An industrial approach to near-wall turbulence modelling for unstructured finite volume methods.

A thesis submitted to The University of Manchester for the degree of Doctor of Philosophy in the faculty of Engineering and Physical Sciences

September 2006

Juan C. Uribe

School of Mechanical, Aerospace and Civil Engineering
Contents

List of Figures .......................................................................................................................... 9
List of Tables ............................................................................................................................. 11
Abstract .................................................................................................................................. 13
Declaration ................................................................................................................................. 15
Copyright .................................................................................................................................. 17
Acknowledgements ................................................................................................................... 19
Nomenclature ............................................................................................................................. 21

1 Introduction ............................................................................................................................ 27
   1.1 Study Objectives .............................................................................................................. 29
   1.2 Outline of the thesis ........................................................................................................ 31

2 Theoretical background .......................................................................................................... 33
   2.1 Turbulence modelling ..................................................................................................... 33
   2.2 Reynolds Averaged Navier-Stokes equations ................................................................ 34
       2.2.1 Reynolds stress modelling ...................................................................................... 35
       2.2.2 Eddy viscosity models ............................................................................................ 39
   2.3 Large Eddy Simulation ................................................................................................... 46
   2.4 Hybrid models ................................................................................................................. 48
   2.5 Wall treatment ................................................................................................................. 51
<table>
<thead>
<tr>
<th>2.5.1 Standard wall function</th>
<th>51</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5.2 Scalable wall function</td>
<td>53</td>
</tr>
</tbody>
</table>

### 3 Wall modelling

<table>
<thead>
<tr>
<th>3.1 Wall effects</th>
<th>55</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.2 Damping functions</td>
<td>57</td>
</tr>
<tr>
<td>3.3 Two component limit</td>
<td>60</td>
</tr>
<tr>
<td>3.4 Elliptic relaxation</td>
<td>61</td>
</tr>
</tbody>
</table>

### 4 The $\varphi - f$ model

<table>
<thead>
<tr>
<th>4.1 Introduction</th>
<th>75</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.2 Model derivation</td>
<td>76</td>
</tr>
<tr>
<td>4.3 Boundary conditions</td>
<td>76</td>
</tr>
<tr>
<td>4.4 Closure constants</td>
<td>78</td>
</tr>
<tr>
<td>4.5 Budgets</td>
<td>78</td>
</tr>
<tr>
<td>4.6 The $\zeta - f$ model</td>
<td>78</td>
</tr>
</tbody>
</table>

### 5 Numerical method

<table>
<thead>
<tr>
<th>5.1 Introduction</th>
<th>81</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2 The finite volume method</td>
<td>82</td>
</tr>
<tr>
<td>5.3 Time discretisation</td>
<td>84</td>
</tr>
<tr>
<td>5.4 Boundary conditions</td>
<td>85</td>
</tr>
<tr>
<td>5.4.1 Inlet</td>
<td>86</td>
</tr>
<tr>
<td>5.4.2 Outlet</td>
<td>86</td>
</tr>
<tr>
<td>5.4.3 Walls and symmetries</td>
<td>86</td>
</tr>
<tr>
<td>5.5 Notes on the implementation of new models</td>
<td>88</td>
</tr>
</tbody>
</table>

### 6 Channel flow

<table>
<thead>
<tr>
<th>6.1 Introduction</th>
<th>93</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2 Numerical framework</td>
<td>95</td>
</tr>
<tr>
<td>6.2.1 Eddy viscosity models</td>
<td>95</td>
</tr>
<tr>
<td>6.2.2 Second moment closure</td>
<td>96</td>
</tr>
<tr>
<td>Chapter</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>6.3</td>
<td>Results</td>
</tr>
<tr>
<td>7</td>
<td>Turbulent heated cavity</td>
</tr>
<tr>
<td>8</td>
<td>Asymmetric plane diffuser</td>
</tr>
<tr>
<td>9</td>
<td>Flow over periodic hills</td>
</tr>
<tr>
<td>10</td>
<td>Hybrid modelling using the $\varphi - f$ model</td>
</tr>
<tr>
<td>11</td>
<td>Conclusions and recommendations for future work</td>
</tr>
</tbody>
</table>
Bibliography 172
# List of Figures

2.1 Sketch of the domain separation on RANS-LES zonal methods ........................................ 50

3.1 Wall effects .................................................................................................................. 57

4.1 Budgets of $\varphi$ in a channel flow ........................................................................... 79

4.2 Budgets of $f$ in a channel flow .................................................................................. 79

5.1 Notations for the spatial discretisation .............................................................. 83

5.2 Representation of the first near-wall cell .............................................................. 90

6.1 Plane channel flow .................................................................................................. 93

6.2 Plane channel flow $Re_\tau = 395$, Velocity profile ........................................... 95

6.3 Channel flow $Re_\tau = 395$, SST, $k - \omega$ and $k - \varepsilon$ Lauder-Sharma .......... 97

6.4 Channel flow $Re_\tau = 395$, $\omega^2 - f$, LDM and $\varphi - f$ models ................. 98

6.5 Comparison of the neglected terms in logarithmic scale (absolute values) .......... 100

6.6 Terms in the $f$ equation ......................................................................................... 100

6.7 Profile of $f$ in a Channel flow $Re_\tau = 640$ .......................................................... 101

6.8 Channel flow $Re_\tau = 640$ ..................................................................................... 102

6.9 Channel flow, elliptic relaxation models $Re_\tau = 640$ ........................................... 103

6.10 Mesh refinement in a channel flow, Standard wall function, SSG Model ........ 104

6.11 Mesh refinement in a channel flow, Scalable wall function, SSG Model ........... 105

7.1 Turbulent heated cavity ............................................................................................. 109

7.2 Velocity profiles at $Ra = 0.86 \times 10^6$, $\omega$ based models .................................. 112

7.3 Temperature profiles at $Ra = 0.86 \times 10^6$, $\omega$ based models ....................... 113
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.4</td>
<td>Velocity profiles at $Ra = 0.86 \times 10^6$, $\varepsilon$ based models.</td>
<td>114</td>
</tr>
<tr>
<td>7.5</td>
<td>Temperature profiles at $Ra = 0.86 \times 10^6$, $\varepsilon$ based models.</td>
<td>115</td>
</tr>
<tr>
<td>7.6</td>
<td>Velocity profiles at $Ra = 1.43 \times 10^6$, $\omega$ based models</td>
<td>116</td>
</tr>
<tr>
<td>7.7</td>
<td>Temperature profiles at $Ra = 1.43 \times 10^6$, $\omega$ based models</td>
<td>117</td>
</tr>
<tr>
<td>7.8</td>
<td>Velocity profiles at $Ra = 1.43 \times 10^6$, $\varepsilon$ based models.</td>
<td>118</td>
</tr>
<tr>
<td>7.9</td>
<td>Temperature profiles at $Ra = 1.43 \times 10^6$, $\varepsilon$ based models.</td>
<td>119</td>
</tr>
<tr>
<td>8.1</td>
<td>Asymmetric plane diffuser. Geometry description</td>
<td>121</td>
</tr>
<tr>
<td>8.2</td>
<td>Asymmetric plane diffuser. Friction coefficient along the inclined wall</td>
<td>124</td>
</tr>
<tr>
<td>8.3</td>
<td>Asymmetric plane diffuser. Streamlines for the SSG model</td>
<td>125</td>
</tr>
<tr>
<td>8.4</td>
<td>Asymmetric plane diffuser. Pressure coefficient along the inclined wall</td>
<td>125</td>
</tr>
<tr>
<td>8.5</td>
<td>Asymmetric plane diffuser. Velocity profiles, first half, $\varepsilon$ based models</td>
<td>127</td>
</tr>
<tr>
<td>8.6</td>
<td>Asymmetric plane diffuser. Velocity profiles, second half, $\varepsilon$ based models</td>
<td>128</td>
</tr>
<tr>
<td>8.7</td>
<td>Asymmetric plane diffuser. Contours of $f$ for the $\varphi - f$ model</td>
<td>129</td>
</tr>
<tr>
<td>8.8</td>
<td>Asymmetric plane diffuser. Contours of $\varphi$ for the $\varphi - f$ model</td>
<td>129</td>
</tr>
<tr>
<td>8.9</td>
<td>Asymmetric plane diffuser. Velocity profiles, first half, $\omega$ based models</td>
<td>130</td>
</tr>
<tr>
<td>8.10</td>
<td>Asymmetric plane diffuser. Velocity profiles, second half, $\omega$ based models</td>
<td>131</td>
</tr>
<tr>
<td>8.11</td>
<td>Asymmetric plane diffuser. Contours of blending function $f_1$ for the SST model</td>
<td>132</td>
</tr>
<tr>
<td>8.12</td>
<td>Asymmetric plane diffuser. Contours of viscosity limiter for the SST model</td>
<td>132</td>
</tr>
<tr>
<td>8.13</td>
<td>Asymmetric plane diffuser. Velocity profiles, first half, SSG model</td>
<td>133</td>
</tr>
<tr>
<td>8.14</td>
<td>Asymmetric plane diffuser. Velocity profiles, second half, SSG model</td>
<td>134</td>
</tr>
<tr>
<td>9.1</td>
<td>Flow over periodic hills. Computational domain</td>
<td>135</td>
</tr>
<tr>
<td>9.2</td>
<td>Streamlines</td>
<td>139</td>
</tr>
<tr>
<td>9.3</td>
<td>Flow over periodic hills. Velocity profiles for the elliptic relaxation models</td>
<td>141</td>
</tr>
<tr>
<td>9.4</td>
<td>Flow over periodic hills. Velocity profiles. SST, $k - \omega$ and Launder-Sharma models</td>
<td>142</td>
</tr>
<tr>
<td>9.5</td>
<td>Flow over periodic hills. Velocity profiles. SSG model</td>
<td>143</td>
</tr>
<tr>
<td>9.6</td>
<td>Flow over periodic hills. Shear stress profiles, elliptic relaxation models</td>
<td>144</td>
</tr>
<tr>
<td>9.7</td>
<td>Flow over periodic hills. Shear stress profiles. SST, $k - \omega$ and Launder-Sharma models</td>
<td>145</td>
</tr>
</tbody>
</table>
9.8 Flow over periodic hills. Shear stress profiles, SSG model . . . . . . . . . . . . 146
9.9 Flow over periodic hills. Kinetic energy profiles, elliptic relaxation models . . 147
9.10 Flow over periodic hills. Kinetic energy profiles, SST, $k - \omega$ and Launder-Sharma models. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 148
9.11 Flow over periodic hills. Kinetic energy profiles, SSG model. . . . . . . . . . 149

10.1 Length scales in a channel flow at $Re_\tau = 395$ . . . . . . . . . . . . . . . . . 154
10.2 Velocity profiles with different blending functions . . . . . . . . . . . . . . . . 155
10.3 Blending functions with different coefficients . . . . . . . . . . . . . . . . . . 156
10.4 Velocity history in the middle of the channel . . . . . . . . . . . . . . . . . . 156
10.5 Velocity profile for case C1. $Re_\tau = 395$ . . . . . . . . . . . . . . . . . . . . . 159
10.6 Shear stress for case C1. $Re_\tau = 395$ . . . . . . . . . . . . . . . . . . . . . . 159
10.7 Normal stresses for case C1. Solid line: model, Dotted line: LES . . . . . . . . 160
10.8 $R_{11}$ streamwise correlation. $y^+ = 5$, case C1. . . . . . . . . . . . . . . . . . 160
10.9 $R_{11}$ spanwise correlation. $y^+ = 5$, case C1 . . . . . . . . . . . . . . . . . . 160
10.10 Contours of $u'$ at $y^+ = 5$. Standard LES, case C1 . . . . . . . . . . . . . . 161
10.11 Contours of $u'$ at $y^+ = 5$. Hybrid model, case C1 . . . . . . . . . . . . . . 161
10.12 Blending function, $f$ for cases C1, C2 and C3 . . . . . . . . . . . . . . . . . 161
10.13 Velocity profiles. No symbol C1, + C2, * C2, o DNS . . . . . . . . . . . . . . 162
10.14 Shear stress. No symbol C1, + C2, * C2, o DNS . . . . . . . . . . . . . . . . 162
10.15 $R_{11}$ streamwise correlation. $y^+ = 5$, cases C1 and C3 . . . . . . . . . . . . . 163
10.16 $R_{11}$ spanwise correlation. $y^+ = 5$, cases C1 and C3 . . . . . . . . . . . . . 163
10.17 Normal stresses for cases C1 and C3 . . . . . . . . . . . . . . . . . . . . . . . 163
10.18 Velocity profiles for cases C1, C4, C6, C8, and C10 . . . . . . . . . . . . . . 164
10.19 Normal stresses for case C8 . . . . . . . . . . . . . . . . . . . . . . . . . . . 164
10.20 Shear stress for case C8 . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 164
10.21 Instantaneous streamwise velocity contours at $y^+ = 1$ for case C8 . . . . . 165
10.22 Instantaneous streamwise velocity contours at $y^+ = 1$ for case C10 . . . . . 165
# List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Coefficients of the LRR model</td>
<td>38</td>
</tr>
<tr>
<td>2.2</td>
<td>Coefficients of the SSG model</td>
<td>39</td>
</tr>
<tr>
<td>2.3</td>
<td>Coefficients of the standard $k - \varepsilon$ model</td>
<td>42</td>
</tr>
<tr>
<td>2.4</td>
<td>Coefficients of the standard $k - \omega$ model</td>
<td>43</td>
</tr>
<tr>
<td>2.5</td>
<td>Coefficients of the SST model</td>
<td>45</td>
</tr>
<tr>
<td>3.1</td>
<td>Coefficients of the SSG-Chen Low Reynolds model</td>
<td>58</td>
</tr>
<tr>
<td>3.2</td>
<td>Coefficients of the SSG-Chen Low Reynolds model for the dissipation rate</td>
<td>59</td>
</tr>
<tr>
<td>3.3</td>
<td>Coefficients of the Elliptic Relaxation model</td>
<td>63</td>
</tr>
<tr>
<td>3.4</td>
<td>Coefficients of the $\tau^2 - f$ original model</td>
<td>67</td>
</tr>
<tr>
<td>3.5</td>
<td>Coefficients of the LDM</td>
<td>71</td>
</tr>
<tr>
<td>4.1</td>
<td>Coefficients of the $\varphi$ model</td>
<td>78</td>
</tr>
<tr>
<td>9.1</td>
<td>Separation and reattachment points for various models</td>
<td>140</td>
</tr>
<tr>
<td>10.1</td>
<td>Summary of cases</td>
<td>158</td>
</tr>
</tbody>
</table>
Abstract

This thesis deals with the wall effects on turbulent flows from an industrial point of view using an unstructured finite volume solver. The main objective being to take into account the wall effects without a large increase in computational requirements. Three different methods of computing wall bounded flows are used; wall functions, low Reynolds number models and a hybrid RANS-LES model.

The scalable wall function approach is studied in two dimensional flows with separation and, although less sensitive to mesh refinement, it is found to be limited even when used with a second moment closure. Two-equation model that can be integrated inside the viscous sublayer are studied, with emphasis on the elliptic relaxation method. A new model, called $\varphi - f$, is derived which has similar properties to the original $\nu^2 - f$ but can be implemented in segregated solvers without loss of performance. The model is based on solving a transport equation for the ratio $\varphi = \nu^2 / k$. The model is tested in channel flows, a heated cavity, an asymmetric plane diffuser and series of periodic hills. Other eddy viscosity models are also tested with these cases. The new model yields good prediction of mean quantities in all cases and globally performs better than the other models studied herein.

A new formulation for a hybrid approach is developed using the $\varphi - f$ for the RANS contribution which incorporates the near wall effects into the LES field. The averaged velocity field is used to compute the RANS equations and in turn the averaged turbulent viscosity. The residual stress tensor is split into two contributions, one isotropic computed with the LES subgrid viscosity and fluctuating strain, and one inhomogeneous computed with the RANS viscosity and averaged strain. The two contributions are joined together via a blending function parametrised by the turbulent length scale and the filter width. The approach is tested in channel flow at different Reynolds numbers up to $Re_\tau = 4000$, with very coarse meshes. It is found that the method reproduces well mean velocity and stresses at all Reynolds numbers.
Declaration

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

Copyright in text of this thesis rests with the author. Copies (by any process) either in full, or of extracts, may be made only in accordance with instructions given by the author and lodged in the John Rylands University Library of Manchester. Details may be obtained from the Librarian. This page must form part of any such copies made. Further copies (by any process) of copies made in accordance with such instructions may not be made without the permission (in writing) of the author.

The ownership of any intellectual property rights which may be described in this thesis is vested in The University of Manchester, subject to any prior agreement to the contrary, and may not be made available for use by third parties without the written permission of the University, which will prescribe the terms and conditions for such agreements.

Further information on the conditions under which disclosures and exploitation may take place is available from the Head of School of Mechanical, Aerospace and Civil Engineering.
Acknowledgements

I would like to thank my supervisor, professor Dominique Laurence for all his help during the course of this work. His continuous advice and patience have been fundamental for the my development as a researcher. His encouragement and his support have been a constant source of motivation during all these years. I would also like to thank professor Sergei Utyuzhnikov, whose help with the development of this thesis is greatly appreciated. I’m grateful to Dr. Rob Prosser who has been very helpful and critical in reviewing the manuscripts, which has certainly improved their quality. Finacial support from the FLOMANIA project is gratefully acknowledge[1].

My thanks goes to my office mates, Dr. Alistair Revell and Dr. Charles Moulinec for the great atmosphere we shared. Without their help and friendship I would certainly not have achieved this much. They have been excellent company and I am very pleased I was able to share these years with them in the lab.

My thanks goes to the EDF team, Valerie Guimet, Sofiane Benhamadouche, Frederich Archambeau and Ivan Fournier. Their patience helping me to understand the numerical code is greatly appreciated.

I thank all my friends in Manchester, Eduardo, Juan Carlos, Bea, Amaya, Adri, Nicolas, Birin and all the others that made my stay here so enjoyable.

I sincerely want to thank my girlfriend Anna, whose help, patience and understanding

1This work has been supported by the FLOMANIA project (Flow Physics Modelling - An Integrated Approach) is a collaboration between Alenia, AEA, Bombardier, Dassault, EADS-CASA, EADS-Military Aircraft, EDF, NUMECA, DLR, FOI, IMFT, ONERA, Chalmers University, Imperial College, TU Berlin, UMIST and St. Petersburg State Technical University. The project is funded by the European Union and administrated by the CEC, Research Directorate-General, Growth Programme, under Contract No. G4RD-CT2001-00613.
during the past years has been the driving force behind all my efforts. She has been the wind at my back.

Last but by no means least, I would like to thank my parents. I am what I am because of them. They have thought me that it is possible to achieve anything, as long as you are willing to put all the effort required. Their love and support has been there always, constant, never fading.
Nomenclature

Greek letters

\( \beta \)  Thermal expansion coefficient
\( \Delta \)  Filter width
\( \delta_{ij} \)  Kronecker delta
\( \Gamma \)  Diffusion coefficient
\( \kappa \)  von Karman constant.
\( \lambda \)  Integral length scale
\( \mu \)  Molecular viscosity
\( \nu \)  Dynamic viscosity
\( \nu_r \)  Sub grid eddy viscosity
\( \nu_t \)  Turbulent viscosity
\( \omega \)  Dissipation rate per unit of kinetic energy
\( \Omega_{ij} \)  Rotation rate tensor
\( \phi_{ij} \)  Pressure strain-rate of Reynolds Stresses
\( \rho \)  Density
\( \tau_{ij} \)  Sub grid tensor
\( \tau_{ij}^R \)  Viscous stress
\( \varepsilon \)  Isotropic dissipation
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varphi$</td>
<td>Normalised wall normal Reynolds stress ($\overline{\nu^2}/k$)</td>
</tr>
<tr>
<td>$\tilde{\varepsilon}$</td>
<td>Modified dissipation rate</td>
</tr>
<tr>
<td>$\phi_{ij}$</td>
<td>Redistribution tensor</td>
</tr>
<tr>
<td>$\varepsilon_{ij}$</td>
<td>Turbulent dissipation rate tensor</td>
</tr>
<tr>
<td><strong>Latin letters</strong></td>
<td></td>
</tr>
<tr>
<td>$n$</td>
<td>Unit vector representing the wall-normal direction</td>
</tr>
<tr>
<td>$A$</td>
<td>Lumley’s flatness parameter</td>
</tr>
<tr>
<td>$A_+$</td>
<td>Van Driest damping coefficient.</td>
</tr>
<tr>
<td>$a_{ij}$</td>
<td>Anisotropy tensor</td>
</tr>
<tr>
<td>$b_{ij}$</td>
<td>Normalised anisotropy tensor</td>
</tr>
<tr>
<td>$D_{ij}^\nu$</td>
<td>Viscous diffusion of Reynolds Stresses</td>
</tr>
<tr>
<td>$D_{ij}^T$</td>
<td>Turbulent diffusion of Reynolds Stresses</td>
</tr>
<tr>
<td>$F_1$</td>
<td>First blending function for the SST model</td>
</tr>
<tr>
<td>$F_2$</td>
<td>Second blending function for the SST model</td>
</tr>
<tr>
<td>$f_{ij}$</td>
<td>Normalised redistribution tensor</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravity</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$L$</td>
<td>Turbulent length scale</td>
</tr>
<tr>
<td>$l$</td>
<td>Mixing length scale</td>
</tr>
<tr>
<td>$P$</td>
<td>Production of $k$</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$p'$</td>
<td>Pressure fluctuation</td>
</tr>
<tr>
<td>$P_k$</td>
<td>Production of turbulent kinetic energy</td>
</tr>
</tbody>
</table>
### Nomenclature

- $P_{ij}$: Production of Reynolds Stresses
- $Pr$: Prandtl number
- $Re$: Reynolds number
- $S$: Strain rate invariant
- $S_{ij}$: Strain tensor
- $T$: Turbulent time scale
- $u'$: Fluctuating velocity, $i^{th}$ component
- $u_i$: Instantaneous velocity, $i^{th}$ component
- $u_k$: Friction velocity based on $k$
- $u_\tau$: Friction velocity
- $x_i$: Coordinate, $i^{th}$ component
- $y^+$: Nondimensional wall distance
- $\langle U_i \rangle$: Average velocity, $i^{th}$ component
- $\overline{U_j}$: Filtered velocity, $i^{th}$ component

### Acronyms

- **BSL**: Baseline Model
- **CDS**: Central Differencing Scheme
- **CFD**: Computational Fluid Dynamics
- **DES**: Detached Eddy Simulation
- **DNS**: Direct Numerical Simulation
- **EVM**: Eddy Viscosity Model
- **GGDH**: Generalised Gradient Diffusion Hypothesis
- **LDM**: Lien and Durbin Model
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LRR</td>
<td>Launder, Reece and Rodi model</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>SCWF</td>
<td>Scalable Wall Function</td>
</tr>
<tr>
<td>SMC</td>
<td>Second Moment Closure</td>
</tr>
<tr>
<td>SSG</td>
<td>Speziale, Sarkar and Gatski model</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>UDS</td>
<td>Upwind Differencing Scheme</td>
</tr>
</tbody>
</table>
A mi abuelo, mis padres y mis hermanos.
Chapter 1

Introduction

Computational Fluid Dynamics (CFD) has been growing as an important tool in fluid prediction and control over the past 40 years due to its relatively cheap cost compared to experimental work. The accurate prediction of flows is important in many areas of engineering such as aerodynamics, atmospheric science and heat transfer. In industry, a good design can provide advantages over competitors, and ultimately increase in revenues. Therefore all of the tools that can make the design process faster and cheaper are always interesting for companies. Although CFD plays an important role in the design process, it is still far from being the only and absolute tool to predict the flow characteristics of different configurations; the main problem is the lack of reliability in turbulent flow predictions. Due to the nature of turbulent flow, unsteadiness and randomness have an important effect in the flow physics and their mathematical representation becomes more complicated and difficult to solve. Almost all industrial problems need to be tackled by approximating the equations and solving them in an iterative manner. This introduces errors since there will always be terms that need to be neglected when approximating a continuous equation by a discretised domain. Even if the discretisation error is small (by having a large number of grid points or a higher order approximation) there are still other issues to take into account. Turbulence has a random nature in which chaotic motion is present. This poses a mathematical problem since the molecules do not move with a constant velocity relative to each other. They move around colliding with each other and transferring momentum and then creating viscous stresses. In a turbulent flow, where the Reynolds number is high enough, the random fluctuations increase the dissipation
of energy. From a mathematical point of view, the fact that these fluctuations are changing in time and space makes it necessary to have a very small distance between the discretised points which in turn makes the computation highly expensive in terms of CPU and memory. Therefore, to solve complex industrial flows, physical approximations have to be introduced to account for all the activity in very small regions. This is where turbulence modelling comes into play.

