15(d) and 8.16(d)) incorrectly shows a separated flow at $x/H = 0$, although the flow at $x/H = 0.27$ is in good agreement.
with the experimental data. The contours show a strong vortex core at both of these locations. The SA DDES model however shows a much larger recirculation region at the end of the car in Figure 8.15(c) - this was also seen in the streamlines plots. There is little sign of the counter-rotating vortices that are clearly present for both the SST and $\varphi - f$ DDES models, which confirms what was seen in the iso-surfaces. At $x/H = 0.27$ (Figure 8.16(c)) the flow is also quite different than that of the experimental data as well as the SST and $\varphi - f$ DDES models, which all show a smaller recirculation region than is observed with the SA DDES model.

Further downstream at $x/H = 0.69$ the SST DDES model shows too large a recirculation region (Figure 8.17(b)), as the experimental data shows no sign of recirculation at this point (Figure 8.17(a)). This may be due to the grid resolution being poor in this region, as the onset flow is similar to the experimental data. The final position for the SST DDES model at $x/H = 1.74$ is similar to the experimental data with only a pair of weak counter-rotating vortices present (Figures 8.18(a) and 8.18(b)). The $\varphi - f$ DDES model matches the experimental data at $x/H = 0.69$ in terms of the lack of a recirculation zone, although the positioning of the counter-rotating vortices is different than the experiment and SST DDES results. This is even more clear at $x/H = 1.74$ (Figure 8.18(d)), where the vortices appear to be moving outwards rather than inwards, as the experimental would suggest. The SA DDES model is worse than both the SST DDES and $\varphi - f$ DDES models and shows far too strong a recirculation region at $x/H = 0.69$ (Figure 8.17(c)), and thus at $x/H = 1.74$ (Figure 8.18(c)), the strength of the vortices is stronger than shown in the experimental data.

**DDES results: Mean stream-wise velocity profiles**

The mean stream-wise velocity profiles over the slant back at $y = 0$ shown in Figure 8.19 illustrate more clearly what was observed in the streamlines plots. The SST DDES model matches the experimental data closely, but the SA DDES and to a lesser extent the $\varphi - f$ DDES under-predict the stream-wise velocity and remain separated over the entire surface. Behind the car body (shown in Figure 8.20), the SST DDES still matches the experimental reasonably well. At this position the $\varphi - f$ DDES model also recovers to give good agreement with the experimental results. The SA DDES however still under-predicts the stream-wise velocity because of the overly large recirculation zone.
DDES results: Mean wall-normal velocity profiles

The same theme is observed for the mean wall-normal velocity components shown in Figures 8.21 and 8.22. As the separation region is too strong for the $\varphi - f$ and SA DDES models, this lifts the flow too much, which results in too high a value for the wall-normal velocity. Behind the car body the $\varphi - f$ DDES model matches the experimental results reasonably well but the SA DDES model still over-predicts this velocity component. The SST DDES model under-predicts this component of the velocity initially but then agrees reasonably well with the experimental data once the flow is reattached further down the slanted back. The under-prediction of the initial wall-normal velocity helps the flow to reattach earlier as the recirculation region is thinner.

DDES results: Mean TKE profiles

The relative performance of each DDES formulation can be largely attributed to the amount of turbulent kinetic energy (TKE) in the flow field, as illustrated in Figures 8.23 and 8.24. At the first three profile positions the amount of TKE is in good agreement with the experimental data. As the flow turns over the slanted back of the car ($X = -180\,mm$), the experimental data shows an increase in TKE within the separated shear layer but both the $\varphi - f$ DDES and SA DDES under-predict this. The SST DDES model does show a much larger rise in the TKE value but the shape of the profile suggests too small a boundary layer growth at this point. The peak value is correct but it is confined to a third of the size of the boundary layer predicted by the experimental data. This pattern repeats itself at the next profile position (5) where the SST DDES again has the correct peak value but is is confined to a much smaller region. The level of resolved turbulence in the initial shear layer is important for the models ability to correctly predict separation and reattachment. Having too little resolved turbulence in this slant back region means that less turbulent mixing is present and as such the separation extends further, which is the case for the SA and $\varphi - f$ DDES models. Further down the slanted back of the car body, the $\varphi - f$ DDES shows an increase in TKE, which helps it to reduce the size of the recirculation compared to the SA DDES. The SA DDES model fails to reach the experimental values, which again explains why this model shows the largest separation region. At the back of the car body (Figure 8.24), it is possible to see the second vortex being shed from the lower edges of the car body, which was visible in the streamlines previously.
The $\varphi - f$ DDES models shows the largest TKE value here, which ties in with the model predicting the smallest recirculation region.