Modern CFD users are required to have a wide knowledge of issues that range from the numerical discretisation procedure to the physics of fluid flows. Turbulence has an important role in industrial flows, whether it is beneficial (to enhance mixing or heat transfer) or detrimental (drag in vehicles etc). Therefore, its correct modelling is essential to the design process. These reasons explain why much effort has been made by the scientific community to understand and accurately model turbulence.

There are many aspects of turbulence that are desirable to control, but all of these require a great amount of effort to understand and model. In a simple flow it might only be required to be able to predict mean quantities such as velocity and pressure to obtain values for lift and drag. However, there are some other aspects of industrial flow that can be important, such as thermal fluctuations or separation points. This wide range of important parameters in different types of industries has led to the development of turbulence models into a series of branches that can predict better certain desired features of the flow. Therefore is not possible to have an 'universal' model that will be able to cope with all turbulent characteristics within a reasonable CPU time.

When applying CFD to the design process, the accurate representation of flow physics is a key aspect. However, the scientific comunity has developed models that although provide improved physical description, can be complicated and often highly resource demanding. Therefore, industry tends to use code friendly models that provide higher accuracy while keeping the CFD process within a reasonable time frame. The FLOMANIA (FLOW-physics Modelling - AN Integrated Approach) project was proposed as a close collaboration between industries and research institutes and universities across Europe to overcome exiting problems in CFD, particularly in turbulence modelling. Improvement of current industrially used turbulence models striving for more robust and reliable models is one of the main topics of
the project. This helps to bridge the gap between the most advanced turbulence models and its applications in daily industrial practice. The work presented here has been carried out within this project and its industrial relevance has been kept in mind throughout the present research.

1.1 Study Objectives

One of the interesting aspects of the study of turbulence is the near wall layer. In wall bounded flows, there is a thin layer near the wall where the viscous effects become important and introduce anisotropy to the flow. In many flow configurations these effects can play an important role and it becomes necessary to be able to represent them accurately.

The wall effects can be responsible for important criteria such as separation, heat transfer and friction. Although the near wall layer can be very thin, it is here where many important aspects of turbulence take part. At the wall all the velocity components go to zero creating large velocity gradients in the vicinity of the wall. This in turn causes the production of turbulent kinetic energy to peak in this region. These are important aspects that need to be taken into account when modelling the turbulence effects. In industrial applications for many years, the near wall layer has been bridged by using the ‘wall function’ approach, which saves computational time. However, this approach is far from universal since the physics represented in the derivation of such functions are not general. Therefore, when the near wall effects are important, a model capable of resolving the turbulent variables in this region is required. The cost of computing the near wall layer is substantially higher than the use of wall functions. When the wall function fails, it is desirable to have a model that has many of the characteristics of the turbulence models used in the outer part of the flow. In industry, the simple two equation models have been used for many years due to the fact that they are robust and cheap. Models with higher complexity have found little attention from CFD vendors and users since they usually imply stiffness and encounter difficulties when applied to unstructured mesh codes. Unstructured mesh codes have become the main type of codes used in industry since they allow meshes that are easily created and refined while maintaining a reasonable number of cells. Therefore, it is desired that a model for the near wall region
to be easily implemented in an unstructured code and that it is robust, allowing the user to perform calculations in a wide range of geometries and flow conditions.

In the present work, an industrial unstructured code from Electricité de France (EDF) namely *Code Saturne* has been used to highlight the benefits and drawbacks of relatively new models that can compute the viscous sublayer. The models chosen have been implemented by the author. A new model has been derived based on the concept of elliptic relaxation and as a result of the tests presented here, it has been incorporated in the future version of the code. In summary the objective of the thesis is to bridge the gap between; an industrial code, developed with the traditional two equation models and wall function approach and; models with higher complexity that can reproduce the near wall layer correctly while remaining user friendly and not too expensive in terms of CPU usage.

There are three ways presented in this work of approaching the near wall layer, the reason behind all of them being the need for more accurate representation of the wall effects while keeping the computational cost relatively low. The first approach considered has been the wall function approach. During the course of this project a higher accuracy Reynolds stress model was implemented in the code with a recently developed type of wall function that, although it does not introduce more physics into the problem, allows the user to create meshes without a lower limit on the size of the first cell near the wall. The second approach was the actual physical model of turbulence in the near wall layer. A new model was developed in order to make an existing model applicable to industrial cases and unstructured codes while retaining the near wall physics. While developing this model, a compromise had to be made between accuracy and complexity. The industrial requirements have been kept in mind during the whole modelling process. The outcome of the research in this area has led to a derivation of the new $\varphi - f$ model. This model was implemented along with another set of existing models used in industry to resolve the near wall region, and compare predictions in different test cases. The last approach of this work deals with the instantaneous behaviour of the near wall layer. In more complex flows, where the usual statistically averaged models fail, the simulation of the instantaneous turbulent fields can be very helpful. The Large Eddy Simulation (LES) technique is studied from the industrial point of view, where the constraints introduced by the wall make it difficult for use in complex cases at higher Reynolds numbers.
The statistical model developed in the second part of the thesis has been combined with LES in order to be able to ease the near wall three dimensional mesh refinement requirements.

1.2 Outline of the thesis

The work presented here is organised as follows. After this introduction, Chapter 2 gives a brief overview of the existing turbulence models used during the course of this project. In Chapter 3 a brief review of the wall modelling strategies is given, with emphasis on the elliptic relaxation approach. In Chapter 4 the new model, named $\phi - f$, is presented. The numerical aspects of the code used are presented in Chapter 5 with an explanation of the discretisation methods used and notes on the details of the model implementation issues encountered during this work. Chapter 6 presents the basic flow configuration used to test the models; a channel flow between two infinite plates. This case is relatively simple to compute and provides useful information about turbulence, especially about the near wall behaviour. Another relatively simple case is considered in Chapter 7. Here a tall cavity is studied in which the turbulent flow is driven by buoyant forces originating from the temperature difference between the side walls. This case has also been studied extensively, experimentally and numerically, and can be considered as a validation for the new model. It is relevant because the wall effects present play an important role in turbulence generation making it difficult to predict the correct behaviour with the use of wall functions.

In Chapter 8 a two dimensional asymmetric diffuser is studied. This is a relevant case due to the separation that occurs, which is not dictated by the geometry, but by the adverse pressure gradient resulting from the low angle of the diffuser. This makes it an interesting flow for testing the ability of the models to predict separation occurring at the inclined wall. In this case the wall function approach combined with a Reynolds Stress Model has also been studied. Chapter 9 presents the flow over a series of periodic hills where there is also a smooth wall separation, but with a large recirculation region. This flow has proven to be difficult to predict for simple models due to the variety of effects that are encountered.

After the cases where the eddy viscosity models are studied and compared, the new hybrid approach is presented in Chapter 10. The hybrid approach mixes the LES technique (in which
the instantaneous velocities are resolved) with the statistical approach developed in Chapter 4. After the description of the approach, its performance is presented for the channel flow. The channel flow serves as a validation of the hybrid rationale and a series of different mesh configurations are presented.

Finally the conclusions and recommendations for future work are presented in Chapter 11.
Chapter 2

Theoretical background

The majority of the flows in real industrial applications are turbulent, which means their velocity field varies significantly and irregularly both in position and time [96]. The effects of turbulence can be important in a variety of cases. A correct approach to model these effects can lead to a reduction of time and money in industrial design. In this chapter, a short review of the existing ways of dealing with turbulence within the CFD framework is given. The focus is on the problem of the near wall modelling. This problem is important because of the large amount of computational resources that are needed to predict accurately the large changes that occur in this region. In industry, a compromise between accuracy and computational cost has to be achieved when treating flows where the near wall effects are important.

2.1 Turbulence modelling

The Navier-Stokes equations describe the motion of the fluid, they relate the fluid particle acceleration to the surface and body forces that act on the fluid. The momentum conservation equation for a Newtonian incompressible fluid reads:

\[
\rho \frac{\partial (u_i)}{\partial t} + \rho \frac{\partial (u_j u_i)}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial p}{\partial x_i} + \rho g_i \quad (2.1)
\]

where \((u_i)_{i=1,3}\) represents the velocity, \(\rho\) the pressure and \(g\) the gravity in a Cartesian frame with \((x_i)_{i=1,3}\) representing the coordinates. The viscous stress \(\tau_{ij}\) is defined as:

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.2)
\]
where $\mu$ is the molecular viscosity. The continuity equation ensures the mass in conserved is the whole the domain:

$$\frac{\partial (u_i)}{\partial x_i} = 0$$

(2.3)

By solving directly these equations, the fluid velocity and pressure can be obtained. This is known as Direct Numerical Simulation (DNS). In order to obtain accurate results, all the scales must be taken into account when performing DNS. This means that the grid spacing must be of the order of the Kolmogorov scales, therefore imposing a large constraint in the number of points required to solve the flow. DNS is a technique that is not available for industrial flows with present computational power. For instance, to simulate isotropic turbulence, the number of points needed to simulate a cube whose size is the same as the integral length scale ($\lambda$) is $O(Re^{9/2})$. In order to avoid this high constraint on CPU resources, two methods will be discussed in this work, Large Eddy Simulation (LES) and Reynolds Averaged Navier-Stokes (RANS).

### 2.2 Reynolds Averaged Navier-Stokes equations

The Reynolds decomposition of the instantaneous velocity into averaged and fluctuating parts can be written as:

$$u(x, t) = \langle U(x) \rangle + u'(x, t)$$

(2.4)

where the average can be defined as:

$$\langle U_i \rangle = \lim_{\tau \to \infty} \frac{1}{\tau} \int_{t_0}^{t_0+\tau} u_i dt$$

(2.5)

By taking the average of equation (2.1) and using the Reynolds decomposition to evaluate $\langle U_j U_i \rangle = \langle U_j \rangle \langle U_i \rangle + \langle u'_j u'_i \rangle$, the RANS equations are obtained,

$$\rho \frac{\partial \langle U_i \rangle}{\partial t} + \rho \frac{\partial \langle U_j U_i \rangle}{\partial x_j} = -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} (\tau_{ij})$$

$$\rho \frac{\partial \langle U_i \rangle}{\partial t} + \rho \frac{\partial \langle U_j \rangle \langle U_i \rangle}{\partial x_j} = -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \tau_{ij} - \rho \langle u'_i u'_j \rangle \right)$$

(2.6)

$$\frac{\partial \langle U_i \rangle}{\partial x_i} = 0$$

(2.7)
The second term on the right hand side of equation (2.6) represents a correlation of velocity fluctuations called the Reynolds stresses since the term is treated in a similar manner as the viscous stress. All the other terms in equation (2.6) can be calculated but the Reynolds stress needs to be modelled. It is in this closure problem that turbulence modelling comes into play. There are a number of turbulence models that have been developed to capture the most important features in the flow field, ranging from simple algebraic expressions to more complicated differential models. Reynolds stress modelling is presented in the next section before the introduction of more simple eddy viscosity models.

### 2.2.1 Reynolds stress modelling

To solve the closure problem presented in the previous section, it is possible to obtain a transport equation for each Reynolds stress by taking second moments of the fluctuating momentum equation. These equations involve some terms that contain triple moments and therefore further modelling is required. By subtracting the mean momentum equation (2.6) from the instantaneous (2.1) it is possible to obtain an expression for the transport of the stress tensor \( \langle u'_iu'_j \rangle \). By using the Reynolds averaging and taking the second moment of the results, the transport equation reads:

\[
\frac{\partial \langle u'_iu'_j \rangle}{\partial t} + \langle U_k \rangle \frac{\partial \langle u'_iu'_j \rangle}{\partial x_k} = D_{ij}' + D_{ij}^T + \phi_{ij} + P_{ij} + \varepsilon_{ij} \tag{2.8}
\]

where \( D_{ij}' \) is the viscous diffusion, \( D_{ij}^T \) is the turbulent diffusion, \( \phi_{ij} \) is the pressure strain-rate correlation term, \( P_{ij} \) is the production term and \( \varepsilon_{ij} \) is the turbulent dissipation rate term. These terms are:
\[ D'_{ij} = \nu \frac{\partial^2 \langle u'_i u'_j \rangle}{\partial x_k \partial x_k} \quad (2.9) \]

\[ D^T_{ij} = -\frac{\partial}{\partial x_k} \left( \langle u'_i u'_j u'_k \rangle + \left( \frac{p'}{\rho} u'_j \right) \delta_{ik} + \left( \frac{p'}{\rho} u'_i \right) \delta_{jk} \right) \quad (2.10) \]

\[ \phi_{ij} = -\frac{1}{\rho} \left\langle p' \frac{\partial u'_i}{\partial x_j} \right\rangle - \frac{1}{\rho} \left\langle p' \frac{\partial u'_j}{\partial x_i} \right\rangle \quad (2.11) \]

\[ P_{ij} = -\langle u'_i u'_k \rangle \frac{\partial \langle U_j \rangle}{\partial x_k} - \langle u'_j u'_k \rangle \frac{\partial \langle U_i \rangle}{\partial x_k} \quad (2.12) \]

\[ \varepsilon_{ij} = -2\nu \left\langle \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} \right\rangle \quad (2.13) \]

where \( p' \) is the pressure fluctuation and \( \delta_{ij} \) is the Kronecker delta. The terms that require further modelling are the turbulent dissipation rate (2.13), the third order correlations and pressure strain rate correlations (2.10), (2.11).

**Dissipation**

The dissipation term can be written as the sum of the isotropic and deviatoric parts:

\[ \varepsilon_{ij} = \frac{2}{3} \varepsilon \delta_{ij} + D\varepsilon_{ij} \quad (2.14) \]

In most models, the isotropic part is calculated via a transport equation and the deviatoric part is lumped into the pressure-strain correlation:

\[ \phi_{ij, modelled} = \phi_{ij} - D\varepsilon_{ij} \quad (2.15) \]

The transport equation for the isotropic part of the dissipation rate can be derived from the fluctuating momentum equation [41]. The result is a transport equation with higher order terms that require further modelling. Although the inaccuracy of the modelling involved on the dissipation equation is an obvious deficiency, the resulting equation is widely used and is almost standard for all the models, including many two-equations models. The standard transport equation for the dissipation rate proposed by Hanjalić and Launder [50] is:

\[ \frac{\partial \varepsilon}{\partial t} + \langle U_k \rangle \frac{\partial \varepsilon}{\partial x_k} = C_\varepsilon \frac{P_k \varepsilon}{k} - C_\varepsilon^2 \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left( C_\varepsilon \frac{k}{\varepsilon} \left\langle u'_i u'_j \right\rangle \frac{\partial \varepsilon}{\partial x_j} \right) \quad (2.16) \]
2.2. Reynolds Averaged Navier-Stokes equations

where $P_k$ is defined as:

$$P_k = - \langle u'_i u'_j \rangle \frac{\partial \langle U_i \rangle}{\partial x_j}$$  \hspace{1cm} (2.17)

The coefficients $C_\varepsilon$, $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ vary according to the pressure-strain closure but in general, $C_{\varepsilon 2}$ is set to 1.9 to match the decay rate of isotropic turbulence; $C_\varepsilon$ is set between the values 0.15 and 0.18 and $C_{\varepsilon 1}$ usually takes the value of 1.44 [41].

**Diffusion**

The most popular way of representing the diffusive terms (eqn. (2.10)) is the generalised gradient diffusion hypothesis, GGDH [26] which can be written as:

$$D_{ij}^T = - \frac{\partial}{\partial x_k} \left( C_s \frac{k}{\varepsilon} \langle u'_k u'_l \rangle \frac{\partial \langle u'_i u'_j \rangle}{\partial x_l} \right)$$  \hspace{1cm} (2.18)

where $C_s = 0.22$ is a constant determined from model optimisation [41]. Other models have been proposed and can be found in [50], [96] and [41]. It is important to note that some of the diffusion models do not take into account the pressure-diffusion terms; they only model the triple correlation terms present in (2.10). This might be seen as a source of inaccuracy and although it does not seem to be critical in the modelling, it has been called into question [90]. The inclusion of the velocity-pressure correlation can be seen in the value of the empirical constant $C_s$. The model constant obtained without the inclusion of the pressure diffusion is about 20% greater than the usual 0.22.

**Pressure strain**

The pressure-strain correlation term is the term where most of the modelling effort has been made over the last thirty years. This term is important because it has the same order as the production term and it tends to redistribute the energy between the Reynolds stress components, diminishing the difference between them.

Derived from the Navier-Stokes equations and the continuity equation, the pressure field satisfies the Poisson equation:

$$\nabla^2 p = -\rho \frac{\partial u_i}{\partial x_j} \frac{\partial u_k}{\partial x_i}$$  \hspace{1cm} (2.19)
and the fluctuating pressure satisfies:

\[
\nabla^2 p' = -2\rho \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} \left( u'_i u'_j - \langle u'_i u'_j \rangle \right)
\]

(2.20)

The first part is directly dependent on the mean velocity which makes it respond rapidly to the velocity changes. The second part is nonlinear and involves interaction between fluctuating velocities. This decomposition is also carried out into most of the commonly used Second Moment Closures, the pressure-strain term is divided into a slow and a rapid part and then a correction term is added such as the *wall echo* term:

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

(2.21)

One of the most popular models is the linear Launder, Reece and Rodi (LRR) [67] which models the pressure strain term as:

\[
\phi_{ij} = -C_1\varepsilon b_{ij} + C_2 k S_{ij} + C_3 k (b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij})
\]

\[
+ C_4 k (b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik})
\]

(2.22)

where \( b_{ij} \) is the normalised anisotropy tensor, \( S_{ij} \) is the strain tensor and \( \Omega_{ij} \) is the rotation rate tensor defined as:

\[
b_{ij} = \frac{a_{ij}}{2k} = \frac{\langle u_i' u_j' \rangle}{2k} - \frac{1}{3} \delta_{ij}
\]

(2.23)

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right)
\]

(2.24)

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

(2.25)

The constants have the values shown in Table 2.1:

<table>
<thead>
<tr>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_3 )</th>
<th>( C_4 )</th>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0</td>
<td>0.8</td>
<td>1.75</td>
<td>1.31</td>
<td>1.44</td>
<td>1.90</td>
</tr>
</tbody>
</table>

Table 2.1: Coefficients of the LRR model

Another popular model for the pressure-strain correlation term is the Speziale, Sarkar and Gatski model (SSG) [104] which has a quadratic behaviour included originally as a higher
order correction to the slow part of the pressure-strain correlation \[41\]. The model for the pressure-strain term is:

\[
\phi_{ij} = -C_1 \varepsilon b_{ij} + C_1' \varepsilon (b_{ik} b_{kj} - \frac{1}{3} b_{mn} b_{nm}) + C_2 k S_{ij} \\
+ C_3 k (b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij}) \\
+ C_4 k (b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik})
\] (2.26)

The model uses the coefficients showed in Table 2.2. In the coefficient \(C_1\), a dependence on \(P/\varepsilon\) (with \(P = \frac{1}{2} P_{ii}\)) is introduced to achieve the correct asymptotic behaviour of the Taylor series expansion of \(\phi_{ij}\).

<table>
<thead>
<tr>
<th>(C_1)</th>
<th>(C_1')</th>
<th>(C_2)</th>
<th>(C_3)</th>
<th>(C_4)</th>
<th>(C_{e1})</th>
<th>(C_{e2})</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.4 + 1.8P/\varepsilon</td>
<td>4.2</td>
<td>0.8 - 1.3(b_{ij} b_{ij})^{0.5}</td>
<td>1.25</td>
<td>0.4</td>
<td>1.44</td>
<td>1.83</td>
</tr>
</tbody>
</table>

Table 2.2: Coefficients of the SSG model

There are many other pressure-strain models including higher order models that will not be discussed here and can be found in \[41\] and \[96\]. As the complexity of the models increase, also does the computational cost and the implementation difficulties. Although the Reynolds stress models are capable of predict individual components of the stress tensor, they are not the most used in industry since they require large memory resources and CPU time (7 additional transport equations). Most commercial CFD codes have recently started to implement these models but their usage has been limited partially due to the fact that most models are not code-friendly.

### 2.2.2 Eddy viscosity models

Second Moment Closures (SMC) provide information on the Reynolds stresses in the RANS equations but they do it at the expense of rather complex differential equations and the need to model higher order correlations. Their computational costs and implementation complexity have kept them far from industrial use. In industrial applications the need for fast and reliable models makes the use of SMC unappealing and therefore there is the need to obtain an approximation of the Reynolds stress term in a different way. The turbulence viscosity
approximation introduced by Boussinesq in 1877 states that the deviatoric Reynolds stress is proportional to the mean rate of strain, that is:

$$-\rho \langle u'_i u'_j \rangle + \frac{2}{3} \rho k \delta_{ij} = \rho \nu_t \left( \frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right)$$

(2.27)

Here, $\nu_t$ is the turbulent viscosity or eddy viscosity. Based on this approximation, the closure problem of calculating 6 different equations for $\langle u'_i u'_j \rangle$ is changed to model only one scalar, $\nu_t$.

There are several different models for calculating the turbulent viscosity: zero, one and two equation models; linear and non-linear models and models with elliptic relaxation. But although they are relatively cheap, the Boussinesq approximation prevents them from giving accurate results in inhomogeneous flows with strong anisotropy.

**Zero and one-equation models**

Prandtl’s mixing length theory (Prandtl, 1925) is a zero equation model since the eddy viscosity is assumed to have the form:

$$\nu_t = l^2 \left| \frac{du}{dy} \right|$$

(2.28)

Where $u$ is the velocity in the stream-wise direction and $y$ is the wall normal direction. Here, $l$ is the mixing length scale that has to be defined for each specific case; for example in a free shear flow, the mixing length would be the characteristic measure of the width of the shear layer. In the log-layer, $l = \kappa y$ but in the outer part of the flow, the mixing length is made proportional to the boundary layer thickness:

$$l = \kappa y, \quad y < 0.2\delta$$

(2.29)

$$l = 0.2\kappa \delta, \quad y \geq 0.2\delta$$

For the near-wall region, Van Driest proposed a damping formula to reduce turbulent mixing:

$$l = \kappa y \left( 1 - e^{-y^+ / A_+} \right)$$

(2.30)

Here $\kappa = 0.41$ is the von Karman constant, $A_+ = 26$ is the damping coefficient and $y^+ = u_+ y / \nu$ is the nondimensional distance.
2.2. Reynolds Averaged Navier-Stokes equations

One-equation models solve a differential transport equation for a scalar that can give a better approximation when calculating the eddy viscosity since it is no longer based on the explicit definition of the length scale. Prandtl (1945) proposed a one-equation model to calculate the turbulent viscosity as:

\[ \nu_t = \frac{k^{\frac{3}{2}}}{l} \]  

(2.31)

Where \( k \) is the turbulent kinetic energy given by \( k = \frac{1}{2} \langle u'_i u'_i \rangle \). To calculate \( k \), Prandtl provided a differential transport equation as:

\[ \frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \left( \nu + \nu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right) \right] \]  

(2.32)

Here \( P_k \) is the production of turbulent kinetic energy and \( \varepsilon \) is the dissipation, given by:

\[ P_k = 2 \nu_t S_{ij} S_{ij} \]  

(2.33)

\[ \varepsilon = C_D \frac{k^2}{l} \]  

(2.34)

Another popular one-equation model is the Spalart-Allmaras [102] model that uses a transport equation for the viscosity including eight closure coefficients and three damping functions, in a similar way to the Baldwin-Barth models (see [118] for more details).

Two-equation models

In order to get a more general idea of the time and length scales that govern the eddy viscosity in different flows, two-equation models are usually referred to as complete models that can be used to predict properties of a given turbulent flow with no prior knowledge of the turbulent structure [118]. Most of the two-equation models solve the transport equation for \( k \) and some other turbulent quantity that can be used to compute the turbulent viscosity. It is in the choice of the second variable where the main difference lies. Some models calculate the dissipation \( \varepsilon \) and then the length scale is obtained as \( l \sim \frac{k^{\frac{3}{2}}}{\varepsilon} \). Others use the turbulent frequency \( \omega \) (dissipation rate per unit of kinetic energy) and then the length scale is \( l \sim \frac{k^{\frac{3}{2}}}{\omega} \). There are models that solve directly for the \( k \) and \( l \) or for \( k \) and turbulence dissipation time [118].

One of the most popular (and still widely used in industry) two-equation models is the \( k - \varepsilon \) model by Jones and Launder [60]. By solving two turbulent quantities, the turbulent
kinetic energy $k$ and the rate of dissipation $\varepsilon$, the eddy viscosity is calculated as:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}$$  \hspace{1cm} (2.35)

The model equation for the dissipation is more complicated than for the $k$ equation, and although the exact equation can be derived from the Navier-Stokes equations, it is of little use since it involves several new unknown double and triple correlations of fluctuating velocities, pressure and velocity gradients \[118\]. The actual $\varepsilon$ equation is best viewed as being entirely empirical \[96\], and is given by:

$$\frac{\partial \varepsilon}{\partial t} + \langle U_j \rangle \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{P_k \varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \frac{\sigma_k}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$ \hspace{1cm} (2.36)

The closure coefficients are shown in Table 2.3. These coefficients are obtained by matching experiments with simple cases, such as decaying turbulence and homogeneous shear flows \[96\] but there is a range of values for the constant $C_{\varepsilon 1}$. This model has been widely used and has been implemented in almost all major commercial codes. It has many advantages from the user’s point of view. It is robust, easy to implement and it does not need much computer power compared to the second moment closures. Although it is reasonably accurate for simple flows, in more complex flows it can be qualitatively inaccurate because it does not take into account anisotropy of the Reynolds stresses. One of the main advantages is that it has been tested for many years in a wide range of cases so its deficiencies are known and there have been several proposed corrections to improve the universality of the model.

The standard $k - \varepsilon$ is a high Reynolds model, that means it has been developed to account for the turbulent effects outside the viscous sublayer, far from solid boundaries. There have been different formulations to try to adapt the model to a low-Reynolds form that will resolve the equations all the way down to the wall. It is a difficult task since the viscous sublayer is small compared with the log-layer and there is the need of refining the mesh near the wall to capture the effects of the rapid change of velocities and of turbulence variables. Different strategies to do so will be discussed in Chapter 3.

Table 2.3: Coefficients of the standard $k - \varepsilon$ model

<table>
<thead>
<tr>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_\mu$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.44</td>
<td>1.92</td>
<td>0.09</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>
2.2. Reynolds Averaged Navier-Stokes equations

Another popular model in CFD commercial codes is the Wilcox $k - \omega$ [118], which uses the dissipation per unit kinetic energy $\omega \sim \varepsilon/k$ as a second variable, as proposed first by Kolmogorov in 1942. In the case of the $k - \omega$ model, the turbulent viscosity is defined as:

$$\nu_t = \frac{k}{\omega}$$

and the turbulent kinetic energy and dissipation rate are obtained from their own transport equations:

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{\partial \omega}{\partial t} + \langle U_j \rangle \frac{\partial \omega}{\partial x_j} = \alpha \omega k P_k - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \omega}{\partial x_j} \right]$$

Using the two variables, the dissipation and length scale can be obtained by:

$$\varepsilon = \beta^* \omega k \quad \text{and} \quad l = \frac{k^{\frac{1}{2}}}{\omega}$$

The closure coefficients are shown in Table 2.4.

<table>
<thead>
<tr>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>$\beta^*$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.555</td>
<td>0.075</td>
<td>0.09</td>
<td>2</td>
<td>2</td>
</tr>
</tbody>
</table>

Table 2.4: Coefficients of the standard $k - \omega$ model

A remarkable feature of the $k - \omega$ model is that no damping functions are needed to compute the near wall turbulence. The boundary conditions at the wall are:

$$k \to 0 \quad \text{and} \quad \omega \to 6 \frac{\nu}{\beta^* y^2} \quad \text{as} \quad y \to 0$$

(2.41)

For free shear flows, all two-equation models present a discontinuity in the derivatives of flow properties in the turbulent-non turbulent interface. The three turbulent variables, $k$, $\omega$ and $\varepsilon$ vanish approaching the edge of the shear layer. The $k - \varepsilon$ model is not very sensitive to these free-stream values, but the $k - \omega$ model is, and that is one of its main disadvantages. The $k - \omega$ model is still very attractive for wall bounded calculations since it has been proved to have a better performance in separated flows [118]. These two models have been widely used in commercial codes. In many respects, the $k - \varepsilon$ model is robust and reliable, it has a wide
range of applicability although it tends to give the wrong behaviour on flows with adverse pressure gradients. In addition, the model does not behave well in the viscous sublayer and most of the modifications proposed for a low Reynolds formulation are based upon empirical relations tuned for specific cases. The $k - \omega$ model performs better in cases with variable pressure gradients and have similar numerical properties to the $k - \varepsilon$ but it is extremely sensitive to the free-stream boundary condition on $\omega$.

The different advantages of the previous two models have been combined in the Shear Stress Transport (SST) model of Menter [81]. As a first step, Menter proposed a blending Baseline Model (BSL) model between the $k - \omega$ and the $k - \varepsilon$. By rewriting the $k - \varepsilon$ equations in terms of the rate of dissipation per unit of kinetic energy $\omega$, then the extra cross gradient term that emerges is multiplied by a blending function, allowing a return to the original $k - \omega$ near wall boundaries. The blending function is also used in the calculation of all the closure coefficients. The equations of the BSL model are:

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

(2.42)

$$\frac{\partial \omega}{\partial t} + \langle U_j \rangle \frac{\partial \omega}{\partial x_j} = \alpha \omega \kappa P_k - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \left( 1 - F_1 \right) \frac{1}{\sigma_\omega^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

(2.43)

The model uses a value of any coefficient $c$ computed from:

$$c = F_1 c_1 + (1 - F_1) c_2$$

(2.44)

where the subscript 1 represents the original coefficient of the $k - \omega$ model and the subscript 2 represents the coefficients of the transformed $k - \varepsilon$ model. The blending function is defined such that it makes a smooth transition from $k - \omega$ at the wall to $k - \varepsilon$ far from it. In this way the sensitivity of $\omega$ in the free stream is reduced and the problem in the viscous sublayer of the $k - \varepsilon$ does not play a part in the solution. The blending function is given by:

$$F_1 = \tanh(\arg_1^4)$$

(2.45)

$$\arg_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09 \omega y^2}; \frac{500 \nu}{y^2 \omega} \right); \frac{4 \rho k}{\sigma_\omega^2 CD_{k\omega} y^2} \right]$$

(2.46)

$$CD_{k\omega} = \max \left( 2 \rho \frac{1}{\sigma_\omega^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-20} \right)$$

(2.47)
2.2. Reynolds Averaged Navier-Stokes equations

here, \( y \) represents the distance from the wall, which can be difficult to define in complex geometries. This wall distance dependence is undesirable but not uncommon. Many low Reynolds number models need some sort of parameter that can give an estimation of the distance to the nearest wall.

The second step presented by Menter [81] is the Shear Stress Transport model (SST), in which a function similar to the blending function \( F_1 \) of the BSL is used to limit the value of the eddy viscosity, therefore maintaining the shear stress proportional to the turbulent kinetic energy in the wake of the boundary layer. The SST model uses the same transport equations as the BSL model for \( k \) and \( \omega \) eqs. (2.42), (2.43) and the same definition for \( F_1 \) (2.46) but with the addition of:

\[
\nu_t = \frac{a_1k}{\max(a_1\omega; SF_2)}
\]

(2.48)

\[
F_2 = \tanh(\arg g_2^2)
\]

(2.49)

\[
\arg g_2^2 = \max\left(\frac{2\sqrt{k}}{0.09\omega y}; \frac{500\nu}{\nu^2\omega}\right)
\]

(2.50)

where \( S = \sqrt{2S_{ij}S_{ij}} \). This model has been tested in academic cases such as the backward facing step flow, NACA 4412 aerofoil and transonic bump flow [81] with good results, and better predictions than any of the other two models. It has also been tested on an asymmetric plane diffuser [7] with better results compared to the \( k - \omega \) and to the \( k - \varepsilon \). The closure coefficients for the SST model are shown in Table 2.5. The boundary condition on \( \omega \) proposed by Menter is given by:

\[
\omega_w = 10\frac{6\nu}{\beta_1 y^2}
\]

(2.51)

<table>
<thead>
<tr>
<th>( \alpha_1 )</th>
<th>( \beta_1 )</th>
<th>( \sigma_{k1} )</th>
<th>( \sigma_{\omega1} )</th>
<th>( a_1 )</th>
<th>( \kappa )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \beta_1/\beta^* - \kappa^2/(\sigma_{\omega1}\sqrt{\beta^*}) )</td>
<td>0.075</td>
<td>1.176</td>
<td>2</td>
<td>0.31</td>
<td>0.41</td>
</tr>
<tr>
<td>( \alpha_2 )</td>
<td>( \beta_2 )</td>
<td>( \sigma_{k2} )</td>
<td>( \sigma_{\omega2} )</td>
<td>( \beta^* )</td>
<td></td>
</tr>
<tr>
<td>( \beta_2/\beta^* - \kappa^2/(\sigma_{\omega2}\sqrt{\beta^*}) )</td>
<td>0.0828</td>
<td>1.0</td>
<td>1.1682</td>
<td>0.09</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.5: Coefficients of the SST model
Non-linear eddy viscosity models

Another approach to model the eddy viscosity is to express the anisotropy tensor $a_{ij}$ as a nonlinear dependence on the strain and vorticity terms. Pope [95] showed that if $a_{ij}$ depends only on $S_{ij}$ and $\Omega_{ij}$, it can be expressed in terms of ten tensorial groups

$$a_{ij} = \beta_1 S_{ij} + \beta_2 (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) + \beta_3 (S_{ik} S_{kj} - \frac{1}{3} S_{lk} S_{kl} \delta_{ij}) +$$
$$\beta_4 (\Omega_{ik} \Omega_{kj} - \frac{1}{3} \Omega_{lk} \Omega_{kl} \delta_{ij}) + \beta_5 (\Omega_{it} S_{lm} S_{mj} - S_{it} S_{lm} \Omega_{mj}) +$$
$$\beta_6 (\Omega_{it} \Omega_{tm} S_{mj} - S_{it} \Omega_{tm} \Omega_{mj} - \frac{2}{3} S_{lm} \Omega_{mn} \Omega_{nt} \delta_{ij}) +$$
$$\beta_7 (\Omega_{ik} \Omega_{kl} S_{tm} S_{mj} - \Omega_{ik} \Omega_{kl} S_{lm} \Omega_{mj}) + \beta_8 (S_{ik} S_{kl} S_{tm} \Omega_{mj} - S_{ik} S_{kl} \Omega_{tm} \Omega_{mj}) +$$
$$\beta_9 (\Omega_{ik} \Omega_{kl} S_{tm} S_{mj} + S_{ik} S_{kl} \Omega_{tm} \Omega_{mj} - \frac{2}{3} S_{kl} S_{tm} \Omega_{mn} \Omega_{nk} \delta_{ij})$$

(2.52)

Some examples of models developed this way are the Craft et al. [22] and Yoshizawa [122] but it is beyond the scope of this work.

2.3 Large Eddy Simulation

Another approach to resolve turbulent flows is the Large Eddy Simulation (LES) technique, in which only the large scales are resolved and the small scales are modelled. The theory on which LES is based is the assumption that the larger scales in the flow are the ones containing most of the energy, are anisotropic, and depend on the flow geometry, therefore not universal. The small scales can be considered more isotropic, dissipative, and more universal and can be modelled in a simple way. Most of the pioneering work on LES dates back more than 30 years ([100], [72] and [31]) when it was introduced to compute atmospheric boundary layers. LES is a useful tool when considering highly unsteady or separated flows where most RANS models fail to give accurate prediction. It is also a cheaper option than DNS. The main issue of LES is how to separate the ’large’ scales from the ’small’ ones. This is done by applying a filter to the velocity field, by splitting the total velocity into resolved and residual parts:

$$u(x, t) = \overline{U}(x, t) + u'(x, t)$$

(2.53)
The filtering procedure can be done with a variety of filters (hat, Gaussian, spectral [96]) but in practice, the filtering procedure and the numerical uncertainties are mixed together by using the grid as a filter. That is why the filtered field is usually referred to as 'resolved' and the residual field as 'subgrid'. The filtered momentum equations can be written as (assuming that the filter and derivative operators commute):

$$\frac{\partial \bar{U}_j}{\partial t} + \frac{\partial \bar{U}_i \bar{U}_j}{\partial x_i} = \nu \frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_i} - \frac{1}{\rho} \frac{\partial P}{\partial x_j}$$  \hspace{1cm} (2.54)

This differs from the Navier Stokes equations because the product of the filtered velocities is not the same as the filter of the product of the velocities i.e. $\bar{U}_i \bar{U}_j \neq \bar{U}_i \bar{U}_j$. The product $\bar{U}_i \bar{U}_j$ can be expressed in term of the residual stress tensor as $\bar{U}_i \bar{U}_j = \bar{U}_i \bar{U}_j + \tau^R_{ij}$ and the momentum equation can be written as:

$$\frac{\partial \bar{U}_j}{\partial t} + \frac{\partial \bar{U}_i \bar{U}_j}{\partial x_i} = \nu \frac{\partial^2 \bar{U}_j}{\partial x_i \partial x_i} - \frac{\partial \tau^R_{ij}}{\partial x_i} - \frac{1}{\rho} \frac{\partial P}{\partial x_j}$$  \hspace{1cm} (2.55)

The sub grid tensor $\tau^R_{ij}$ must be modelled in terms of the resolved (filtered) velocities. A simple model was introduced by Smagorinsky [100] based on a linear eddy viscosity approximation (same idea as the Bousinessq approximation for RANS):

$$\tau^R_{ij} - \frac{1}{3} \tau^R_{ii} \delta_{ij} = -2 \nu_r \bar{S}_{ij}$$  \hspace{1cm} (2.56)

where is $\nu_r$ is the eddy viscosity of the residual motions. This viscosity is modelled by analogy to the mixing length hypothesis; it is assumed to be proportional to the length scale and the filtered strain. The length scale is obtained by using the filter width $\Delta$ [32].

$$\nu_r = (C_s \Delta)^2 \bar{S}$$  \hspace{1cm} (2.57)

Here $C_s$ is the Smagorinsky constant. The filter width is proprotional to the cell volume $\Delta = 2(\Delta_x \Delta_y \Delta_z)^{1/3}$. To evaluate $C_s$ Lilly [72] assumed an inertial range spectrum and deduced a value of $C_s = 0.17$. Deadroff [32] found that the value of the constant should be reduced in the presence of mean shear. Values around $C_s \approx 0.065$ are generally used for wall bounded flows.

A way of dynamically compute the coefficients for the subgrid scale stresses was proposed by Germano et al. [42] by defining a test filter whose width is larger than the grid filter. More information on other types of subgrid scale models can be found in [96].
2.4 Hybrid models

The use of the LES technique has been growing in recent years due to the attractive prospect of being able to solve 3D flows accurately. By resolving the large scales and modelling the small ones, the main characteristics of the flow can be predicted reasonably well. Since the small scales can be modelled in a fairly simple way, a well resolved LES is a powerful tool. The great disadvantage of LES becomes obvious in boundary layers where the number of points required to resolved the near wall layer increases with $Re^2$ [10]. Due to the anisotropy near the wall, the resolution of a LES computation must increase since it is assumed that the small scales are basically isotropic. In an attempt to ease the computational requirements in wall bounded flows, many approaches have been suggested. One method is to use so-called "wall functions" to bridge the viscous sublayer and provide a suitable boundary condition for the wall cells [93]. This can range from a log-law approximation [99] to a solution of a system of simplified equations in the near wall region [11]. A method where the wall shear stress is imposed dynamically was studied by Templeton et al. [111]. They used a correction of the turbulent viscosity based on equating the pressure gradient for the RANS and LES averaged equations in an channel flow.

Another approach is the use of RANS equations near the wall to provide the outer layer with correct information. The main problem of this type of approach is how to connect a statistically averaged flow (RANS) with the instantaneous filtered field (LES). A way to couple the two types of flows is the Detached Eddy Simulation (DES) ([101], [113]) in which the turbulent lengthscale in the RANS equation is switched to a lengthscale based on the mesh filter width in order to reduce the viscosity in the separated region:

$$l_{DES} = \min(l_{RANS}, C_{DES} \Delta)$$ (2.58)

where $l_{DES}$ is the length scale used in DES, $l_{RANS}$ is the length scale from the RANS model, $C_{DES}$ is a model constant and $\Delta = \max(\Delta_x, \Delta_y, \Delta_z)$ is a length scale based on the grid dimension. The main idea of the DES approach is to solve the attached boundary layers with a RANS models and the separation region with a LES-like technique. One of the main issues of the DES approach is the fact that in regions near the wall where the grid is refined, the turbulent length scale dictated by the RANS model can become larger than the grid length.
2.4. Hybrid models

scale therefore making the model reduce the turbulent viscosity and leading to a *grid induced separation* [79]. Originally the DES formulation used the Spalart-Allmaras model, but a range of RANS models have since been tested [82]. As its name implies, DES can only be applied for separated flows and when the resolution of near wall fluctuations is required, another approach should be used.

Other approaches are ’zonal’, in which a part of the domain is set to be computed using RANS equations and the rest is computed with LES (see Figure 2.1). Davidson and Peng [30] used the $k - \omega$ model in a region near the wall ($y^+ \leq 60$) and a one equation model for the sub-grid stresses in the outer region. The location of the interface was fixed and Neumann boundary conditions were applied for the RANS variables. The results for the channel flows calculations were better than those using pure LES on the same grid, but they showed dependence on the matching plane location. Additionally the velocity profile showed a kink or a sudden acceleration at the interface. On the flow over periodic hills, the kink was much smaller probably due to the smoothing produced by the convective and diffusive transport. Davidson and Peng [30] found that the RANS part does not contain enough turbulent characteristics and therefore the LES region is not fed with the correct information. This led to impose turbulent fluctuations at the interface either from a DNS database [29] or from a synthetic method [28]. By imposing fluctuations, it was found found that the channel flow predictions are in much better agreement with DNS than without forcing. They also tested the method on an asymmetric plane diffuser and found that it performed better than most RANS predictions, especially in the recovery region after reattachment.

Another zonal approach has been developed by Temmerman et al. [108] where continuity of the turbulent viscosity is imposed at the interface. Different RANS models and different locations for the interface have been tried. Using this constraint, the coefficient $C_\mu$ is extracted from the interface and then adapted via exponential functions to increase the RANS contribution as the wall is approached. The results for a channel flow simulations at high Reynolds number, although not perfect, were far superior then a standard LES on the same coarse mesh. The methodology was also tested in a flow over a series of periodic hills, in which the results were not as good as the pure LES on the same grid, possibly due to the defect in the wall shear stress returned by the RANS model. The method seemed to be only
Hamba [48] tried coupling a near-wall RANS region with an outer LES by using a lengthscale computed from DNS. The approach resulted in an acceleration of the velocity profile at the interface, which was compared to another approaches where there is a similar effect [30], [85]. To get rid of the problem an extra filtering is added and another velocity is defined at the interface and hence the velocity mismatch is reduced.

The direction followed by Abe [4] was based on the blending of the total stress \( \tau_{ij} \) as:

\[
\tau_{ij} = (1 - f_{hb}) \langle u_i' u_k' \rangle_{RANS} + f_{hb} \tau_{ij}(LES)
\]  

(2.59)

A non-linear eddy viscosity model was used for the calculation of the stresses in both regions, the only difference being that the RANS part was calculated using transport equations for \( k \) and \( \varepsilon \) whereas the LES part used algebraic definitions. The blending function was parametrised by the ratio of the distance to the wall and the grid size. Results were shown for channel flows of Reynolds numbers up to \( Re_\tau = 10000 \) with good results for mean quantities, but with a stronger stress anisotropy predicted due to the suppression of the energy redistribution from the streamwise to the wall normal and spanwise components.

In the zonal approach, the treatment of the interface has always been of importance for the success of the method since the RANS information does not provide turbulent characteristics. Some ways to deal with this issue are the introduction of backscatter [94], damping the modelled stresses [109], the addition of fluctuations at the LES side of the interface [29].
or the use numerical smoothing [114].

2.5 Wall treatment

In turbulent flows, the region far from the wall is characterised by small viscous stresses compared to turbulent stresses and it is under this assumption that all the high-Reynolds models have been designed. But near the wall, as the viscous sublayer is approached, the turbulent stresses vanish due to the no-slip condition and the viscous stresses start playing an important role. One way of dealing with the wall effects in a turbulent flow is to include them in the model equations and integrate the equations over the whole domain. This approach is discussed in Chapter 3. This methodology usually requires a fine grid near the wall since the gradients of the turbulent variables are greater in this region which implies a higher computational cost.