**DDES results: Conclusions**

The Ahmed car body illustrates a complex 3D turbulent flow in action. The link between the strength of the counter-rotating vortices and the initial turbulence in the separated shear layer give rise to the differences observed between the DDES formulations. The underlying RANS model here clearly plays an influential role in the prediction of the flow. The SA DDES performs worst in this case by not capturing the strength of the counter-rotating vortices which then leads to massive separation, resulting in a flow field which is similar to that seen at higher slant back angles ($> 30^\circ$). The SST DDES performs the best of the DDES formulations tested, by correctly capturing the strength of the counter-rotating vortices and partly capturing the strength of the initial turbulence after separation (although not perfectly). The $\varphi - f$ DDES model performs somewhat in-between these two formulations, as while it does capture the strength of the counter-vortices, it fails to predict the increase in the turbulence level just after separation which leads to too long a separation region. It recovers better than the SA DDES but ultimately fails because of the incorrect recirculation bubble on the slanted back surface.

Interestingly the results from the coarse grid for both the SST DDES and $\varphi - f$ DDES are very similar (Figures 8.25 and 8.26), yet with the refined grid the SST DDES makes more of an improvement than the $\varphi - f$ DDES model. It is likely therefore that the solution would likely be influenced by further refinement of the grid that might allow the $\varphi - f$ DDES to capture correctly the initial turbulence level and as a result the correct recirculation zone. The SA DDES shows no separation on the coarse mesh but then massive separation on the fine grid, highlighting the sensitivity to the mesh. Further work to evaluate a finer grid in the region would be useful but unfortunately, due to computational resources, this was not possible during this research.
Figure 8.1: Dimensions of the Ahmed car body

Figure 8.2: Coarse mesh for the Ahmed car body

Figure 8.3: Fine mesh for the Ahmed car body
Figure 8.4: Mean streamlines over the Ahmed car body from experimental results

Figure 8.5: Mean streamlines over the Ahmed car body for the SST DDES model
Figure 8.6: Mean streamlines over the Ahmed car body for the $\varphi - f$ DDES model

Figure 8.7: Mean streamlines over the Ahmed car body for the SA DDES model
Figure 8.8: Iso-surfaces of the Q-criterion over the Ahmed car body for the SST DDES model, coloured by mean stream-wise velocity.

Figure 8.9: Iso-surfaces of the Q-criterion over the Ahmed car body for the $\varphi - f$ DDES model, coloured by mean stream-wise velocity.
Figure 8.10: Iso-surfaces of the Q-criterion over the Ahmed car body for the SA DDES model, coloured by mean stream-wise velocity.

Figure 8.11: Iso-surfaces of the Q-criterion over the Ahmed car body for the SST DDES model, coloured by mean stream-wise velocity.
Figure 8.12: Iso-surfaces of the Q-criterion over the Ahmed car body for the $\varphi-f$ DDES model, coloured by mean stream-wise velocity.

Figure 8.13: Iso-surfaces of the Q-criterion over the Ahmed car body for the SA DDES model, coloured by mean stream-wise velocity.
Figure 8.14: Position of iso-contours cut planes [4]

Figure 8.15: Iso-contours of mean velocity at $x/H = 0$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body.
Figure 8.16: Iso-contours of mean velocity at $x/H = 0.27$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body.
Figure 8.17: Iso-contours of mean velocity at $x/H = 0.69$ for (a) experimental data [4], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi-f$ DDES model for the Ahmed car body.
Figure 8.18: Iso-contours of mean velocity at $x/H = 1.74$ for (a) experimental data \[4\], (b) SST-DDES model, (c) SA-DDES model, and (d) $\varphi - f$ DDES model for the Ahmed car body.
Figure 8.19: Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].
Figure 8.20: Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].
Figure 8.21: Mean wall-normal velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].
Figure 8.22: Mean wall-normal velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models. Experimental data from [4].
Figure 8.23: Mean turbulent kinetic energy profiles for the Ahmed car body using the SST DDES, \( \varphi - f \) DDES and SA DDES models. Experimental data from [4].
Figure 8.24: Mean turbulent kinetic energy profiles for the Ahmed car body using the SST DDES, $\phi - f$ DDES and SA DDES models. Experimental data from [4].
Figure 8.25: Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, \( \varphi - f \) DDES and SA DDES models using the coarse 2.7 million cell mesh. Experimental data from [4].
Figure 8.26: Mean stream-wise velocity profiles for the Ahmed car body using the SST DDES, $\varphi - f$ DDES and SA DDES models using the coarse 2.7 million cell mesh. Experimental data from [4].
Chapter 9
Conclusions and suggestions for further work