2.5.1 Standard wall function

Another way of dealing with the wall effects is the use of so-called wall functions. This approach is based under the assumption that the viscous effects play an important role in a very thin layer near the wall and, providing there is a way of calculating the turbulent variables outside this layer, a solution can be obtained without integrating right up to the wall. In the derivation of the conventional wall functions, a steady two-dimensional boundary layer is assumed for which the momentum equation reduces to:

\[ \frac{\partial}{\partial y} \left( \mu \frac{\partial U}{\partial y} - \rho \langle u'v' \rangle \right) = 0 \] (2.60)

where \( U \) is the streamwise component (\( \langle U_1 \rangle \)) and \( y \) is the wall normal direction \( (x_2) \). This equation can be integrated with the boundary conditions at \( y = 0, \rho \langle u'v' \rangle = 0 \) hence obtaining:

\[ \frac{\partial U}{\partial y} - \rho \langle u'v' \rangle = \tau_w \] (2.61)

Outside the buffer layer (around \( y^+ = 30 \)) the viscous stresses are small in comparison with the turbulent stresses and the first term in equation (2.61) can be neglected. By using the
mixing length hypothesis with $l = \kappa y$ a value for the derivative of velocity is obtained:

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

(2.62)

$$\frac{\partial U}{\partial y} = \frac{\sqrt{\tau_w / \rho}}{\kappa y}$$

(2.63)

which can be integrated to obtain:

$$U^+ = \frac{1}{\kappa} \ln y^+ + C$$

(2.64)

where $U^+ = U/u_*$, $y^+ = y u_*/\nu$ and $u_* = \sqrt{\tau_w / \rho}$. The constants usually have the values: $\kappa = 0.41$ and $C = 5$. The wall function method has as strong physical constraints, since it is based on the universality of the log law, but it is usually used in many types of flow, mainly because of its principal strength: the low computational cost. For the $k - \varepsilon$ model, the values at the log layer boundary are:

$$k = \frac{u_*^2}{\sqrt{C \mu}}, \quad \varepsilon = \frac{u_*^3}{\kappa y_p}$$

(2.65)

These are an exact solution to the equations in a constant stress layer, with $y_p$ being the distance from the wall to the nearest computational point. One problem posed by these functions is at separation points where $u_* = 0$. To avoid such problem the boundary conditions are expressed in terms of $u_k$, given by:

$$u_k = \sqrt{\frac{k}{C \mu^{1/2}}}$$

(2.66)

$$\frac{dU}{dy} = \frac{u_k}{\kappa y_p}, \quad \varepsilon = \frac{u_k^3}{\kappa y_p}, \quad \frac{dk}{dy} = 0$$

(2.67)

The wall function should be applied only in the region where it is prescribed for, that is at a $y^+ \geq 30$. In practice, ensuring that the first point of the mesh has a specific value of $y^+$ is difficult since a solution of the velocity field is necessary to obtain the value of $y^+$, so the iterative process of refining the mesh can become difficult for complex geometries. Even if the mesh has been set up for a correct $y^+$ value for the first point near the wall, the universality of the wall function is not consistent with flows with strong lateral pressure gradients, separation, reattachment or strong curvature so the predictions can deteriorate.
Other types of wall functions that can take into account such effects have been developed, for instance the two layer model of Chen and Patel [19] or the Unified Modelling through Integrated Sublayer Treatment [24], [40] but they will not be discuss here.

### 2.5.2 Scalable wall function

A similar approach of industrial interest is the use of scalable wall functions. In flows where it is not necessary to resolve the viscous effects, a wall function approach can be used. Apart from the assumption that the log-law is universal, the main disadvantage of this approach is the $y^+$ dependence. From the industrial point of view, in complex geometries, making sure that all the near wall cells are within a specific range of values of $y^+$ can be difficult. It also makes mesh dependence studies more time consuming. The scalable wall function approach [44] is based on limiting the minimum value of $y^+$ so the value of the velocity gradient at the first cell will be the same as if it was on the edge of the viscous sublayer. First $u_k$ is calculated as:

$$u_k = (C_\mu) \frac{1}{4} \sqrt{k}$$  \hspace{1cm} (2.68)

Then a new value of $y^+$ is calculated, denoted by $y_b^+$, using $u_k$

$$y_b^+ = \max \left( \frac{u_k y}{\nu}, y_{\text{lim}}^+ \right) \quad \text{with} \quad y_{\text{lim}}^+ = 11.06$$  \hspace{1cm} (2.69)

Once the values of $y_b^+$ are obtained, the log law can be used to calculate the friction velocity:

$$u_T = \frac{U}{\frac{1}{k} \log(y_b^+)} + c$$  \hspace{1cm} (2.70)

In a finite-volume framework (see Chapter 5), with the values of $u_k$ and $u_T$, the flux over the wall boundary face can be computed as:

$$\tau_{\text{wall}} = \rho u_T u_k$$  \hspace{1cm} (2.71)

The physical interpretation of this procedure is that the wall can be seen as the limit of the logarithmic layer. When the grid is fine, below $y_{\text{lim}}$, the relation between $u_T$ and $U$ becomes linear, that is: $U = \alpha u_T$ with $\alpha = \frac{1}{k} \log(11.06) + 5 = 10.86$ instead of the buffer layer relation.
where $\alpha$ is a complex function of $y$ and $u_\tau$ ending with $U = u_\tau^2 y / \nu$ in the viscous sublayer. Note that the production:

$$P = - \langle u' v' \rangle \frac{\partial U}{\partial y} = - \langle u' v' \rangle u_\tau \frac{\partial \alpha}{\partial y} = 0$$

is shut off when $y^+ < y_{lim}$, regardless of the rate at which $\langle u' v' \rangle$ is going to zero.

The formulation does not take into account what is really happening inside the viscous sublayer and therefore for fine grids the resulting velocity profile will not be correct for all points inside the viscous sublayer. But this is also the case for the standard wall function approach in general. The other advantage of the scalable wall function is that in finite-volume based codes it allows for more points inside the boundary layer without a high expansion ratio between the first cell and the second.
Chapter 3

Wall modelling

3.1 Wall effects

Most of the RANS models described so far are designed to work outside the viscous sublayer, that means that they do not take into account the near wall effects and therefore have to be used in conjunction with a wall function approach (see Section 2.5). In the near wall region, viscosity plays an important role. It is where most of the turbulence is produced and is strongly inhomogeneous. The behaviour of this thin layer influences the skin friction coefficient, separation points and mass and heat transfer. It is therefore desirable to model the effects of the wall when any of these parameters are important. In a 2D boundary layer (where \( U = \langle U_1 \rangle \) and \( V = \langle U_2 \rangle \)), some of the effects of the wall proximity are:

- Low Reynolds number since \( Re_t = k^2/(\varepsilon \nu) \) tends to zero.

- High shear rate, since \( \partial U/\partial y \) increases near the wall. This is where most of the turbulent kinetic energy is produced.

- Two component turbulence, since as the wall is approached \( \langle v'v' \rangle \) behaves like \( y^4 \) whereas \( \langle u'u' \rangle \) and \( \langle w'w' \rangle \) vary as \( y^2 \).

- Wall blocking effects, since the impermeability condition \( V = 0 \) affects the flow via the pressure field. The damping of the wall-normal fluctuations creates an increase in the pressure that can be felt up to an integral length scale away from the wall [77].
• Echo effects. In the presence of the wall, the pressure redistribution term is increased by a contribution of the 'mirror points' on the other side of the wall.

The first three items are dynamic effects, the last two are kinematic or non-local effects (see Figure [3.1]). The no slip condition on the stresses $\langle u_i' u_j' \rangle = 0$ ensures the correct value at the wall but it is often of importance how these stresses approach the wall. The correct asymptotic behaviour is desireable in a near wall turbulence model. If $y$ is the wall normal direction, the instantaneous velocity can be expanded in Taylor series as:

$$u_i = a_i + b_i y + c_i y^2 \ldots$$  \hspace{1cm} (3.1)

From the no slip condition, $a_i = 0$. The continuity equation $\partial u_i / \partial x_i = 0$ imposes $b_2 = 0$ therefore the asymptotic behaviour as $y \to 0$ can be written as:

$$\langle u' u' \rangle = O(y^2) \quad \langle v' v' \rangle = O(y^4) \quad \langle w' w' \rangle = O(y^2)$$  \hspace{1cm} (3.2)

The kinematic blocking can be seen as an addition to the pressure strain term by a reflective wave on the wall. The rapid part of the fluctuating pressure equation (2.20) can be solved using the free space Greens function $1/4\pi|x - x'|$ as:

$$p'(x) = \frac{1}{2\pi} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{0}^{\infty} \frac{\rho \partial U_k / \partial x_i \partial u'_l / \partial x_k}{[(x - x')^2 + (z - z')^2 + (y - y')^2]^{1/2}} dy'dx'dz'$$

$$+ \frac{1}{2\pi} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{0}^{\infty} \frac{\rho \partial U_k / \partial x_i \partial u'_l / \partial x_k}{[(x - x')^2 + (z - z')^2 + (y + y')^2]^{1/2}} dy'dx'dz' \quad (3.3)$$

From this equation it can be seen that there is a contribution from a mirror point on the other side of the wall ($\pm y$) (see Figure [3.1]). The two integrals can be regarded as representations of the contributions of the incident pressure wave and of the reflected one, enhancing the total pressure fluctuations. Manceau et al. [77] showed that, contrary to many second closure models, the wall echo increases the redistribution.

Most of the suggested modifications for the near wall treatment include a series of damping functions that usually depends on the wall distance. The wall echo effects have been modelled by adding a term to the pressure strain like the one proposed by Gibson and Launder
3.2. Damping functions

but they have had little success for complex flows \[21\]. They also have the disadvantage that they need to be readjusted for each homogeneous redistribution model.

![Figure 3.1: Wall effects](image)

### 3.2 Damping functions

In order to obtain solutions inside the viscous sublayer, the models derived for the outer layer need to be modified to capture the wall effects. Over the years a constant practice has been the use of damping functions. These functions suppress the effects of the equilibrium assumption in an attempt to obtain a correct behaviour of the variables to solve. The earliest damping function was proposed by Van Driest \[116\] in the context of the mixing length hypothesis in which the length is defined as

\[
l = \kappa y \left( 1 - \exp \left( y^+/26 \right) \right)
\]

where \( y^+ \) is the nondimensional distance to the wall. Although this damping is empirical, it has been used widely in CFD and it is also used in LES with the Smagorisky model to damp the subgrid scale turbulent viscosity near the wall.

An example of a second moment closure using damping functions is the Low-Reynolds SSG model of Chen \[20\] which introduces two damping functions; one for the pressure-strain correlation model and another for the dissipation equation.
The pressure-strain term is written as:

$$\phi_{ij} = -C_1 \left(1 - \left(1 - \frac{1}{C_1}\right) f_w \right) \varepsilon a_{ij} - C'_1 \left(1 - f_w \right) \varepsilon (a_{ik} a_{kj} - \frac{1}{3} a_{ij} a_{ji} \delta_{ij}) + C_2 k (S_{ij} - \frac{1}{3} S_{kk} \delta_{ij})$$

$$+ C_3 k (a_{ik} S_{jk} + a_{jk} S_{ik} - \frac{2}{3} a_{mn} S_{mn} \delta_{ij})$$

$$+ C_4 k (a_{ik} \Omega_{jk} + a_{jk} \Omega_{ik})$$

$$+ f_w \left(0.45(P_{ij} - \frac{2}{3} P_{kk} \delta_{ij}) - 0.03(Q_{ij} - \frac{2}{3} Q_{kk} \delta_{ij}) + 0.08k(2S_{ij})\right)$$

(3.5)

with $Q_{ij} = \langle -u'_i u'_k \rangle \partial_j \langle U_k \rangle - \langle u'_j u'_k \rangle \partial_i \langle U_k \rangle$.

Here the damping function $f_w$ only acts near the wall and returns to the original SSG model far from it. The coefficients used in this model are presented in Table 3.1.

<table>
<thead>
<tr>
<th>$C_1$</th>
<th>$C'_1$</th>
<th>$C_2$</th>
<th>$C_3$</th>
<th>$C_4$</th>
<th>$f_w$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.7 + 0.9P/\varepsilon</td>
<td>-1.05</td>
<td>0.8 - 0.65(a_{ij}a_{ij})^{0.5}</td>
<td>0.625</td>
<td>0.2</td>
<td>$exp(-(0.0184 \sqrt{\nu y})^4)$</td>
</tr>
</tbody>
</table>

Table 3.1: Coefficients of the SSG-Chen Low Reynolds model

For the rate of turbulent kinetic energy dissipation $\varepsilon$ the transport equation is:

$$\frac{\partial \varepsilon}{\partial t} + \langle U_k \rangle \frac{\partial \varepsilon}{\partial x_k} = C_{\varepsilon 1} (1 + C_{\varepsilon 4} f_w) \frac{P_k \varepsilon}{k} - C_{\varepsilon 2} f_\varepsilon \varepsilon^* \varepsilon + \nabla \cdot \left(\nu \delta_{ij} + C_{\varepsilon} \frac{k}{\varepsilon} \langle u'_i u'_j \rangle \frac{\partial \varepsilon}{\partial x_j} \right)$$

(3.6)

where,

$$f_\varepsilon = 1 - \frac{2}{9} \exp\{- (R_t/6)^4\}$$

(3.7)

$$R_t = \frac{k^2}{\nu \varepsilon}$$

(3.8)

$$\varepsilon^* = \varepsilon - 2 \frac{\nu k}{y^2}$$

(3.9)

$$\xi = \left[(-2 + \frac{7}{9} C_{\varepsilon 2}) \frac{\varepsilon^* \varepsilon}{k} - \frac{\varepsilon^*}{2k}\right] f_w$$

(3.10)

The values of the coefficients used in eq. (3.6) are presented in Table 3.2. As it can be seen from equations (3.5) to (3.10), in order to account for the wall effects, the original SSG model has to be extensively modified and the final outcome can be more stiff and complex.

For the eddy viscosity models, numerous damping functions have been used. The Launder-Sharma [66] $k - \varepsilon$ model is one of the adaptations of the standard model for low Reynolds
3.2. Damping functions

<table>
<thead>
<tr>
<th>$C_\epsilon$</th>
<th>$C_{\epsilon 1}$</th>
<th>$C_{\epsilon 2}$</th>
<th>$C_{\epsilon 4}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.15</td>
<td>1.35</td>
<td>1.8</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Table 3.2: Coefficients of the SSG-Chen Low Reynolds model for the dissipation rate numbers. In this model, the dissipation equation is replaced with a transport equation for a new variable $\tilde{\epsilon}$ that is zero at the wall. Two empirical exponential damping functions are added to ensure the near wall behaviour which increases non-linearity. The equations of the model are:

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = P_k - (\tilde{\epsilon} + D\epsilon) + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (3.11)

$$\frac{\partial \tilde{\epsilon}}{\partial t} + \langle U_j \rangle \frac{\partial \tilde{\epsilon}}{\partial x_j} = C_{\epsilon 1} \frac{\tilde{\epsilon}}{k} P_k - C_{\epsilon 2} f_2 \frac{\tilde{\epsilon}^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\epsilon} \right) \frac{\partial \tilde{\epsilon}}{\partial x_j} \right] + 2 \nu \nu_t \left( \frac{\partial^2 \langle U_i \rangle}{\partial x_j \partial x_k} \right)$$  \hspace{1cm} (3.12)

where,

$$D\epsilon = 2\nu \left( \frac{\partial \sqrt{k}}{\partial x_j} \right)$$  \hspace{1cm} (3.13)

$$\tilde{\epsilon} = \epsilon - D\epsilon$$  \hspace{1cm} (3.14)

$$Re_t = \frac{k^2}{\nu \epsilon}$$  \hspace{1cm} (3.15)

$$\nu_t = C_\mu f_\mu \frac{k^2}{\epsilon}$$  \hspace{1cm} (3.16)

and

$$f_\mu = \exp \left( \frac{-3.4}{\left( 1 + \frac{Re_t}{50} \right)^2} \right)$$  \hspace{1cm} (3.17)

$$f_2 = 1 - 0.3 \exp \left( -Re_t^2 \right)$$  \hspace{1cm} (3.18)

$$f_\mu = \exp \left( \frac{-3.4}{\left( 1 + \frac{Re_t}{50} \right)^2} \right)$$  \hspace{1cm} (3.19)

For this model, the same coefficients shown in Table 2.3 are used. Since the model resolves the turbulent equations in the near wall region, it is necessary to prescribe boundary conditions. At the wall, a low-Reynolds model should guarantee an asymptotic behaviour like:

$$k \sim y^2 \quad \text{and} \quad \epsilon \to \frac{2\nu k}{y^2} \quad \text{as} \quad y \to 0$$  \hspace{1cm} (3.20)
The use of $\tilde{\varepsilon} = 0$ at the wall ensures the correct behaviour of the dissipation $\varepsilon$. Although the modifications to the standard $k - \varepsilon$ model produce a reasonably good behaviour inside the viscous sublayer for simple test cases, its use has been moderate, since the damping functions are empirical and the non-linearities introduced makes it a more stiff model. Some aspects such as mesh refinement and proximity to the wall have to be taken strictly into account for a computation with this model. Other models have been proposed to account for the near wall effects based on the $k - \varepsilon$ model, more information can be found on [118]. For the $k - \omega$ model, Wilcox [118] also proposed a damping function. Although the original model can be integrated all the way down to the wall, the behaviour of the turbulent quantities in the viscous sublayer is not satisfactory. In the original model $k$ does not behave as $y^2$ near the wall. Sarkar and So [98] performed and study with ten two-equation models with damping functions in different types of flow: a Couette flow, a boundary layer, a channel flow and a backward facing step. It was found that only those models that were asymptotically consistent were able to correctly predict the spatial distribution of kinetic energy in Couette flows. It was also found that those models whose damping functions were independent on $y^+$ yielded better results. In the case of the backstep flow, the models that are dependent on $y^+$ in the viscosity damping function produce a friction coefficient distribution that is not very realistic.

Although damping functions can be devised to enforce the model to behave in a particular way, they remain empirical and do not generally take into account the source of the turbulence reduction, but rather try to mimic the effect. Therefore other approaches have been derived to consider the wall effects in a somehow more universal way.

### 3.3 Two component limit

Another approach to modelling near wall behaviour in second moment closure is the two-component limit adopted by Craft and Launder [25]. In their approach they use “normalised length scale gradients” that are defined as:

$$d_i = \frac{N_i}{0.5 + (N_k N_k)^{0.5}}$$

with:

$$N_i = \frac{\partial (k^{1.5}/\varepsilon)}{\partial x_i}$$

(3.21)

$$d_i^A = \frac{N_i^A}{0.5 + (N_k A^A)^{0.5}}$$

with:

$$N_i^A = \frac{\partial (k^{1.5} A^{0.5}/\varepsilon)}{\partial x_i}$$

(3.22)
3.4 Elliptic relaxation

These parameters have a value close to unity in a region with strong inhomogeneity and become small near the homogeneous regions thus providing an indication of the wall proximity. $A$ is Lumley’s flatness parameter defined by:

$$A = 1 - \frac{9}{8} (a_{ij} a_{ij} - a_{ij} a_{jk} a_{ki})$$  \hspace{1cm} (3.23)

The model uses the previous definitions in the modelling of the dissipation, pressure correlations, turbulent diffusion and dissipation rate to insure the two component limit near the wall. The model has been tested in separated flows configurations such as the backward facing step and the asymmetric plane diffuser with promising results [23].

3.4 Elliptic relaxation

Another way of taking into account the wall effects in the pressure-strain correlation term was proposed by Durbin [33]. It is a different approach since instead of modelling the pressure strain term, it attempts to model the two point correlation in equation (3.3). In this way the non-local effects can be accounted for and there is no need to use the ad hoc damping functions that are not universal.

Equation (3.3) can be used to obtain the velocity-pressure gradient correlation term in the pressure-strain term:

$$\left< u'_i \frac{\partial p'(x)}{\partial x_j} \right> = \frac{1}{4\pi} \int \int \int_{-\infty}^{\infty} \frac{\left< u'_i((x)\partial S(x')/\partial x_j) \right>}{|x - x'|} d^3x'$$  \hspace{1cm} (3.24)

with

$$S(x') = \rho \frac{\partial \langle U_k \rangle}{\partial x_l} \frac{\partial u'_l}{\partial x_k}$$  \hspace{1cm} (3.25)

in the elliptic relaxation approach, the two point correlation is modelled as:

$$\left< u'_i(x)\partial S(x')/\partial x_j \right> = \left< u'_i(x')\partial S(x')/\partial x_j \right> \exp \left(-|x - x'|/L\right)$$  \hspace{1cm} (3.26)

The kernel of the integral becomes Green’s function for the modified Helmotz form of the equation, which leads to:

$$\phi_{ij} - L^2 \nabla^2 \phi_{ij} = \phi^b_{ij}$$  \hspace{1cm} (3.27)
Here, $\phi_{ij}^h$ is the homogeneous model for the pressure strain term. Durbin actually considers the redistribution tensor term written as:

$$\rho_{ij} = \phi_{ij} + \left( \varepsilon_{ij} - \frac{\langle u_i'u_j' \rangle}{k} \varepsilon \right)$$

(3.28)

to make the new term vanish at no-slip boundaries (in order to be able to model the asymptotic behaviour correctly) [37]. With the use of the intermediate variable $f_{ij} = \phi_{ij}/k$ the elliptic equation can be written as:

$$f_{ij} - L^2 \nabla^2 f_{ij} = \frac{\phi_{ij}^h - \varepsilon b_{ij}}{k}$$

(3.29)

Durbin [37] used the sum of Rotta’s return to isotropy and the isotropisation of production model:

$$\phi_{ij}^h = \frac{1 - C_1}{kT} \left( \langle u_i'u_j' \rangle - \frac{2}{3} k \delta_{ij} \right) - \frac{C_2}{k} \left( P_{ij} - \frac{2}{3} P \delta_{ij} \right)$$

(3.30)

where $k = \frac{1}{2} \langle u_i'u_i' \rangle$, $P = \frac{1}{2} P_{ii}$ and the time and length scales are given by:

$$T = \max \left( \frac{\nu}{\varepsilon}, C_T \sqrt{\frac{\nu}{\varepsilon}} \right)$$

(3.31)

$$L = C_L \max \left( \frac{\nu^3}{\varepsilon}, C_\eta \left( \frac{\nu^3}{\varepsilon} \right)^{\frac{1}{4}} \right)$$

(3.32)

Here the Kolmogorov timescale is used in the viscous sublayer to prevent singularities near the wall. The large value of the length scale bound ($C_\eta = 80$) has been justified by the fact that the two point correlation length scale does not go to zero at the wall [77]. Other pressure-strain models have been used with elliptic relaxation, such as the SSG, and the Craft and Launder cubic models, but the effect of the model used is not as important as the use of the elliptic relaxation [119].

For the dissipation, Durbin adopted the standard model of Patel et al. [37]

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{C_{\varepsilon 1} P_k - C_{\varepsilon 2} \varepsilon}{T} + \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right)$$

(3.33)

with $\nu_T = C_\mu \langle u_i'u_j' \rangle T$ and $C_{\varepsilon 1}^* = C_{\varepsilon 1}(1 + A_1 P/\varepsilon)$. The constants used in this model are shown in table [3.3]. The elliptic relaxation method has been proved to be accurate in some cases such as the rotating square duct [92], but the computational cost is increased due to
3.4. Elliptic relaxation

<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 3.3: Coefficients of the Elliptic Relaxation model

the fact that there are 6 more equations to solve ($f_{ij}$). In addition, it has the disadvantage of having numerical difficulties due to the stiffness of the boundary conditions on $f$ which depends on $y^4$:

$$
\begin{align*}
  f_{22} &= -\frac{20\nu^2 \langle v'v' \rangle}{\varepsilon^2 y^4}, & f_{12} &= -\frac{20\nu^2 \langle u'v' \rangle}{\varepsilon^2 y^4}, & f_{23} &= -\frac{20\nu^2 \langle v'w' \rangle}{\varepsilon^2 y^4} \\
  f_{11} &= -\frac{1}{2} f_{22}, & f_{33} &= -\frac{1}{2} f_{22}, & f_{13} &= -\frac{20\nu^2 \langle u'w' \rangle}{\varepsilon^2 y^4}
\end{align*}
(3.34)

Elliptic relaxation is not the best method available but it has interesting features that provide a deeper understanding of the near-wall physics, by allowing the boundary conditions to enter at the wall and relax naturally into the flow far from it. In a local RSM only one boundary condition can be imposed ($\langle u'_{i} u'_{j} \rangle = 0$) and therefore the asymptotic behaviour has to be determined from the model. Using elliptic relaxation, boundary conditions can be applied to both $\langle u'_{i} u'_{j} \rangle$ and $f_{ij}$ to achieve the same goal.

A simplification of this method has been proposed by Manceau and Hanjalić [75] called the *Elliptic Blending Model* which has been developed to use only one equation for all the $f_{ij}$ and to have a value of zero as a boundary conditions at the wall. The model is based on the assumption that since the elliptic form is based on the length scale it should be possible to have a similar equation that can represent all the processes for all the stresses. The pressure strain correlation is modelled as:

$$
\phi_{ij} = (1 - k\alpha)\phi_{ij}^w + k\alpha\phi_{ij}^h
(3.35)
$$

and the dissipation as:

$$
\varepsilon_{ij} = (1 - Ak\alpha) \frac{\langle u'_{i} u'_{j} \rangle}{k} \varepsilon + Ak\alpha \frac{2}{3} \varepsilon_{ij}
(3.36)
$$

solving an elliptic relaxation equation for $\alpha$:

$$
\alpha - L^2 \nabla^2 \alpha = \frac{1}{\varepsilon T}
(3.37)
$$
A is Lumley’s flatness parameter,

\[ A = 1 - \frac{9}{8} (a_{ij} a_{ij} - a_{ij} a_{jk} a_{ki}) \]  

(3.38)

Here the far-from-the-wall part, \( \phi^h_{ij} \) is calculated with the standard high Reynolds SSG model, and the near wall part, \( \phi^w_{ij} \) is calculated as:

\[ \phi^w_{ij} = -\frac{5}{k} \varepsilon \left( \langle u_i' u_k' \rangle n_j n_k + \langle u_j' u_k' \rangle n_i n_k - \frac{1}{2} \langle u_k' u_l' \rangle n_k n_l (n_i n_j - \delta_{ij}) \right) \]  

(3.39)

were \( n \) is a unit vector representing the wall-normal direction everywhere inside the domain, calculated as:

\[ n = \frac{\nabla \alpha}{\|\nabla \alpha\|} \]  

(3.40)

This formulation ensures that the near wall behaviour is correctly represented. The model takes advantage of the elliptic relaxation model with only one equation instead of six, and it can be seen as a compromise of simplicity and consistency with the physics.

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

Although the \( k - \omega \) model can be solved all the way down to the wall, it does not give a quantitatively good prediction of turbulence variables in the viscous sublayer. This is due mainly to the two-equations model not being designed to account for the wall echo effects and not preserving the two component limit near solid boundaries. The \( \overline{v^2} - f \) model is designed to obtain a measure of non-local effects via elliptic relaxation. In order to account for non-local effects in the framework of an eddy viscosity model, the elliptic relaxation approach described before is simplified in the case of a plane channel flow. Taking into account that the appropriate velocity scale near the wall is \( \overline{v^2} \) [33], the turbulent viscosity can be expressed as:

\[ \nu_t = C_{nu} \overline{v^2} T \]  

(3.41)

where \( T \) is given by (3.31). In order to obtain a transport equation for \( \overline{v^2} \) a simplification is applied to the equations of the wall normal components (\( \langle u_2' u_2' \rangle \) and \( f_{22} \)) in the full elliptic
3.4. Elliptic relaxation

relaxation second moment closure. The model equations are:

\[
\frac{\partial \overline{v^2}}{\partial t} + \langle U_j \rangle \frac{\partial \overline{v^2}}{\partial x_j} = k f - \overline{v^2} \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \overline{v^2}}{\partial x_j} \right]
\]

(3.42)

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

(3.43)

In two dimensional steady flows \( \overline{v^2} \) can be seen as a scalar representing the velocity fluctuation normal to the streamlines, since it is no longer a component of the tensor \( \langle u'_i u'_j \rangle \) [34]. A more general view of \( \overline{v^2} \) is that it represents a velocity scale that satisfies the boundary conditions suitable for the wall normal component of turbulence intensity. In this way the standard \( k - \varepsilon \) model can be used without the need of any damping functions. The model solves the transport equations for \( k \) and \( \varepsilon \) with the introduction of the bounded timescale:

\[
\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]
\]

(3.44)

\[
\frac{\partial \varepsilon}{\partial t} + \langle U_j \rangle \frac{\partial \varepsilon}{\partial x_j} = \frac{C_{\varepsilon 1} P_k - C_{\varepsilon 2} \varepsilon}{T} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]
\]

(3.45)

The \( \varepsilon \) equation was developed from a homogeneous model and certain terms that can be important near the wall have not been taken into account [33]. This involves the boundary condition at the wall. To avoid imposing a value \( \varepsilon \) at the wall in the first proposals of the model, (Durbin [33], [36], [34]) solved the equations in a coupled manner, that is, \( k - \varepsilon \) in a first step and then \( \overline{v^2} - f \) in the second step. This way of solving makes it possible to avoid using boundary conditions for \( \varepsilon \) by setting the \( k \) boundary conditions as:

\[
k = \frac{\partial k}{\partial n} = 0
\]

(3.46)

The near-wall behaviour is introduced via the elliptic equation for \( f \) with the boundary condition taken as:

\[
\overline{v^2} \rightarrow \frac{\varepsilon_w y^4 f_w}{20\nu}
\]

(3.47)

For uncoupled solvers, it is also possible to write the limiting boundary conditions for \( \varepsilon \) and \( f \) as:

\[
\varepsilon_w = \frac{2\nu k}{y^2} \quad \text{and} \quad f_w = -\frac{20\nu \overline{v^2}}{\varepsilon y^4},
\]

(3.48)
which shows the strong dependency on $\overline{v'^2}$, making it necessary to use a coupled solver at least for $\overline{v'^2}$ and $f$. The closure constants have been changed since the model’s original presentation \cite{33} with the value of $C_{\varepsilon 1}$ being the most important change. It is known that the value of this constant has a considerable impact on the prediction of the spreading rate of a shear layer. The constant value of $C_{\varepsilon 1} = 1.44$ was replaced by a linear dependence of the production to dissipation ratio $C_{\varepsilon 1} = 1.44(1 + A_1 P/\varepsilon)$ with a value of $A_1 = 0.1$ found adequate \cite{33}. In this way the coefficient can represent the production due to local anisotropy, but a non-linear dependency would be necessary to accommodate larger values of $P/\varepsilon$. It has been found that this dependency can lead to numerical difficulties and prevent convergence to a steady solution for complex flows \cite{36}. The value of $C_{\varepsilon 1} = 1.55$ has been found suitable for wall bounded flows but a lower value of $C_{\varepsilon 1} = 1.3$ is needed to predict the correct spreading rate for plane mixing layers \cite{34,36}. A more pragmatical approach was proposed by Durbin \cite{34} in which an interpolation function was devised in the form of:

$$C_{\varepsilon 1} = 1.3 + \frac{0.25}{[1 + (y/2L)^2]^4} \tag{3.49}$$

where $y$ is the distance to the nearest wall and $L$ is given by eqn (3.32). For a near-wall model, the use of a parameter such as the distance from the wall is not convenient since it can be ill-defined in complex 3D geometries and is not practical to use in multi-zone computations, therefore another way of calculating $C_{\varepsilon 1}$ is required. The use of the structural parameter $k/\overline{v'^2}$ was proposed by Parneix et al \cite{89}:

$$C_{\varepsilon 1} = 1.4 \left( 1 + A_1 \left( \frac{k}{\overline{v'^2}} \right) \right) \tag{3.50}$$

This formulation was used in the same test cases as the equation (3.49): a zero pressure gradient 2D turbulent boundary layer, a low Reynolds number fully developed channel flow and a swept bump, with only minor differences in the results \cite{89}. The closure coefficients vary according to the equation used for $C_{\varepsilon 1}$. In Table 3.4 the coefficients for the model using equation (3.50) are listed taken from Durbin and Pettersson-Reif \cite{38}.

The $\overline{v'^2} - f$ model has been tested and calibrated in fully developed channel flows at various Reynolds numbers \cite{36} showing the correct behaviour of mean and turbulent quantities in the near wall region, it also showed no dependency on free stream values in the calculation of zero and adverse pressure gradient boundary layer. The model has also been tested
### 3.4. Elliptic relaxation

<table>
<thead>
<tr>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_\mu$</th>
<th>$\sigma_\varepsilon$</th>
<th>$\sigma_k$</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$C_L$</th>
<th>$C_\eta$</th>
<th>$A_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eq. (3.50)</td>
<td>1.9</td>
<td>0.19</td>
<td>1.3</td>
<td>1.0</td>
<td>1.4</td>
<td>0.3</td>
<td>0.3</td>
<td>70</td>
<td>0.045</td>
</tr>
</tbody>
</table>

Table 3.4: Coefficients of the $\overline{u'^2} - f$ original model

in separated flows such as flow over backward facing step [34] where it predicted a good recirculation length at low Reynolds number ($Re = 5000$), but at a higher Reynolds number ($Re = 37500$) the reattachment was predicted earlier than in the experiments. The asymmetric plane diffuser was also computed in [34] and the influence of equation (3.49) was presented showing an improvement in the velocity profiles in the expansion section. The vortex shedding behind a triangular cylinder was also studied in unsteady mode with a good agreement on the Strouhal number. The time-averaged velocity profiles showed a good agreement with experimental data but no comparison was made with other turbulence models.

A comparative study of two versions of the $\overline{u'^2} - f$ model was made by Parneix et al. [89] for a flow over a swept bump. They compared the effects of using equation (3.50) instead of (3.49). Qualitatively, the pressure coefficient predictions were very similar for both formulations showing a slow recovery after the bump with the second being closer to the experimental data. In terms of main velocities, the predictions were in good agreement with the experimental data and the difference between the two formulations was difficult to see. The most notable discrepancy is in the level of the stream-wise skin friction coefficient over the bump. Both models over predict the experimental level but the use of equation (3.50) results in a closer match to the experiments. They also tested the model (with eq. (3.50)) in a 3D flow around an appendage-body junction (an elliptical nose and a NACA020 tail joined at the maximum thickness) which is a difficult case involving separation and a horseshoe vortex. The model gave qualitatively good results, predicting all these phenomena. The horseshoe vortex was reproduced accurately both in terms of its location and intensity although the secondary flow was slightly underestimated. The model also predicted the correct damping of turbulence near the wall, specially in the impingement regions where other models such as the $k - \varepsilon$ are known to predict a spurious production of turbulence.

The model has also been tested in heat transfer problems and to avoid the overproduction of kinetic energy in impingement regions, Durbin proposed a realisability constraint on the
time scale [35]. Based on the assumption that if the timescale becomes very large in equation (3.45), the production of dissipation is reduced and an overprediction of $k$ is obtained.

The Boussinesq approximation (2.27) gives an erroneous normal stress difference. If $k = \frac{1}{2} \langle u'_i u'_j \rangle$ then $2k \geq \overline{v^2} \geq 0$, using the eigenvalues $\lambda_\alpha$ of the strain rate tensor $S_{ij}$, (2.27) can be written as:

$$\overline{u'_\alpha^2} = -2\nu_t \lambda_\alpha + \frac{2}{3}k$$

(3.51)

and by imposing the more stringent constraint $\overline{u^2} \geq 0$ it is possible to obtain:

$$2\nu_t \lambda_\alpha \leq \frac{2}{3}k$$

(3.52)

Solving the characteristic equation for $\lambda_\alpha$, for three dimensions:

$$|\lambda_\alpha| \leq \sqrt{2 |S|^2 / 3}$$

(3.53)

using equations (3.52) and (3.53) the realisable limit on $\nu_t$ can be obtained:

$$\nu_t \leq \frac{k}{3 \max \lambda_\alpha}$$

(3.54)

by using the definition of the turbulent viscosity on the model (3.41) the time scale bound can be derived as:

$$T = \min \left[ \max \left( \frac{k}{\varepsilon}, C_\varepsilon \sqrt{\frac{\nu}{\varepsilon}} \right), \frac{0.6k}{\sqrt{6}C_\mu \nu^2 S} \right]$$

(3.55)

It is not clear why Durbin uses the constraint on all the equations containing the time scale instead of using the constraint only on the viscosity equation, where it comes from. The necessity to limit the time scale (according to Durbin [35]) comes from the $\varepsilon$ equation, where a large value of $T$ would lead to a low value of the production term in the dissipation equation and hence high levels of turbulent kinetic energy. As pointed by Sveningsson and Davidson [107], by taking a closer look at the first two terms in equation (3.45), using equations (3.41) and (2.33), they can be written as:

$$\frac{C_\varepsilon P_k - C_{\varepsilon 2} \varepsilon}{T} = \frac{C_\varepsilon 2\nu_t S_{ij} S_{ij} - C_{\varepsilon 2} \varepsilon}{T} = 2C_\varepsilon 1 C_\mu \overline{v^2} S_{ij} S_{ij} - C_{\varepsilon 2} \varepsilon$$

(3.56)

which clearly shows that the production term is not affected by the time scale, the only effect of an increase in the time scale is to lower the dissipation of $\varepsilon$. The real effect of the constraint is in the definitions of the production, i.e. equation (2.33) which directly affects the level of
turbulent kinetic energy. This makes it difficult to see why the realizability constraint should be applied everywhere $T$ is present instead of using it only in the calculation of the production of turbulent kinetic energy [107]. This modification has been proven to be successful on various test cases but it has been reported to be somewhat difficult to implement [106]. A similar bound is used on the length scale given by:

$$L = C_L \max \left[ \min \left( \frac{k^{3/2}}{\varepsilon}, \sqrt[3]{6C_\mu \nu^2 S} \right), C_\eta \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \right]$$

(3.57)

The issue of the overprediction of turbulent kinetic energy was also addressed by Guimet and Laurence [46] with a linear production model proposed. Since the production term is proportional to the rate of strain (see eq.(2.12)) it can be written as:

$$P_{exact} = -kS_{ij}a_{ij}$$

(3.58)

But the modelled production term is proportional to the square of the strain rate, recall eq (2.33). The linear production model defines the production as:

$$P = [a_{12}]_{log}(k \cdot S)$$

(3.59)

$$[a_{12}]_{log} = C_{\mu}^{1/2} \min(1, C_{\mu}^{1/2} \eta)$$

(3.60)

where $\eta = \tau \cdot S$ is the strain parameter and can be solved by an ODE described in [46].

Menter [80] also proposed a limiter on the production in the form of

$$\hat{P} = \min(P, 20\varepsilon)$$

(3.61)

Another modification to the production term was proposed much earlier by Kato and Launder [64] in which the quadratic strain contribution is replaced by $S_{ij} \Omega_{ij}$.

The $u^2 - f$ model with the realizability constraint has been applied to heat transfer problems such as confined and unconfined impingement jets [12, 13] at various Reynolds numbers and different jet-to-target distances. The results proved to be closer to experiments when compared with the standard $k - \varepsilon$ model and encourage the use of the model in different cases involving heat transfer. Ooi et al. [87] used the model in a flow in a ribbed channel it was shown that the model has a superior performance compared to the $k - \varepsilon$ two-layer model. Here the separation of the flow plays a very important role in the heat transfer calculations.
and since the model was proved to predict a good behaviour on impinging flows and on cases with separated flow, this case was expected to show the model capabilities. The results were compared with experimental values and it was shown that both models predicted the correct flow behaviour but the heat transfer is predicted correctly only by the $\overline{v^2} - f$ model. A similar case was computed by Manceau et al. \[76\] with and without heat transfer and the model implemented in a industrial finite element code based on unstructured meshes, showing that the results are not code-dependent even if the numerical methods used are completely different, which is not always the case for all turbulence models.

The good results produced by the $\overline{v^2} - f$ model for heat transfer were the basis for its use on turbine design test cases. Kalitzin and Iaccarino \[62\] tested the model in a turbine cascade configuration in two and three dimensions. This is a difficult case since there are phenomena like transition, separation and heat transfer. The results presented by the authors were compared with the Spalart-Allmaras model, but none of the models gave a good prediction in terms of heat transfer. Issues such as grid refinement, numerical procedure and inlet profiles were identified as being responsible for the differences between numerical predictions and experimental data.

**Code Friendly variation**

In order to avoid the strong coupling of $\overline{v^2}$ and $f$ at the boundaries, a modification was suggested by Lien and Durbin \[70\]. Recall the definition of $f$:

$$ kf = \phi_{22} - \varepsilon_{22} + \frac{\overline{v^2}}{k} \varepsilon $$

(3.62)

The asymptotic behaviour of $\phi_{22}$ and $\varepsilon_{22}$ are given by:

$$ \phi_{22} = -2 \frac{\overline{v^2}}{k} \varepsilon \quad \text{and} \quad \varepsilon_{22} = 4 \frac{\overline{v^2}}{k} \varepsilon $$

(3.63)

So $kf$ tends to

$$ kf \rightarrow -5 \frac{\overline{v^2}}{k} \varepsilon $$

(3.64)

By introducing a change in variable that tends to zero at the wall, the Lien and Durbin model \[70\] (referred herein as LDM) can be obtained by using:

$$ \overline{f} = f + 5 \frac{\overline{v^2}}{k^2} \varepsilon $$

(3.65)
Equation (3.65) can be inserted into equations (3.42) and (3.43) to obtain:

$$\frac{\partial \bar{v}^2}{\partial t} + \langle U_j \rangle \frac{\partial \bar{v}^2}{\partial x_j} = k \bar{f} - 6 \bar{v}^2 \varepsilon \left( \frac{\nu}{\sigma_k} \right) \frac{\partial \bar{v}^2}{\partial x_j}$$

(3.66)

$$L^2 \frac{\partial^2 \bar{f}}{\partial x_j^2} - \bar{f} = \frac{1}{T} \left[ (C_1 - 6) \frac{\bar{v}^2}{k} - \frac{2}{3} (C_1 - 1) \right] - C_2 \frac{P_k}{k}$$

(3.67)

With this modification the boundary condition for \( \bar{f} \) at the wall is zero. In transforming the equation from the original model into the LDM, there is a term that is neglected:

$$-5L^2 \frac{\partial^2}{\partial x_j^2} \left( \frac{\bar{v}^2}{k^2 \varepsilon} \right)$$

(3.68)

With the usual log-layer assumptions for a channel flow (\( \bar{v}^2 \) and \( k \) constant, \( L \sim y, \varepsilon \sim 1/y \)) this term is proportional to \( 1/y \), hence just as large as the other terms in the \( f \) equation. The form of the elliptic operator is exactly derived from the Poisson equation only when considering the pressure strain term as a primitive variable. Subsequently Durbin divided this quantity by the kinetic energy as a convenience for obtaining the desired near-wall asymptotic behaviour. From there on it makes no difference, from the theoretical point of view, whether \( f \) or \( \bar{f} \) is inserted in the elliptic operator. In other words, here the expression neglected term means neglected in comparison to Durbins original model, but not in terms of an exact equation. The fact that this term is not taken into account means that the equation for \( \bar{f} \) does not necessarily have the same properties as the original one. It simply is a different way of modelling the source term in the \( \bar{v}^2 \) equation, but it should be considered as a different model.

As it will be shown later, the term given by equation (3.68) is important in the near-wall region up to a distance of \( y^+ = 100 \) for a channel flow. The model coefficients according to Lien and Kalitzin [69] are shown in Table 3.5. As the \( \bar{v}^2 - f \) model (and in particular the code friendly version) is fairly recent, the constants are being optimised over time. Chronologically, the first paper that introduces the Code Friendly modification is by Lien and Durbin in

<table>
<thead>
<tr>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
<th>( C_\mu )</th>
<th>( \sigma_\varepsilon )</th>
<th>( \sigma_k )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_L )</th>
<th>( C_\eta )</th>
<th>( A_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eq. (3.50)</td>
<td>1.9</td>
<td>0.22</td>
<td>1.3</td>
<td>1.0</td>
<td>1.4</td>
<td>0.3</td>
<td>0.23</td>
<td>70</td>
<td>0.05</td>
</tr>
</tbody>
</table>

Table 3.5: Coefficients of the LDM
Coefficients were later modified in Lien et al. [71], then again in 1999 by Kalitzin [61]. The constants used by Wu and Durbin [120] are again slightly modified. Finally, the set of constants given in Lien and Kalitzin [69] is the one used herein.

This model has been tested and calibrated in fully developed channel flows and by-pass transitional flow over a flat plate [70]. The model variant was also tested by Lien et al. [71] with slightly different coefficients on a double circular arc compressor blade, where the influence of the time scale realisability constraint was also studied. The two versions of the model were reported to have superior predictive capabilities in the test cases when compared to $k - \varepsilon$ models, and a slight difference was encountered between the two formulations. A more complete and recent comparison between the two formulations was made by Lien and Kalitzin [69] in a Derely bump case and a RAE 2822 transonic aerofoil. Although the models returned qualitatively similar results, a difference was obtained in terms of separation and shear stress predictions. As a general conclusion, the LDM version of the model tends to predict a higher level of wall shear stress before the separation region, which locates the separation point further downstream. Within the separation bubble, the LDM predicts a lower level of shear stress and hence a lower rate of fluid entrainment making the bubble thicker. The original model was reported to predict a faster recovery after the reattachment. Kalitzin [61] made a comparison of different aerofoil configurations using the original model with equations (3.49) and (3.50), the LDM model and the SST model. It was found that the use of the distance from the wall in the definition of $C_{\varepsilon_1}$ i.e. eq. (3.49) gave the best predictions compared to experiments. The use of equation (3.50) resulted in an over-prediction of pressure on the suction surface of the flap, also over-predicting the skin friction coefficients over the largest part of the wing. It is the same case for the results of the LDM model but in addition the larger skin friction prediction gives rise to a larger velocity gradient at the wall and a change in the velocity profile in that region.

Despite the poorer performance reported on the LDM model, it has been proved to be suitable for commercial codes, as reported by Iaccarino [57] where the model was implemented in three different commercial codes: CFX, Fluent and Star-CD. The case studied was the 2D asymmetric plane diffuser for which two sets of experimental data are available as well as a LES calculation. All the codes have the $k - \varepsilon$ Launder-Sharma model build in, but
the $\overline{v^2} - f$ was implemented by the author via user defined subroutines. Although the results for the $k - \varepsilon$ model were code dependent, the $\overline{v^2} - f$ model performance was very similar for all the codes with Fluent being slightly more dissipative.
Chapter 4

The $\varphi - f$ model

4.1 Introduction

As seen in Section 3.4, the $v^2 - f$ model was developed to include the elliptic relaxation method into an eddy viscosity model, but its stiffness makes it difficult to implement in segregated solvers.

As well as the boundary condition for $\varepsilon$, the boundary condition for $f$ (eq. (3.48)) is represented by a removable singularity. Although the limit exists, it is not \textit{a priori} known. It results in substantial difficulties in the numerical solution.