9.1 Conclusions

This thesis has presented the development, implementation and testing of a new formulation of delayed detached-eddy simulation (DDES) based upon elliptic relaxation. This new model has been extensively compared against the Spalart-Allmaras (SA) and SST based DDES models for a range of test cases that are subject to both geometry and pressure-induced separation.

The new DDES formulation is based upon the $\varphi - f$ RANS model that offers improved near-wall modelling compared to the standard SST and SA RANS models. The conclusions of major hybrid RANS-LES projects (FLOMANIA [49], DESider [6] and ATAAC [56]) is that the underlying RANS model can play an important role for cases which are subjected to complex near wall physics (i.e turbulent separation from a smooth surface), hence the need for a DDES formulation based upon a more advanced RANS model has been addressed.

In the initial stages of the work, the $\varphi - f$ DDES model was implemented into a 3D unstructured finite volume code, Code_Saturne, where it was calibrated and validated using decaying isotropic turbulence. After which the model was carefully compared against the SST-DDES and SA-DDES models for a range of test cases. Although the major conclusions from each test case were presented in their respective chapters a brief summary of the key points is repeated here to bring together common conclusions.
Decaying Isotropic Turbulence

- The $C_{DDES}$ constant for each DDES formulation has been calibrated on a range of meshes to give the correct level of dissipation ($C_{DDES} = 0.60$).

- The $\Psi$ correction function (Equation 4.8) for the $\varphi - f$ DDES model has been derived and implemented in order to give the correct sub-grid scale model behaviour.

- The need for a low-dissipative scheme for the LES mode of DDES has been clearly demonstrated for this case, illustrating the need for the hybrid numerical scheme that was developed in the course of this work (Equation 3.14).

2D periodic hills

- At $Re = 10590$ all three DDES formulations gave a good representation of the turbulence levels, with close agreement with experimental and LES data for the separation region.

- At $Re = 37000$ the agreement with the LES and experimental data was not as close as $Re = 10590$; this is due to the grid at this Reynolds number being relatively coarser than in the lower Reynolds number case.

- For both Reynolds numbers, the sensitivity to the underlying RANS model is weak, which is likely due to the periodic nature of the flow ensuring that there is little grey area (i.e the resolved turbulence develops quickly) and that the mesh is still sufficiently fine, thus limiting the influence of the RANS model to a small near-wall region.

- This case also demonstrated a weak dependancy on the numerical scheme, and only showed minor differences between a second order upwind scheme and CDS scheme.

- The results from each model matched those obtained in separate projects (such as the ATAAC project [56]) using the same mesh, demonstrating that the implementation is correct and the level of dissipation in the CFD code is acceptable.
NACA0021 at 60° degrees incidence

- The $\varphi - f$ DDES model demonstrated closer agreement with the experimental data and previously published results [6] than both the SST-DDES and SA-DDES models.

- As this test case has been a standard reference case to evaluate DDES formulations, the promising performance of the $\varphi - f$ DDES model builds confidence in its implementation and calibration.

2D wall-mounted hump

- This case demonstrates that even for a flow whose separation is largely geometry induced, the modelling of the flow upstream of the separation point (which is modelled in RANS mode) can have a significant effect on the size and strength of the recirculation zone.

- The SA-DDES provides the best agreement for the mean skin-friction coefficient but appears to do this for the wrong reasons, with some element of error-cancelation giving rise to good agreement with the experimental values. The $\varphi - f$ DDES provides a better representation of the mean flow and turbulent shear stress than the SST-DDES model.