In the case of a segregated solver when the $f$ equation is solved separately, the boundary value problem for this equation becomes ill-conditioned due to the asymptotic $y^4$ behaviour at the wall. This means that small perturbations of input data (either the numerator or denominator) can lead to pronounced changes in the solution. In an exact solution both the numerator and denominator asymptotically have the same order but in the numerical solution procedure they can have different orders which causes substantial numerical problems. A similar problem takes place for $\varepsilon$ but in the case of the $f$ equation the problem is more severe. Numerically it leads to oscillations or divergence of the solution that are mainly encountered in segregated numerical procedures for solving the $v^2$ and $f$ equations. In many industrial codes this is not desirable because it increases computational needs and coding effort. This boundary condition has also been reported to be a source of numerical problems in transitional flows \cite{70,71,69}. This problem has kept the use of the original $v^2 - f$ model away
from commercial use.

## 4.2 Model derivation

In order to make the $\bar{v}^2 - f$ model more adaptable to an industrial code but without sacrificing its performance, another approach has been followed at the University of Manchester during the course of this work (See: [68]). By introducing a new variable, $\varphi$, defined as:

$$\varphi = \frac{\bar{v}^2}{k}$$

(4.1)

a transport equation can be solved for this new variable, which introduces some advantages. The resulting equations for $\varphi$ and $f$ are:

$$\frac{\partial \varphi}{\partial t} + \langle U_j \rangle \frac{\partial \varphi}{\partial x_j} = f - P \varphi \frac{2}{k} \left( \nu + \frac{\nu_k}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j} \left[ \left( \nu + \frac{\nu_k}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j} \right] + \left( \nu + \frac{\nu_k}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j}$$

(4.2)

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

(4.3)

In isotropic flow $\varphi \rightarrow 2/3$. A clear benefit is in the boundary condition for $f$:

$$\lim_{y \to 0} f = -5 \varepsilon \frac{\bar{v}^2}{k^2} = -5 \varepsilon \frac{\varphi}{k}$$

(4.4)

Although the resulting model has a boundary condition less stiff than the original one, it is non-zero at the wall but resulting from the ratio of three unknown variables, therefore still diminishing the robustness of the model. An interesting benefit is that the term $\varepsilon \bar{v}^2 / k$ is no longer in the transport equation, which can be difficult to reproduce correctly in the near wall region since $\varepsilon$ becomes large and the ratio $\bar{v}^2 / k$ tends to zero.

## 4.3 Boundary conditions

By using the limiting value of $\varepsilon$ near the wall and applying the L’Hôpital theorem, the value of $f$ can be written in terms of $\varphi$:

$$\lim_{y \to 0} \varepsilon = 2 \nu \frac{k}{y^2}$$

(4.5)

$$\lim_{y \to 0} f = -10 \nu \frac{\varphi}{y^2} = -5 \nu \frac{\partial^2 \varphi}{\partial x_j^2} = -5 \nu \Delta \varphi$$

(4.6)
4.3. Boundary conditions

The singularity near the wall is now second order only (ratio of two discretised variables with a \( y^2 \) limit instead of \( y^4 \)). It is essential because the lower the order the less the "stiffness" of the boundary condition and the less the sensitivity of the boundary condition to truncation error. By reformulating the limit of \( f \) at the wall, it is possible to suggest a new change in variable that will lead to the zero boundary condition:

\[
f = \tilde{f} - \frac{2\nu(\nabla \varphi \nabla k)}{k} - \nu \nabla^2 \varphi \quad (4.7)
\]

Considering the limit \( y \to 0 \), it is possible to show that \( \tilde{f} \to 0 \). There are different possible substitutions to reach the homogeneous boundary condition for \( \tilde{f} \). Some of them, e.g. \( f = \tilde{f} - 5\nu \nabla^2 \varphi \), can lead to an ill-posed problem for \( \varphi \) since in the right-hand side we obtain a negative diffusion term: \(-4\nu \nabla^2 \varphi\). Introducing the definition for \( \tilde{f} \), the equations of the model can be found to be:

\[
\frac{D\varphi}{Dt} = \tilde{f} - P_k \frac{\varphi}{k} + \frac{2}{k} \left( \frac{\nu_t}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j} \frac{\partial k}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \varphi}{\partial x_j} \right] \quad (4.8)
\]

\[
L^2 \frac{\partial^2 \tilde{f}}{\partial x_j^2} - \tilde{f} = \frac{1}{T} (C_1 - 1) \left[ \varphi - \frac{2}{3} \right] - C_2 \frac{P_k}{k} - 2\nu \frac{\partial \varphi}{\partial x_j} \frac{\partial k}{\partial x_j} - \nu \frac{\partial^2 \varphi}{\partial x_j^2} \quad (4.9)
\]

In a similar manner as with the LDM model described in Section 3.4, in the equation for \( \tilde{f} \) the term

\[
\nu L^2 \nabla^2 \left[ \frac{2(\nabla \varphi \nabla k)}{k} + \nabla^2 \varphi \right] \quad (4.10)
\]

has been neglected from the introduction of the change of variable \( 4.7 \) in the elliptic operator. It is difficult to estimate and compare the influence of the neglected terms, both code-friendly models (LDM and \( \varphi - f \)), analytically in a general case. At least it is easy to see that in the isotropic flow the term neglected in the \( \varphi \)-model equals zero while in the LDM model it is \( 10/3 L^2 \nabla^2 (\varepsilon/k) \). The boundary conditions at the wall are as follows:

\[ k(0) = 0, \quad \varepsilon(0) \to \frac{2\nu k}{y^2}, \]

\[ \varphi(0) = 0, \quad \tilde{f}(0) = 0. \]

The boundary conditions for both \( \tilde{f} \) and \( \varphi \) are zero in the wall, which makes it possible to solve the system uncoupled.
4.4 Closure constants

The term $C_{\varepsilon 1}$ is changed to:

$$C_{\varepsilon 1} = 1.4 \left( 1 + A_1 \sqrt{\frac{1}{\varphi}} \right)$$  \hspace{1cm} (4.11)

Due to the introduction of the approximation in equation (4.7), the coefficients used in this formulation have been optimised using DNS data for a channel flow at the Reynolds number $Re_\tau = 395$ as a target. The results at different Reynolds numbers can be seen in Chapter 6. The proposed coefficients are shown in Table 4.1. Most of the coefficients have not been changed with the major differences appearing only in the values of $C_L$ and $C_\eta$, which control the length scale effect on the $f$ equation.

<table>
<thead>
<tr>
<th></th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_\mu$</th>
<th>$\sigma_\varepsilon$</th>
<th>$\sigma_k$</th>
<th>$C1$</th>
<th>$C2$</th>
<th>$C_L$</th>
<th>$C_\eta$</th>
<th>$A_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eq. (4.11)</td>
<td>1.9</td>
<td>0.22</td>
<td>1.3</td>
<td>1.0</td>
<td>1.4</td>
<td>0.3</td>
<td>0.3</td>
<td>100</td>
<td>0.06</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.1: Coefficients of the $\varphi$ model

4.5 Budgets

In Figure 4.1 the budgets of the terms in equation (4.8) are presented. The production of $\varphi$ comes from $f$, while the destruction is represented by $P_{\varphi/k}$. The diffusion term on the right hand side of (4.8) compensates for the misalignment of the maximum between the production and destruction by transporting $\varphi$ into the near-wall region. The cross-gradient term is mainly a sink term, and is only positive in the viscous sub-region. In figure 4.2 the budgets of the $\bar{f}$ equation are shown. The additional terms introduced by the change of variable (eq. 4.7) are negligible after $y^+ \approx 10$ as would be expected from a viscous term. They only act in the near wall region to bring the boundary value of $\bar{f}$ to zero.

4.6 The $\zeta - f$ model

A similar approach has been pursed by Hanjalić et al. [52] at Delf University, in which the advantages of substition (4.1) are shown and another model is derived (called $\zeta - f$, where $\zeta = \ldots$)
4.6. The $\zeta - f$ model

The work by the two groups (Manchester and Delft) was carried out independently and ultimately both groups presented their rationale for the same conference where a joint publication was concluded [51]. In the approach taken by Delft, the cross-gradient term is neglected in order to keep a simple source-sink-diffusion form and the coefficients are re-tuned to balance the terms. They also introduced the elliptic relaxation based on the quasi-linear SSG model instead of the traditional LRR model. It is interesting to note that both formulations modify the same coefficient by similar amounts, namely $C_L$, $C_\eta$ and $A_1$. 
Chapter 5

Numerical method

5.1 Introduction

In order to solve the Navier-Stokes equations a finite-volume code is used. Code_Saturne is an industrial code developed at Electricité de France (EDF) [8]. All model implementations are done using this code for all the results presented herein. Code_Saturne is the in-house code at EDF, validated and released in 2001 for problems requiring local 3D analyses with refined flow modelling. The code is capable of solving steady or transient, single-phase, incompressible, laminar or turbulent flows. The equations, written in conservative form, over control volumes with a co-located arrangement for all the variables. Structured and unstructured meshes with different types of cell shape including hybrid meshes with arbitrary interfaces can be used. Velocity-pressure coupling is obtained by a predictor-corrector scheme. Three RANS models are available, namely, the standard $k - \varepsilon$ model, the Launder Sharma model and a Second Moment Closure (LRR) model. It also contains the LES Smagorinsky and dynamic models. All the other models used in this work are implemented by the author.

The code has been used in different applications such as nuclear flow problems, process engineering, heating and ventilation, dispersion and combustion, pollutant predictions in gas turbines and thermal shock in a pressure water reactor vessel [8]. It has been developed under a quality assurance framework. The discretisation is similar to the method used in other commercial codes and is described in Ferziger and Perić [39].
5.2 The finite volume method

In this chapter, capital letters \((I, J)\) refer to the cell on either side of a face (see Figure 5.1) while the lowercase letters reference the usual tensor quantities. The method relies on dividing the domain into control volumes over which the equations are integrated. The general transport equation for a variable \(\phi\) can be written in integral form as:

\[
\int_V \frac{\partial \rho \phi}{\partial t} dV + \int_V \frac{\partial}{\partial x_j} (\rho u_j \phi) dV = \int_V \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) dV + \int_V S dV \tag{5.1}
\]

where \(S\) represents the source terms, \(V\) the volume of the cell and \(\Gamma\) the diffusion coefficient. Using Gauss’ theorem the volume integrals of the divergence can be transformed into surface integrals, which can be written as:

\[
\int_V \frac{\partial \rho \phi}{\partial t} dV + \int_A \rho \phi n_j dA = \int_A \Gamma \frac{\partial \phi}{\partial x_j} n_j dA + \int_V S dV \tag{5.2}
\]

where \(A\) is the area of the face and \(n_i\) is the face normal vector. The volume integrals are approximated by the product of the value at the cell centre \(S_I\) and the cell volume \(V\).

\[
\int_V S dV \simeq S_I V \tag{5.3}
\]

**Face integrals**

The surface integrals can be approximated by the mid-point rule, the product of the face centre value and the area of the face. In the colocated arrangement, an interpolation is required in order to obtain the values at the face centres, since all the variables are stored at the cell centres. In *Code_Saturne* the convection can be calculated either by using an upwind differencing scheme (UDS) or a central differencing scheme (CDS). The code has also a slope test based on the product of the gradients at the cell centres to dynamically switch from CDS to UDS. Using the upwind scheme, the value at the face can be obtained as (see fig 5.1):

\[
\phi_F = \phi_I \quad \text{if} \quad (U_i n_i)_F > 0 \tag{5.4}
\]

\[
= \phi_J \quad \text{if} \quad (U_i n_i)_F < 0
\]

This scheme is robust and stable but introduces additional numerical diffusion which can become large if the grid is coarse. For a centred scheme, the value at the face can be computed
as:

\[ \phi_F = \alpha \phi_I + (1 - \alpha) \phi_J + \left( \frac{1}{2} \left[ \frac{\partial \phi}{\partial x_J} \bigg|_I + \frac{\partial \phi}{\partial x_J} \bigg|_J \right] O F_j \right) \] (5.5)

with \( \alpha = \frac{F_j'}{F_j} \). The last term in equation (5.5) is added for non-orthogonal grids, where the centre of the face does not lie in the midpoint between the cell centres. The diffusion integral can be computed as:

\[ \int_A \Gamma \frac{\partial \phi}{\partial x_j} n_j dA = \sum_{Neigh} \Gamma \frac{\partial \phi}{\partial x_j} n_j A \] (5.6)

with a linear approximation for the gradient the face centre, the diffusion is computed as:

\[ \sum_{Nei} \Gamma \frac{\partial \phi}{\partial x_j} n_j A = \sum_{Neigh} \Gamma \frac{\phi_{J'} - \phi_{I'}}{IJ} n_j A \] (5.7)

The values of \( \phi_{J'} \) and \( \phi_{I'} \) can be computed by using the gradient at the cell centre:

\[ \phi_{I'} = \phi_I + \frac{\partial \phi}{\partial x_j} \bigg|_{I'} I_j \] (5.8)
Gradient reconstruction

The calculation of the gradients is achieved by an iterative solver (see [83]) in which the gradient is expressed as:

$$\frac{\partial \phi}{\partial x_j} |_I = \frac{1}{V} \int_V \frac{\partial \phi}{\partial x_j} dV = \frac{1}{V} \sum_{Neigh} \int_A \phi n_j dA$$  \hspace{1cm} (5.9)

the surface integral can be approximated using the midpoint rule so that it becomes:

$$\frac{\partial \phi}{\partial x_j} |_I = \frac{1}{V} \sum_{Neigh} \phi_F A_F n_j$$  \hspace{1cm} (5.10)

To obtain the value of $\phi_F$, a Taylor series expansion can be applied to obtain:

$$\phi_F = \phi_O + OF_j \frac{\partial \phi}{\partial x_j} |_O + O(\| OF \|^2)$$  \hspace{1cm} (5.11)

The value of $\phi_O$ can be obtained by a linear interpolation and the gradient at the same point, $\frac{\partial \phi}{\partial x_j} |_O$ from an averaged between the values of $\frac{\partial \phi}{\partial x_j} |_I$ and $\frac{\partial \phi}{\partial x_j} |_J$. Finally the system to solve can be written as:

$$\frac{\partial \phi}{\partial x_j} |_I = \frac{1}{V} \sum_{Neigh} \left\{ \alpha \phi_I + (1 - \alpha) \phi_J + \frac{1}{2} OF_j \left( \frac{\partial \phi}{\partial x_j} |_I + \frac{\partial \phi}{\partial x_j} |_J \right) \right\} A_F n_j$$  \hspace{1cm} (5.12)

5.3 Time discretisation

The time discretisation in Code_Saturne is achieved through a fractional step scheme (Euler implicit) that can be associated with the SIMPLEC method [115]. The solution algorithm consists of a prediction - correction method. In the first step the momentum equation is solved using an explicit pressure gradient from the previous time step. With $Q_i^n = \rho u_i^n$ being the momentum at timestep $n$, the system to solve at the first step of the method is:

$$\frac{Q_i^* - Q_i^n}{\Delta t} + \frac{\partial}{\partial x_j} \left( u_i^* Q_j^n - \mu \frac{\partial u_i^*}{\partial x_j} \right) = - \frac{\partial P^n}{\partial x_i} + S^*$$  \hspace{1cm} (5.13)

where $S^*$ includes all the source terms that can be made implicit or explicit, i.e. $S = A_i^n + B_i^n u_i$ and $\Delta t$ is the time step. After this prediction step, a new velocity field is obtained (denoted by (*)) which is usually not divergence free. The second step consists of calculating
5.4 Boundary conditions

the pressure gradient in order to satisfy the continuity equation. By taking the divergence of
the momentum equation, the Poisson equation for the pressure can be written as:

\[
\frac{\partial}{\partial x_j} \left( \Delta t \frac{\partial (P^{**} - P^*)}{\partial x_j} \right) = \frac{\partial Q_i^*}{\partial x_j} \quad (5.14)
\]

Rhie and Chow [97] interpolation is used to avoid checkerboard oscillations. Finally, once the
updated pressure \((P^{**})\) has been obtained, the velocity field is corrected. This is done by
neglecting convection and diffusion variations:

\[
Q_i^{**} - Q_i^* = -\Delta t \frac{\partial}{\partial x_i} (P^{**} - P^*) \quad (5.15)
\]

Finaly the velocities and pressure are updated, that is \(Q^{n+1} = Q^{**}\) and \(P^{n+1} = P^{**}\). When
a turbulence model is used, the resolution of the turbulent variables takes place after the
velocities are computed. The dependence of other variables is fully explicit so that each
equation is solved separately. For LES a second order scheme is available.

5.4 Boundary conditions

For the resolution of the discretised equations described previously, the boundary conditions
have to be prescribed for any variable \(\phi\) in:

- The convection terms, a value for the normal flux \(Q_i^* n_j \phi A\)
- The diffusion terms \(\frac{\partial}{\partial x_j} \left( \mu \frac{\partial \phi}{\partial x_j} \right)\) need boundary value for the stress \(\mu \frac{\partial \phi}{\partial x_j} n_j\)
- The source terms which include the gradient of the variable, including the gradient of
  the pressure in the momentum equation.
- The terms of the Poisson equation \((5.14)\)
- The pressure gradient in eq. \((5.15)\) when the momentum values are updated.

The values for these terms are prescribed in different ways depending on the type of boundary.
5.4.1 Inlet

At the inlet a Dirichlet condition is prescribed for all transport variables (velocity, scalars, turbulent variables ...) so the values for $\phi_{\text{inlet}}^{n+1}$ and $Q_{\text{inlet}}^{n+1}$ are prescribed by the user. A homogeneous Neumann condition (zero flux) is imposed on the pressure.

The convection term $Q_j n_j \phi$ is calculated directly from the prescribed values. For the diffusion terms, the boundary value is calculated as:

$$\Gamma \frac{\partial \phi}{\partial x_j}^{\text{inlet}} = \Gamma \phi_{\text{inlet}}^{n+1} - \phi^* I' F_j n_j$$

(5.16)

For the source terms that have the gradient of the variable, the prescribed value $\phi_{\text{inlet}}^{n+1}$ is used as a boundary value. The pressure gradient normal to the face is prescribed as zero although an extrapolation from the previously obtained results is possible. The term $(\Delta t \frac{\partial \delta P}{\partial x_j} n_j)$ is set to zero in the Poisson equation and the term on the right hand side is calculated as:

$$Q^*_j n_j = \rho_{\text{inlet}}^{n+1} u_{j(\text{inlet})}^{n+1}$$

(5.17)

In updating the values of the momentum, the required pressure gradient is extrapolated with a first order approximation in space from the cell value of $\delta P_I$ taking into account that $(\Delta t \frac{\partial \delta P}{\partial x_j} n_j)$ is zero.

5.4.2 Outlet

For the outlet, a homogeneous Neumann condition is imposed on the velocity, scalars and turbulent variables. For the pressure, a Dirichlet condition is used on $P_{\text{outlet}}^{n+1}$. The boundary values for the diffusion terms are set to zero, and for the source terms a first order approximation is used to set $\phi = \phi^*$. The Dirichlet condition for pressure provides the boundary value used for the computation of the pressure gradient. The Dirichlet condition is also used in $\delta P$ in the Poisson equation. In the momentum correction equation, the boundary value for $\delta P$ is set to zero.

5.4.3 Walls and symmetries

For the walls and the symmetry faces, a zero mass flux is imposed and both Dirichlet and Neumann conditions can be applied to scalars. For the tangential velocity, a homogeneous
Dirichlet boundary condition is used at the wall whereas at the symmetry faces a homogeneous Neumann condition is applied. The pressure gradient normal to the face is set to zero although it can also be computed via an explicit extrapolation of the value at the boundary cell.

For convection terms, the boundary value of the flux is set to zero. For diffusion terms when a Neumann condition is applied to the variable $\phi$, the value prescribed is used directly. If a Dirichlet condition is applied, the boundary value is calculated in the same way as in the inlet. If the variable has a Dirichlet boundary condition and a source term requiring the gradient of the variable, the prescribed value is used as boundary value, $\phi_b = \phi$. If a Neumann condition is used, an extrapolated value from the boundary cell is used with a first order approximation. For the velocity component normal to the wall, the boundary value is set to zero to ensure zero mass flux.

For the pressure gradient calculation, if the flux of the variable at the face is prescribed, the boundary value is:

$$\phi_b = \phi_f' + I' F \left. \frac{\partial \phi}{\partial x_j} n_j \right|_b$$

(5.18)

To solve the Poisson equation, the values of $\Delta t \frac{\partial \delta P}{\partial x_j} n_j$ and $Q^* n_j$ are set to zero, and for the momentum correction equation, the boundary value for $\delta P$ is obtained from the cell value from a first order approximation using the fact that $\Delta t \frac{\partial \delta P}{\partial x_j} n_j = 0$

The values for the boundary conditions at the wall are prescribed for all velocities and turbulent variables. The way this boundary condition is treated depends on the turbulence model used. Many turbulence models have been designed under a local equilibrium assumption and use the universal logarithmic law to bridge the viscous sublayer, hence solving for the flow outside the buffer layer. To prescribe the velocity at the wall, the total shear stress is required:

$$\tau_w = \rho u^* u_k$$

(5.19)

with $u^*$ is the friction velocity and $u_k = \sqrt{\frac{k}{C_{\mu}^{1/2}}}$. In the code, the shear stress is calculated by using the wall function approach discussed in Section 2.5. Defining the tangential velocity
at the wall as \( u_{ij}^g = u_j - u_{in}n_i n_j \) the shear stress can be calculated from:

\[
u_k = C_\mu^{-\frac{1}{4}} \frac{1}{k^2}
\]

\[
u* = \frac{1}{\kappa} \ln(y^+) + C
\]
with \( \kappa = 0.42 \) and \( C = 5.2 \).

For the turbulent variables, the boundary conditions are obtained from:

\[
\frac{\partial k}{\partial y} = 0 \tag{5.22}
\]

\[
\frac{\partial \varepsilon}{\partial y} = -\frac{u_k^3}{\kappa y^2} \tag{5.23}
\]

in the case of the \( k - \varepsilon \) model and for the second moment closure the conditions are:

\[
\frac{\partial \langle u'_i u'_j \rangle}{\partial x_j} n_j = 0 \quad \text{if} \quad i = j \tag{5.24}
\]

\[
\langle u'_1 u'_2 \rangle = \nu*/u_k \tag{5.25}
\]

\[
\langle u'_1 u'_3 \rangle = \langle u'_2 u'_3 \rangle = 0 \tag{5.26}
\]

\[
\frac{\partial \varepsilon}{\partial y} = -\frac{u_k^3}{\kappa y^2} \tag{5.27}
\]

This in a local frame where \( x_2 \) is normal to the wall.

### 5.5 Notes on the implementation of new models

Although all models implemented through this work have basically the same treatment as described in the previous section, some specific details are given below.

\( \omega \) based models

The boundary condition for \( \omega \) in the original model Wilcox \[118] is given by:

\[
\lim_{y \to 0} \omega = \frac{6\nu}{\beta y^2} \tag{5.28}
\]

This has been implemented by imposing \( y \) as the distance from the centre of the cell to the wall face centre. For the SST model, Menter \[80\] proposed to use a value ten times higher.
This is due to the fact that surface roughness can be taken into account by diminishing the value of $\omega$ at the wall, so a higher value corresponds to small roughness. The $\omega$ based models can also be used with the wall function approach, by imposing the log-law value of $\omega$ as:

$$\omega = \frac{u_r}{C_{\mu^{1/2}}} \frac{1}{2} \mu \kappa y$$

(5.29)

A test on the value of $y^+$ is implemented to automatically switch between eq. (5.28) and (5.29) with the interface at $y^+ = 11.08$. It is also worth mentioning that the SST model requires the distance to the wall at every cell. Although this can be simply defined as the distance between the cell centre and the closest wall face centre, for a parallel calculation (where the nearest wall face may lie on other processor), the treatment needs to be revised.

### Elliptic relaxation models

Out of three elliptic relaxation models implemented, only Durbin’s original model requires special attention. The boundary condition of $f$ at the wall is:

$$\lim_{y \to 0} f = \frac{-20 \nu \bar{v}^2}{\varepsilon y^4}$$

(5.30)

As pointed in Section 2.2.2, this strong dependency on $\bar{v}^2$ and $y$ poses a strong constraint on the time step used, since Code_Saturne is a segregated solver. A relaxation of the turbulent variables is implemented but although it improves robustness, the time step required is still extremely small compared to the other two elliptic models.

### SSG model and the scalable wall function

In Code_Saturne the standard wall function option is available for High Reynolds number models. It is necessary to prescribe a velocity at the cell face so that the code can integrate over the area and obtain the fluxes. The following explanation is based on Figure 5.2 which represents the first cell near the wall. In the standard wall function approach, the theoretical gradient at the wall is used to obtain the velocity at the face $U_F$.

$$G_{\text{theo}} = \frac{u_r}{\kappa d} = \frac{U_G - U_F}{2d}$$

(5.31)
The code is cell centred so the velocity $U_G$ is not known, but it can be approximated by $(U_J + U_I)/2$ so that the calculated gradient becomes:

$$G_{\text{calc}} = \frac{U_J + U_I - 2U_F}{4d} \quad (5.32)$$

The velocity at $J$ can be approximated by:

$$U_J = U_I + IJ \left( \frac{\partial U}{\partial y} \right)_G + \mathcal{O}(IJ^2) \quad (5.33)$$

and assuming that $G$ is within the log-law region:

$$\left( \frac{\partial U}{\partial y} \right)_G = \left( \frac{\partial}{\partial y} \left[ \frac{u_\tau}{\kappa} \log(y^+) + C \right] \right)_G = \left( \frac{u_\tau}{\kappa y} \right)_G = \frac{u_\tau}{\kappa 2d} \quad (5.34)$$

So equation (5.31) can be written as:

$$\frac{u_\tau}{\kappa y} = \frac{U_I + u_\tau}{2\kappa} - U_F \quad (5.35)$$

And the velocity at the face $U_F$ can be written as:

$$U_F = U_I - \frac{3u_\tau}{2\kappa} \quad (5.36)$$

This ensures that the gradient calculated by the code equals the log law prediction.
5.5. Notes on the implementation of new models

The boundary condition on the normal stresses are set by having zero gradient, that is 
\( \partial \langle u'_i u'_i \rangle / \partial n \) = 0. The shear stress is set to the wall shear stress \( \langle u'_1 u'_2 \rangle = u_\tau u_k \) and the other two stresses have a zero value \( \langle u'_1 u'_3 \rangle = \langle u'_2 u'_3 \rangle = 0 \).

A second order approximation is used for the boundary condition on \( \varepsilon \). The theoretical gradient is obtained from:

\[
\frac{\partial}{\partial y} \left( \frac{u^3_k}{\kappa y} \right) = -\frac{u^3_k}{\kappa y^2} \quad (5.37)
\]

The value of \( \varepsilon \) at the midpoint between the face and the centre (see fig. 5.2) can be evaluated as:

\[
\varepsilon_M = \varepsilon_I - \frac{d}{2} \left( \frac{\partial \varepsilon}{\partial y} \right)_{I} + \frac{d^2}{8} \left( \frac{\partial^2 \varepsilon}{\partial y^2} \right)_{I} + O(d^3) \quad (5.38)
\]
\[
\varepsilon_M = \varepsilon_F + \frac{d}{2} \left( \frac{\partial \varepsilon}{\partial y} \right)_{F} + \frac{d^2}{8} \left( \frac{\partial^2 \varepsilon}{\partial y^2} \right)_{F} + O(d^3) \quad (5.39)
\]

By subtracting these two expression the value of \( \varepsilon \) at the face can be written as:

\[
\varepsilon_F = \varepsilon_I - \frac{d}{2} \left( \frac{\partial \varepsilon}{\partial y} \right)_{I} + \frac{d}{2} \left( \frac{\partial \varepsilon}{\partial y} \right)_{F} + O(d^3) \quad (5.40)
\]

In a similar manner, the gradients can be evaluated as:

\[
\left. \frac{\partial \varepsilon}{\partial y} \right|_I = \left. \frac{\partial \varepsilon}{\partial y} \right|_M + \frac{d}{2} \left( \frac{\partial^2 \varepsilon}{\partial y^2} \right)_{M} + O(d^2) \quad (5.41)
\]
\[
\left. \frac{\partial \varepsilon}{\partial y} \right|_F = \left. \frac{\partial \varepsilon}{\partial y} \right|_M - \frac{d}{2} \left( \frac{\partial^2 \varepsilon}{\partial y^2} \right)_{M} + O(d^2) \quad (5.42)
\]

By adding these two equations and using equations (5.40) and (5.37), the final value at the face can be written as:

\[
\varepsilon_F = \varepsilon_I + d \frac{u^3_k}{k(y/2)^2} \quad (5.43)
\]

The rationale presented above has the disadvantage that it is based on a Cartesian grid. As the code is unstructured this has to be taken into account by the user. It is preferable to use meshes that have the first two cells of a similar size, since it is assumed that the cell centres are equally spaced. This in turn means having the first two cells of similar size which can be limiting to the number of points inside the boundary layer. To implement the scalable wall
function, the same procedure can be followed, but taking into account that for all the points below $y_{lim}^+$ the gradient should be the same as at $y_{lim}^+$.

$$\frac{\partial U}{\partial y} = \frac{u^2}{\kappa \nu y_{b}^+}$$  \hspace{1cm} (5.44)

with $y^+$ calculated from equation (2.69). Equation (5.35) can be then written as:

$$\frac{u^2}{\kappa y_{b}^+ \nu} = \frac{U_I + d \frac{u^2}{2\kappa y_{b}^+ \nu} - U_F}{2d}$$  \hspace{1cm} (5.45)

And the velocity at the face would be:

$$U_F = U_I - \frac{3u_r u_k d}{2\kappa y_{b}^+ \nu}$$  \hspace{1cm} (5.46)

Note that it is important to use $u_r u_k$ instead of $u^2_r$ in the previous equation because $u_k$ is the scaling criteria with the definition of $y_{b}^+$. The boundary condition on $\varepsilon$ can be written as:

$$\varepsilon_F = \varepsilon_I + 4d \frac{u_k^5}{k(\nu y_{b}^+)^2}$$  \hspace{1cm} (5.47)
Chapter 6

Channel flow

6.1 Introduction

The flow in a straight channel is the first test case to be studied because of its simple geometry and the extensive knowledge of its properties. It is illustrated in Figure 6.1.

In a fully turbulent plane channel flow, the mean velocity field is only dependent on the wall-normal direction and it is driven by a constant pressure gradient in the streamwise direction. All other statistical derivatives in the spanwise and streamwise directions are zero.

Figure 6.1: Plane channel flow
This simplifies the flow equations to:

\[
- \frac{1}{\rho} \frac{\partial P}{\partial x} = \frac{\partial}{\partial y} \left( \nu \frac{\partial U}{\partial y} \right) \tag{6.1}
\]

\[
- \frac{1}{\rho} \frac{\partial P}{\partial y} = \frac{\partial}{\partial y} \left( \nu \frac{\langle v'v' \rangle}{\partial y} \right) \tag{6.2}
\]

Since \( \frac{\partial \langle v^2 \rangle}{\partial x} = 0 \), the pressure gradient is independent of \( y \), and can be calculated by integrating

\[
- \frac{H}{\rho} \frac{\partial P}{\partial x} \bigg|_{w} = 2 \nu \frac{\partial U}{\partial y} \bigg|_{w} = \frac{2}{\rho} \frac{\tau_w}{\rho} = 2u_*^2 \tag{6.3}
\]

So the momentum equation can be written as:

\[
u_2 = \frac{H}{2} \frac{\partial \langle u'v' \rangle}{\partial y} - \frac{\partial}{\partial y} \left( \nu \frac{\partial U}{\partial y} \right) \tag{6.4}
\]

This can be integrated and using non dimensional values, yields:

\[
\frac{\partial U^+}{\partial y^+} - \langle u'v' \rangle^+ = 1 - \frac{y^+}{Re_\tau} \tag{6.5}
\]

where

\[
U^+ = \frac{U}{u_*}; \quad y^+ = \frac{yu_*}{\nu}; \quad \langle u'v' \rangle^+ = \frac{\langle uv \rangle}{u_*^2}; \quad Re_\tau = \frac{u_* H/2}{\nu} \tag{6.6}
\]

In near wall region of the flow the viscous stresses are dominant, and the velocity varies linearly with the distance from the wall. This zone is called the *viscous sublayer* and is usually defined for values of \( y^+ < 5 \). As the the distance from the wall increases, the velocity profile departs from the linear equation and enters into a zone called the *buffer layer* between \( y^+ > 5 \) and \( y^+ < 30 \). In this region, the turbulent stresses start to become important and the rate of energy production reaches its peak around \( y^+ \approx 10 - 15 \). At this point of maximum production the viscous stresses are in equilibrium with the turbulent stresses, that is \( -\langle uv \rangle^+ = \partial U^+ / \partial y^+ = \frac{1}{2} \). After the buffer layer (\( y^+ > 30 \)), comes the logarithmic profile which is valid up to \( y/H < 0.15 \). These regions can be seen in Figure 6.2 where the DNS data is plotted.

The DNS data of Kim et al. [65] is available for comparison with the results of the models. The channel flow at Reynolds number of \( Re_\tau = 395 \) is the first test case to be computed since it has been used for tuning many turbulence models.
6.2 Numerical framework

6.2.1 Eddy viscosity models

The mesh created for this case has one cell in the streamwise direction and 100 cells in the wall normal direction, with an expansion ratio of 1.1 and a minimum value of $y^+ = 0.5$. Periodic boundary conditions were applied in the stream-wise direction. The initial conditions were taken from analytical profiles for $U$, $k$, $\varepsilon$ and $\overline{v^2}$.

The turbulence models used in this study were:

- Launder Sharma $k - \varepsilon$
- Wilcox $k - \omega$
- Menter’s SST
- Durbin’s original $\overline{v^2} - f$
- LDM
- $\varphi - f$ model.

Figure 6.2: Plane channel flow $Re_\tau = 395$, Velocity profile.
For all the models the same time step was used except for the the original \( \overline{v^2} - f \) model which needed a time step 10000 times smaller. The stiffness of the boundary condition on the original \( \overline{v^2} - f \) model is the principal reason why such a small time step is required.

### 6.2.2 Second moment closure

The SSG model was used in this case to test the behaviour of the scalable wall function implemented in Code_Saturne. A channel flow at \( Re_\tau = 395 \) with four different meshes was computed. The \( y^+ \) values of the first cell centres were 0.5, 5, 8 and 15. All the calculations were performed by applying a fixed pressure gradient in order to impose the correct friction velocity.

### 6.3 Results

#### 6.3.1 Eddy viscosity models

The results for the simulation at \( Re_\tau = 395 \) can be seen in Figure 6.3 for the SST, \( k - \omega \) and Launder-Sharma models, and in Figure 6.4 for all the versions of the \( \overline{v^2} - f \) models. The results for the \( k - \omega \) and SST models are similar since in the SST the blending function \( F_1 \) is one in the whole domain. The Launder-Sharma model behaves in a similar way but with a higher peak of turbulent kinetic energy.

All models return a zero boundary value for the dissipation but for different reasons. Recall that the Launder-Sharma model solves the equation for \( \tilde{\varepsilon} \) which is defined by equation (3.14) and hence a zero boundary condition for the reduced dissipation is imposed. In the case of the \( k - \omega \) model the dissipation values are obtained from \( \varepsilon = C_\mu k \omega \) but not from direct values of a dissipation equation. If the above definition of \( \varepsilon \) is used in the transport equation of the \( k - \varepsilon \) model (2.36), the transformed equation in terms of \( \omega \) would be:

\[
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = C_{\varepsilon 1} \frac{\omega}{k} P_k - C_\mu (C_{\varepsilon 2} - 1) \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{2}{k} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]

which is not exactly the same as the dissipation equation for the \( k - \omega \) model (2.39). The transformed dissipation equation from the \( k - \varepsilon \) model has a cross-gradient term that acts as a
6.3. Results

Figure 6.3: Channel flow $Re_\tau = 395$. SST, $k - \omega$ and $k - \varepsilon$ Launder-Sharma

source of dissipation that is not present in the $k - \omega$ model, which explains the spurious peak around $y^+ = 10$. The other important feature that can be seen is the under prediction of the turbulent kinetic energy in the same region; this is balanced against the erroneous dissipation profile with the net result of a good turbulent viscosity profile and in turn, a good velocity prediction.

The results for the $\overline{v^2} - f$ models are shown in Figure 6.4 where two additional profiles are shown, $\overline{v^2}$ and $\varphi$. Since in the computations the pressure gradient was imposed to obtain a value for $u_*= 1$, there is no constant mass flow rate hence the difference in the area under the
velocity profiles. All the elliptic relaxation models tend to overpredict the velocity towards the centre of the channel. This problem has been addressed by different authors and some corrections to the model have been suggested ranging from changes in the Laplace operator...
to a change in the definition of $f$ to obtain a rescaled model \cite{74}. The kinetic energy profile is well reproduced including the peak and the asymptotic behaviour near the wall. The dissipation profile presents a strong kink in the region where $y^+ \approx 20$ which can be attributed to the fact that the dissipation equation used in the model is the high Reynolds form. A variable $C_{\varepsilon 1}$ coefficient has been added and the time scale has been modified to account for the behaviour near the wall.

A strong difference arises between the LDM model and Durbin’s original model in the profile of $\overline{\nu^2}$. This can come from the fact that the LDM model neglects a term that is non-zero in different parts of the channel as described in Chapter 4. This problem is evident in the profile of the turbulent viscosity which clearly shows an over prediction from the LDM compared to the other two. The results for the $\varphi - f$ model are in good agreement with the DNS data. It is important to see how the $\varphi - f$ model shows good predictions for $\overline{\nu^2}$, $\varphi$, and $\nu_t$ being closer to the original model.

Figure 6.5 shows a comparison of the terms in the $f$ equation that differ (’neglected’ terms) from the LDM and $\varphi - f$ models. They are obtained from the solution of Durbin’s original model. The dashed line is the distribution of the term neglected in the LDM, the dot-and-dash line is the term neglected in the $\varphi - f$ model. At the wall both terms are finite and similar. Outside the viscous sublayer, the neglected term in the $\varphi - f$ model is nearly two orders of magnitude smaller than the one in the LDM model. All terms in the $f$ equation can be seen in Figure 6.6. The so-called homogenous $f$ is the right-hand side of equation (3.43), i.e. the pressure-strain obtained without elliptic relaxation. The figure is split in two for better visualisation. Beyond $y^+ = 100$ the scale is magnified as the scale of all terms decreases rapidly. The left hand side (a) shows the near wall region where it can be seen that the neglected term in the $\varphi$ model goes to zero as of $y^+ = 30$ whereas the neglected term in the LDM only goes to zero around $y^+ = 80$. On the right hand side of the figure (b) the term neglected in the LDM increases near the centre of the channel and is actually larger than $f$ itself. On the contrary the term neglected in the $\varphi - f$ model is almost zero, as expected for a purely viscous term. Near the wall it is also seen that the elliptic term $L^2 \nabla^2 f$ is strongly damping the homogenous part of $f$ \cite{38}, mimicking the blockage that the wall is exerting on pressure-strain redistribution. As discussed by Manceau et al. \cite{74} this term should actually
vanish in the central part of the flow rather than increase the pressure-strain term.

![Figure 6.5: Comparison of the neglected terms in logarithmic scale (absolute values).](image)

![Figure 6.6: Terms in the $f$ equation.](image)

In other words, in the middle of the channel, the elliptic relaxation should vanish and $f$ should be equal to the homogeneous formulation used, in this case, the Launder Reece and Rodi model. The effect of the change of variable to force a zero boundary condition at the wall should only be felt in the near wall region but this is not the case for the LDM model. In Figure 6.7 the profiles for $f$ are shown for the three elliptic relaxation models. For the LDM model $f$ has been obtained from equation (3.65) and for the $\varphi - f$ model from equation (4.7).

The $\varphi - f$ and the original $\bar{\nu}^2 - f$ models present quite similar behaviour in the middle of the channel, but the LDM does not, since the extra term $(5\bar{\nu}^2/k)$ acts as a source term. This in turn makes the profile of $\bar{\nu}^2$ to depart from the homogeneous value so that the difference $(\bar{\nu}^2/k - 2/3)$ is no longer zero. The LDM formulation does not guarantee that the value of the homogeneous pressure strain is recovered far from the wall. This can be interpreted as a modified pressure strain model.

Similar results are obtained at higher Reynolds numbers. In Figure 6.8 the results for $k - \omega$, SST and the Launder-Sharma $k - \varepsilon$ models are shown for $Re_\tau = 640$. In Figure 6.9 the profiles for the elliptic relaxation models can be seen. The results have the same trend as described above for the lower Reynolds number case.
6.3. Results

![Graph showing Profiles of \( f \) in a Channel flow \( Re_\tau = 640 \).](image)

Figure 6.7: Profile of \( f \) in a Channel flow \( Re_\tau = 640 \).

### 6.3.2 Second Moment Closure

In Figure 6.10 the different profiles can be seen. All the values have been made non-dimensional by using \( u_\tau \) computed from the fixed pressure gradient imposed and is the same for all graphs. The velocity decreases with the value of \( y^+ \). In Figure 6.11 the results are shown for the scalable wall function. The changes as the mesh is refined are very small. It is important to note that the use of a finer mesh does not imply that the variables are resolved in the viscous sublayer. The advantage as pointed before is that the solution obtained does not depend on a minimum value of \( y^+ \).

The standard wall function as coded in \textit{Code_Saturne} is very sensitive to the size of the first cell as shown in Figure 6.10, the total friction near the wall is too high because \( \langle u'v' \rangle \) is not negligible. Since the momentum balance is:

\[
\nu \frac{\partial U}{\partial y} - \langle u'v' \rangle = u_\tau^2 \left( 1 - \frac{y}{e} \right)
\]

and the right hand side is given by the pressure gradient, \( \partial U/\partial y \) is reduced to compensate for the overestimation of \( \langle u'v' \rangle \). The shear stress is too high as it is produced by \( \langle v'v' \rangle \) which is also over predicted because no low-Reynolds version of the model is used here.

On the other hand the SSG with scalable wall function results in Figure 6.11 are very satisfactory above \( y^+ = 10 \) and not sensitive to the size of the first cell. What makes the scalable wall function successful is that the computed velocity gradients below \( y^+_l \) are much
smaller, hence there is a smaller production of the stresses alleviating the need for the low Reynolds number damping terms.

Figure 6.8: Channel flow $Re_+ = 640$. 
Figure 6.9: Channel flow, elliptic relaxation models $Re_\tau = 640$. 
Figure 6.10: Mesh refinement in a channel flow. Standard wall function. SSG Model.
Figure 6.11: Mesh refinement in a channel flow. Scalable wall function. SSG Model.
Chapter 7

Turbulent heated cavity

7.1 Introduction

Natural convection is present in many industrial cases involving heat transfer. These applications range from domestic heating to cooling in nuclear plants. In natural convection driven flows, an accurate representation of the gradients near the wall is usually needed due to the low-Reynolds number nature of the flow and the buoyant effects on turbulence. These effects are large where the change in temperature is large, i.e. very near the wall. When heat transfer is introduced over vertical boundaries, the shear stress varies rapidly near the wall and changes sign. This means the usual wall function approach described in Section 2.5 is unable to capture important features of the flow.

The incompressible Navier-Stokes equations can be used in natural convection cases with moderate density variations. The density changes only enter in the form of a gravitational force $\rho g_i$, they read:

$$
\rho \frac{\partial \langle U_i \rangle}{\partial t} + \langle U_j \rangle \rho \frac{\partial \langle U_i \rangle}{\partial x_j} = -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \langle U_i \rangle}{\partial x_j} - \rho \langle u'_i u'_j \rangle \right) + \rho g_i \quad (7.1)
$$

The changes in density can be related to changes in temperature via the thermal expansion coefficient $\beta$ defined as:

$$
\beta = -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \quad (7.2)
$$

A temperature equation needs to be solved in order to couple the changes in temperature to
the velocity field via the gravitational force. The temperature equation reads:

\[ \rho \frac{\partial T}{\partial t} + \langle U_i U_j \rangle \rho \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu}{P r} \frac{\partial T}{\partial x_j} - \rho \langle u'_j T \rangle \right) \]  

(7.3)

Using the Boussinesq approximation, the unknown term \( \langle u'_i u'_j \rangle \) can be evaluated as:

\[ \rho \langle u_i u_j \rangle = \rho \frac{2}{3} k \delta_{ij} - \mu_t \left( \frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \]  

(7.4)

In order to use the eddy viscosity models in buoyant cases, the production of kinetic energy must be altered by adding a gravity term. That is:

\[ P = \mu_t \frac{\partial \langle U_i \rangle}{\partial x_j} \left( \frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) + G \]  

(7.5)

with \( G \) defined as

\[ G = \rho \beta g_i \langle u'_i T \rangle \]  

(7.6)

The term \( \rho \langle u'_i T \rangle \) can be evaluated by using the Boussinesq approximation:

\[ \rho \langle u'_j T \rangle = - \frac{\mu_t}{\rho P r} \frac{\partial T}{\partial x_j} \]  

(7.7)

where \( Pr \) is the Prandtl number (ratio of viscosity to thermal diffusivity). In vertical boundary layers, the temperature gradient \( \partial T/\partial y \) becomes very small, the model term is negligible and the real effect of buoyancy in the gravity term \( G \) is underestimated. Therefore another approximation can be used. The Generalised Gradient Diffusion Hypothesis (GGDH) introduced by Daly and Harlow [26] can be written as:

\[ \rho \langle u'_j T \rangle = - \frac{\rho c_\theta}{\varepsilon} \langle u'_i u'_j \rangle \frac{\partial T}{\partial x_j} \]  

(7.8)

The term \( \rho \langle u'_j u'_i \rangle \) can be evaluated using equation (7.4).

In the models implemented in *Code_Saturne* during the course of this work, the \( k - \omega \) model and SST model need the gravity term in the production term on the kinetic energy equation. In all the explicit relaxation models, the production of kinetic energy also appears in the \( f \) equations and therefore the gravity term must also be added.