- This case demonstrates a strong sensitivity to the numerical scheme, as the rate at which the resolved turbulence in the initial shear layer grows is heavily influenced by the amount of numerical dissipation. The use of a second order upwind scheme (for any of the DDES formulations) gives a much delayed transition to resolved turbulence and a delayed separation region. It is essential to use a low-dissipation numerical scheme in this initial separated shear layer region.

Ahmed Car body

- The link between the strength of the counter-rotating vortices and the initial turbulence in the separated shear layer gives rise to the differences observed between the DDES formulations. The underlying RANS model here clearly plays an influential role in the prediction of the flow.
• The $\varphi - f$ DDES and SST-DDES capture the flow much better than the SA-DDES, which over-predicts the separation region.

• The SST-DDES model provides closer agreement with the experimental values than the $\varphi - f$ model, due to a better representation of the turbulence in initial separation region.

• The mesh resolution is still coarse for this high Reynolds number and it is expected that further grid refinement would have a strong influence on the results.

Final remarks

From the four main test cases studied within this thesis, a general sensitivity to the underlying RANS model has been observed (apart from the 2D periodic hills). For each test case a different DDES formulation has provided the most accurate representation of the flow (although some issues of error cancellation may be present). This agrees with the assertion in the introduction of this thesis that the underlying RANS model has a strong influence on the results of a DDES simulation, especially for flows subject to complex 3D turbulent separation.

Whilst the $\varphi - f$ RANS model does involve two further transport equations compared with the SST model, in all cases, the increased computational cost of the $\varphi - f$ DDES model relative to SST-DDES model was observed to be low; less than 10%. Moreover, no numerical difficulties were encountered with the $\varphi - f$ DDES model and as such there was no need for smaller timesteps or additional numerical dissipation.

Thus, while the $\varphi - f$ DDES model is not a fix for the shortcomings of DDES, it is a practical and robust alternative to the established SST-DDES and SA-DDES variants that have become the de facto choice for many DDES users.

9.2 Suggestions for Future work

This work has demonstrated the need for more focus on the underlying RANS models that make up hybrid RANS-LES approaches. Thus the recommendations for future work are split between; a need for more advanced RANS models to be evaluated within a hybrid RANS-LES approach such as DDES, and; also the need to combine these with alternative hybrid RANS-LES approaches.
Alternative length scale substitution

- In the current work, the turbulent length scale was only replaced in the turbulent kinetic energy equation to produce a formulation that was consistent with the current SST and SA DDES approaches. However it would be informative to undertake a study to evaluate the effect of replacing the turbulent length scale in the other equations of the $\varphi - f$ model, as well as the turbulent viscosity relationship as in [88].

Development of a $\varphi - f$ based zonal hybrid RANS-LES model

- For industrial applications, the use of a zonal hybrid RANS-LES method such as ZDES is attractive because of its ability to prescribe explicitly the RANS and LES zones, which can be particularly useful for complex geometries. ZDES has predominately been based upon the SA RANS model, but given the improved near-wall modelling of the $\varphi - f$ model and the performance it has shown during this work, it would be interesting to see its performance within a ZDES formulation.

Calibration of the DDES blending function for the $\varphi - f$ DDES model

- Here the $\varphi - f$ DDES model has been based upon the same DDES blending function used in the SST and SA-DDES models, i.e no tuning of the $f_d$ function was undertaken. This was decided as most other DDES formulations such as those in Mockett et al. [57] were used with no modification of the $f_d$ constants. It was also decided to focus on the performance of the $\varphi - f$ model and not to skew the results with any further calibration of model constants. However, modifying the blending function for the SST DDES model has recently shown improvements [58] where the transition from RANS to LES mode was reduced, ensuring the grey area was shortened. It would therefore be informative to see whether a re-calibration of these constants would improve the performance of the $\varphi - f$ DDES model.

Development of a non-linear $\varphi - f$ DDES model

- Although the $\varphi - f$ RANS model has been shown to be a good choice for a DDES because of its improved near-wall modelling, further improvements may be possible by using a non-linear version of this model to capture the
anisotropy in complex near-wall flows such as the model developed in Reif et al. [29]. However the question of whether such a model would impair the stability and improve accuracy would need to be addressed before any further development could take place.
Bibliography


[110] D. Greenblatt, K. B. Paschal, C. S. Yao, J. Harris, N. W. Schaeffler, and