The different approximations to the gravity term (eqs (7.7), (7.8)) were studied by Davidson [27] with different turbulence models in a square cavity with Rayleigh number \( Ra = \)
7.1. Introduction

It was found that the low Reynolds number models tend to laminarise the flow giving erroneous predictions. Using wall functions, it was noted that the GGDH approximation was negligible inside the boundary layer but in the outer part it accounted for about 30% of the largest term in the turbulent kinetic energy balance. The GGDH formulation gave a higher value than the Boussinesq approximation for the turbulent heat flux in the whole domain except very near the top wall (y/H=0.99) where it could be seen that the temperature gradient in the vertical direction had an impact on the gravity term. The two different formulations were studied by Tieszen et al. [112] using the $\nu^2 - f$ model in two buoyant cases, a vertical flat plate, and a cavity of aspect ratio 5:1. They found that the GGDH approximation had little effect on the flat plate case, but on the cavity it slightly improved the predictions. Hsieh and Lien [55] also studied the effect of both formulations using linear two equation models in square and rectangular cavities and concluded that the predicted results were virtually identical, suggesting that for these types of models, the buoyancy force does not play an important role in turbulence generation.

Figure 7.1: Turbulent heated cavity
The natural convection cavity studied by Betts and Bokhari [15] has been computed herein (See Figure 7.1). The flow inside the cavity is driven by the temperature difference between the vertical walls creating boundary layers where the shear stress changes rapidly close to the wall, therefore a turbulence model that is capable of resolving correctly the near wall region is needed. The advantage of the tall cavity is that the flow is fully turbulent, since turbulence depends on the aspect ratio and Rayleigh numbers. This configuration represents one of the simplest geometries with industrial applications. The experiments were carried out at two different Rayleigh numbers, \( Ra = 0.86 \times 10^6 \) and \( 1.43 \times 10^6 \) in a cavity with an aspect ratio of \( H/W = 28.68 \). The experiments reported a two dimensional, antisymmetric flow and close attention was paid to the adiabatic boundary condition on the horizontal walls. This makes the case attractive for validation of computational models as proved in two ERCOFTAC workshops [11] and [2]. Boudjemadi et al. [16] studied a case with infinite plates with DNS and second moment closure including elliptic relaxation. It was found that the model performs fairly well on a global scale but improvement could be made in a term-by-term basis. DNS data is also available from Versteegh and Nieuwstadt [117].

### 7.2 Numerical framework

Three cartesian structured grids have been generated for this case, with \( 100 \times 50 \), \( 150 \times 100 \) and \( 200 \times 50 \) cells. The solutions on the three meshes were almost identical therefore ensuring grid independence. The results shown here have been obtained with the coarsest grid. All the calculations have been carried out in two dimensions and the results have been compared with the experiments in the middle of the cavity \( (z/H) = 0 \). The turbulence models that have been tested in this case were:

- Launder Sharma \( k - \varepsilon \)
- Wilcox \( k - \omega \)
- Menter’s SST
- LDM
7.3 Results

- \( \varphi - f \) model.

The computations have been carried out using the same timestep and a steady solution was obtained. The Launder-Sharma model used in this study does not contain the Yap correction [121] which is introduced in order to force the near-wall length scale close to the equilibrium value. This correction improves the results in natural convection as reported by Hsieh and Lien [55]. The results presented here have been obtained using equation 7.7 unless stated otherwise.

7.3 Results

The velocity profiles for the \( \omega \) based models are shown in Figure 7.2 at \( Ra = 0.86 \times 10^6 \). The SST and \( k - \omega \) models give similar predictions close to the experimental results at different heights. In the middle of the cavity \( (y/H = 0.5) \), the \( k - \omega \) model produces a change in the slope of the vertical velocity profile while the SST gives an approximate straight line closer to the experimental data. On the next measurement plane \( (y/H = 0.6) \) the change of slope is present in the experimental data, therefore giving the \( k - \omega \) model prediction a better agreement. The \( k - \omega \) model seems to overestimate this change of slope after the velocity maxima, whereas the SST does not capture it. Near the top, both models have similar behaviour since the blending functions of the SST is designed to approach one near the no-slip boundaries and the viscosity limiter acts towards the middle of the cavity far from the corners. Here the models gives slightly lower values for the velocity peaks near the wall. This is compensated in the middle of the cavity by departing from the straight line that is obtained from the experiments. The temperature results can be seen in Figures 7.3 and 7.5. Here both models give very similar predictions for all experimental planes, with the largest difference at \( y/H = 0.9 \).

The vertical velocity profiles for the \( \varepsilon \) based models in the upper half of the domain can be seen in Figure 7.4. The predictions of the \( \varphi - f \) model are clearly in better agreement with the experiment than the other two models, especially in the middle and the top portion of the cavity for both Rayleigh numbers considered. The LDM model over predicts the velocity peak in the middle of the cavity, with errors at higher Rayleigh number of 40% and 31% for
Figure 7.2: Velocity profiles at $Ra = 0.86 \times 10^6$, $\omega$ based models.

the lower Rayleigh number, whereas the $k - \varepsilon$ model underpredicts the velocity peaks in most of the domain. Near the horizontal walls, the $\varphi - f$ model slightly under predicts the velocity on the decelerating side, but the slope of the velocity profile in the mid-width is closer to the experiment, compared with the LDM. In terms of temperature profiles there is not much difference between both formulations, both give good predictions at the different stations of the cavity. The results for the high Rayleigh number, $Ra = 1.43 \times 10^6$, can be seen in Figures 7.6, 7.7, 7.8 and 7.9. The same conclusions apply for this higher Rayleigh number; the $\varphi - f$ seems to provide the best predictions for the velocity field. The SST model is close to the $k - \omega$ and in good agreement with experiments.
Temperature profiles at $Z = 0$
$Ra=0.86 \times 10^6$

- Exp
- $k - \omega$
- SST

Figure 7.3: Temperature profiles at $Ra = 0.86 \times 10^6$, $\omega$ based models
Velocity profiles at \( Z = 0 \), \( Ra = 0.86 \times 10^6 \)

- Exp
- \( \varphi - f \)
- LDM
- \( k - \varepsilon \) LS

Figure 7.4: Velocity profiles at \( Ra = 0.86 \times 10^6 \), \( \varepsilon \) based models.
Temperature profiles at \( Z = 0 \)
\( Ra=0.86 \times 10^6 \)

- Exp
- \( \Phi - f \)
- LDM
- \( k-\varepsilon \) L-S

Figure 7.5: Temperature profiles at \( Ra = 0.86 \times 10^6 \), \( \varepsilon \) based models.
Velocity profiles at $Z = 0$
$Ra=1.43 \times 10^6$

- $\text{Exp}$
- $k - \omega$
- SST

Figure 7.6: Velocity profiles at $Ra = 1.43 \times 10^6$, $\omega$ based models
Temperature profiles at $Z = 0$
Ra=$1.43 \times 10^6$

- Exp
- $k - \omega$
- SST

Figure 7.7: Temperature profiles at $Ra = 1.43 \times 10^6$, $\omega$ based models
Velocity profiles at $Z = 0$

$Ra = 1.43 \times 10^6$

- Exp
- $\varphi - f$
- LDM
- $k - \varepsilon$ LS

Figure 7.8: Velocity profiles at $Ra = 1.43 \times 10^6, \varepsilon$ based models.
Temperature profiles at $Z = 0$
$Ra=1.43 \times 10^6$

- Exp
- $\phi - f$
- LDM
- $k - \varepsilon$ LS

Figure 7.9: Temperature profiles at $Ra = 1.43 \times 10^6$, $\varepsilon$ based models.
Chapter 8

Asymmetric plane diffuser

8.1 Introduction

The flow in a plane asymmetric diffuser is a challenging test case since its adverse pressure gradient makes the flow separate from the inclined surface, so the separation point is not fixed by the geometry. The geometry of the case can be seen in Figure 8.1. The $x$ axis is taken in the streamwise direction with the origin at the start of the expansion section. The Reynolds number based on the inlet height $H$ and the bulk velocity is $Re = 18000$. This case was examined experimentally first by Obi et al. [86] and later by Buice and Eaton [17]. This diffuser was a test case for the 8th ERCOFTAC/IAHR Workshop [53], where different RANS models were presented. The case was also studied by Kaltenbach et al. [63] using Large Eddy
Simulation with different grid refinements. The case is interesting from the modelling point of view because the well defined boundary inlet conditions (fully developed channel flow), the smooth separation (due to the low angle of the inclined wall), and the recovery region after the reattachment. Determining the separation correctly depends on the models’ abilities to predict a correct turbulent time scale at the accelerated wall. To mimic the reattachment and recovery correctly, the model will need to predict the correct magnitude of the shear stress in a curved free shear layer [7].

Buice and Eaton found that in Obi’s results there was an increase in the measured mass flow rate of about 15% which led them to believe that there were significant 3D effects, and that the results could be affected by developing boundary layers on the channel walls. Therefore they repeated the experiment, paying close attention to the two-dimensionality of the case and also improving the measurements, since more planes for the data extraction were used. The skin friction coefficient on the inclined wall was also measured, a variable that was only measured on the straight wall in the previous experiment. The separation point given by Obi et al. was at 11H whereas for the Buice and Eaton experiment separation occurred at 7.5H. The recirculation length of Obi et al’s experiment was shorter than Buice and Eaton’s with reattachment points at 26H and 29H respectively.

Kaltenbach et al. [63] showed that the LES calculation tended to agree reasonably well with the experimental data from Buice and Eaton but that Obi et al.’s results were only valid up to a distance of 21H measured from the start of the expansion section. The LES predicted a separation point at 7H and reattachment at 27H which lies in the same region as the Buice and Eaton experiment.

The test case has been treated by various authors with a variety of turbulence models. Apsley and Leschziner [7] used the $k - \varepsilon$, SST and the $k - \omega$ models as well as non-linear eddy viscosity models such as the Craft et al. [22] and the Apsley and Leschziner [6] models. They also tested some Reynolds stress models such as the Gibson and Launder [43] and the SSG [104] models. Durbin [34] tested the $\overline{u^2} - f$ model and the LDM model was later tested on the same case by Iaccarino [56].

The $k - \varepsilon$ model fails to predict any separation, whereas the $k - \omega$ model predicts separation but with a smaller recirculation compared to the experiments. The SST model performs
better than the other two models but the separation point is predicted too early \((x/H = 2)\), although the reattachment point is close to the experimental one. The non-linear models were reported to predict separation with good reproduction of the maximum velocity along the diffuser axis but they under predicted the reverse flow velocity. The Reynolds stress models had worse results with the use of wall functions being the main reason for lower performance. The \(\overline{v^2} - f\) model was reported to have a very good performance with separation and reattachment points almost identical to the experiments \([38]\). The LDM model was also reported to yield a good agreement with the experiments although no comparison was made between the two versions.

8.2 Numerical framework

A mesh of \(96 \times 192\) cells was used which was sufficient for the case \([7]\). The maximum non-dimensional distance of the first node from the wall was \(y^+ = 0.62\). The inlet conditions were obtained by running a separate calculation of a periodic channel in order to obtain a complete set of inlet conditions for all the variables. A centred scheme was used for all the velocity components and an upwind scheme was chosen for the turbulent variables. The models used in this case were:

- Launder Sharma \(k - \varepsilon\)
- Wicox \(k - \omega\)
- Menter SST
- LDM
- \(\varphi - f\) model.
- SSG with scalable wall function.

All computations were carried out with a CFL number of 1.6 except for the Launder Sharma model, which needed a CFL number ten times smaller.
8.3 Results

In Figure 8.2 the friction coefficient along the inclined wall is shown. Here it can be seen that the Launder Sharma model fails to predict any separation at all as was observed at the ERCOFTACH/IAHR workshop. The $k-\omega$ model performs much better than the Launder-Sharma model but the separation and reattachment points are too early. The SST gives an even larger recirculation predicting the separation point just after the beginning of the diffuser, but with a reattachment point close to the experiments. The two elliptic relaxation models present good agreement in terms of skin friction coefficient but the LDM model has a slightly earlier separation point, whereas the $\varphi - f$ model predicts both points very near to the experimental results, but with a lower value in the recirculation zone. The skin friction is overpredicted by both models in the recovery region with the LDM being slightly better. The SSG model shows a very small zone where the friction coefficient is negative, but this is due to the double inflection point near the wall. This means that the zone with negative velocity is large, but the velocity becomes positive near the wall, giving an erroneous shape as can be
8.3. Results

seen from Figure 8.3.

Figure 8.3: Asymmetric plane diffuser. Streamlines for the SSG model

<table>
<thead>
<tr>
<th>Experimental data</th>
<th>Launder Sharma</th>
<th>k - ω</th>
<th>SST</th>
<th>LDM</th>
<th>ϕ model</th>
<th>SSG SCWF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 8.4: Asymmetric plane diffuser. Pressure coefficient along the inclined wall

In Figure 8.4 the pressure coefficient along the inclined wall is shown. Here the ϕ − f model gives the best prediction out of the EVMs, followed by the SST which is very close to the LDM. The SSG gives surprisingly good predictions for the pressure coefficient.

The velocity profiles at the first section of the diffuser can be seen in Figure 8.5 for the ε based models. In general, the effect of the elliptic relaxation is to improve the predictions of the k − ε model. The results for the LDM and the ϕ − f models are in good agreement with the experimental data but the LDM tends to underpredict the velocity at mid-height. This could be due to the effect of the change in variable explained in chapter 6, which tends to increase the relaxation effect far from the wall giving an erroneous prediction for \( \overline{v^2} \) and in turn a large turbulent viscosity. In the second half of the domain (\( x/h > 30 \), Figure 8.6) the models do not recover fast enough and the effect of the separation bubble can be seen in
almost all of the remaining domain of the diffuser. Since the $k - \varepsilon$ model does not predict separation, it recovers the channel flow profile around $x/h = 50$. This is not to say that the Launder-Sharma gives better predictions but only an effect of the lack of separation. The effect of the elliptic relaxation can be understood by looking at the contours of $f$ and $\varphi$ in Figures 8.7 and 8.8. The variable $\varphi$ has a value of $2/3$ in isotropic flows, so at high values, the effect of elliptic relaxation is small. It can be seen how $\varphi$ grows as the flow recovers after reattachment. The effect of $f$ is to act as a source of $\varphi$ so at the beginning of the diffuser, $f$ increases but decreases rapidly towards the centre of the flow, far from the wall. In this way the presence of the inclined wall is felt and the modified lower turbulent viscosity leads to separation. This does not happen with the $k - \varepsilon$ model.

For the $\omega$ based models, the velocity profiles can be seen in Figures 8.9 and 8.10. The effect of the SST model is to increase the recirculation length predicted by the standard $k - \omega$ model. The $k - \omega$ model predicts separation too late and a too early reattachment, the SST predicts separation too early but the reattachment is close to the experimental value. This is compensated by an overprediction of the velocity profiles near the straight wall. This effect is carried on along the diffuser, therefore leading to a very slow recovery.

The effect of the SST model over the $k - \omega$ model can be analysed by looking at the two modifications of the model. First, the effect of the blending function $F_1$ (equation (2.46)) which combines the $k - \omega$ near the wall and $k - \varepsilon$ far from it. In Figure 8.11 the contours of $F_1$ are shown. It can be seen how most of the domain is computed as $k - \omega$ except a region under the recirculation zone. The effect on the viscosity limiter can be seen on figure 8.12 where the zone above 1 is where the viscosity becomes larger than the limiter. It might explain why the model predicts early separation since the limiter starts acting at the beginning of the inclined wall. When the limiter acts, the turbulent viscosity is reduced and hence the effect is similar to the one described by the introduction of the elliptic relaxation. The velocity predictions of the SSG model (Figures 8.13 and 8.14) are far from the experiments, due to the fact that the wall effects are not taken into account. The use of the scalable wall function for this flow is not satisfying, proving that here, the log-law assumption is not valid.
Figure 8.5: Asymmetric plane diffuser. Velocity profiles, first half. $\varepsilon$ based models.
Figure 8.6: Asymmetric plane diffuser. Velocity profiles, second half. $\varepsilon$ based models.
Figure 8.7: Asymmetric plane diffuser. Contours of $f$ for the $\varphi - f$ model

Figure 8.8: Asymmetric plane diffuser. Contours of $\varphi$ for the $\varphi - f$ model
Figure 8.9: Asymmetric plane diffuser. Velocity profiles, first half. $\omega$ based models.
Figure 8.10: Asymmetric plane diffuser. Velocity profiles, second half. $\omega$ based models.
Figure 8.11: Asymmetric plane diffuser. Contours of blending function $f_1$ for the SST model

Figure 8.12: Asymmetric plane diffuser. Contours of viscosity limiter for the SST model
Figure 8.13: Asymmetric plane diffuser. Velocity profiles, first half. SSG model
Figure 8.14: Asymmetric plane diffuser. Velocity profiles, second half. SSG model
Chapter 9

Flow over periodic hills

9.1 Introduction

The case consists of a series of two dimensional hills with a polynomial shape described by Almeida et al. [5]. The Reynolds number is $Re = 10595$ based on the hill height and the bulk velocity on the crest of the hill. The computational domain can be seen in Figure 9.1 where the dimensions are, $L_x = 9h$ and $L_y = 3.036h$ where $h$ is the height of the hill. The flow is periodic in the stream wise direction. This case is relevant because of the strong separation after the hill. It has been studied in two ERCOFTAC/IAHR workshops [58] and [73] due to the relevance to turbulence modelling. The strong curvature makes the flow separate relatively early, just after the crest of the hill. The recirculating flow is present halfway throughout the domain. After reattachment comes a short period of recovery, and then the
following hill makes the flow accelerate. The separation is highly time and space dependent [110], therefore the prediction of high turbulence in the near wall region is extremely difficult from a statistical point of view. The RANS models must be able to predict shear stress as it changes rapidly along the surface of the hill. At the crest of the hill, the model is required to predict the high levels of turbulence from the intermittent separation to predict the correct separation point. The separated region is highly anisotropic therefore making it difficult for simple models that do not capture anisotropic effects. The reattachment region is where large energetic eddies impact on the wall with strong pressure fluctuations redistributing energy between the stresses. The RANS models that fail to capture this effects are expected not to be able to predict the correct recirculation region, therefore having a erroneous reattachment point. After reattachment, the model must predict the right rate of recovery, which is a defect of most RANS models. If the reattachment location is predicted too late, the flow has insufficient time to recover before the strong acceleration of the following hill. In the acceleration region the model must correctly predict the strong normal strain in a region where there is no separation in the mean but an incipient instantaneous recirculation [78].

LES data is available for comparison [110], at ten different sections on the domain. Different turbulence models have been tested on this configuration, including $k - \omega$, SST, linear and non-linear $k - \varepsilon$ models, and the Second Moment Closures in the form of the LLR and SSG models. In the 10th ERCOFTAC/IAHR workshop, the case was treated again but with inlet/outlet boundary conditions rather than periodic conditions in order to avoid the build up of errors. The LES profiles at a section of $x/h = 8$ were taken as inlet boundary conditions. Although in many of the contributions, there was a slight difference between the results compared to the periodic case, it could be concluded that the build up of errors was not dramatic and the periodic case was sufficient to estimate the models’ capabilities to predict separation and reattachment. The streamlines obtained by the LES calculation can be seen in Figure 9.2, where the separation point lies at $x/h = 0.22$ and the reattachment point is at $x/h = 4.72$. The results presented in the 9th ERCOFTAC/IAHR workshop showed that the standard $k - \varepsilon$ models predict a smaller recirculation length, whereas the $k - \omega$ models were closer to the LES. The SST model produced an over prediction of the length of the bubble, as did the non-linear $k - \varepsilon$ models. The Second moment closures also predicted a larger recirculation
bubble but they had shear stress profiles closer to the one obtained by LES. The shear stress is an important parameter in this case since it allows the exchange of momentum between the high velocity parts of the flow and the retarded zones. An over prediction of the shear stress near the top of the hill will result in a shorter bubble.

In general the low Re $k - \varepsilon$ models predicted a recirculation length about 60% of the LES with a significant over prediction of the shear stress $\langle u'v' \rangle$ in the last section at $x/h = 8$, which results in a totally wrong profile at the top of the hill. The high-Reynolds versions of the model did not give encouraging results, but presented a bigger recirculation zone compared to their low-Reynolds counterpart.

Jang et al. [59], tested several non-linear models alongside the Lauder-Sharma $k - \varepsilon$ model, the Wilcox $k - \omega$ model and the Low Reynolds version of the SSG model. The conclusion obtained was that the $\omega$ based models performed much better than the $\varepsilon$ based ones. It was also shown that the excessive recirculation length returned by both the $\varepsilon$ based models and the Low Re SSG models was linked to insufficient diffusion in the separated shear layer, which means over prediction of the shear strain.

9.2 Numerical framework

Two meshes were created to be used in Code_Saturne and to study the mesh dependence of the results. The coarse one had $172 \times 120$ cells with an expansion ratio of 1.05 in the $y$ direction with the nearest cell to the wall having a value of $y^+ = 0.82$. The fine grid had $180 \times 150$ cells and the results were almost identical to the coarse one, therefore the coarse mesh has been used for all of the results presented herein. A centred scheme was used for the velocities while an upwind scheme was used for the turbulent variables. The models used in this case were:

- Launder-Sharma $k - \varepsilon$
- Wilcox $k - \omega$
- SST
- Durbin’s original $\bar{v}^2 - f$
• LDM

• \( \phi - f \) model.

• SSG Reynolds Stress Model with Scalable Wall Function

Convergence with all models was obtained with the same CFL number except with the original \( \bar{v}^2 - f \) that required a CFL number one thousand times smaller. This poses a problem in the robustness of the model and shows how numerical difficulties can arise from the stiffness of the boundary condition. The actual computing time for the calculation to converge with Durbin’s original model is far greater than any of the other models implemented, even when started from a previous converged solution obtained from the LDM model. All the other computations were started from constant values for the velocities and turbulent variables.

### 9.3 Results

The predicted streamlines for all models can be seen in Figure 9.2. The \( k - \varepsilon \) model gives a too small recirculation region and completely misses the separation and reattachment points. Although the length of the recirculation zone is well predicted by the elliptic relaxation models (within less than 10% of error) the shape is not accurate. The separation angle is too low resulting in a recirculation bubble that is not thick enough. The velocity predictions for the \( \phi - f \) model and the original model are virtually identical therefore giving a very similar streamline field. The \( \varepsilon \) based models tend to give a slightly delayed separation. The introduction of elliptic relaxation increases the size of the recirculation bubble and brings the reattachment point close to the LES predictions. The \( k - \omega \) model gives a slightly larger and thicker recirculation zone. All \( \omega \) based models have the similar separation point, earlier than the \( \varepsilon \) based ones and closer to the LES. The reattachment location is too far away downstream, specially for the SST model. The SSG model gives a shorter recirculation length probably due to the lack of the introduction of near-wall effects, and presents a double “kink” on the streamlines near the reattachment point. This double inflection point was also reported on other Second Moment Closures at the ERCOFTAC workshop [58]. This behaviour was noted earlier by Hanjalić [49] and a length scale correction term was proposed. The summary
9.3. Results

of separation and reattachment predictions can be seen in Table 9.1. Velocity profiles can be

seen in Figure 9.2 for the three elliptic relaxation models. The predictions for the streamwise velocity of the $\varphi - f$ model and the original $\overline{v^2} - f$ model are in very close agreement, as it could be expected since the purpose of the $\varphi - f$ model is to be able to obtain similar behaviour as the original. The LDM version gives slightly different results especially in the middle region away from the walls, a similar behaviour as in the diffuser case described in Chapter 8. The velocity profiles for the other three EVMs can be seen in Figure 9.4. Here


<table>
<thead>
<tr>
<th>Model</th>
<th>Separation</th>
<th>Reattachment</th>
<th>Length</th>
<th>%</th>
</tr>
</thead>
<tbody>
<tr>
<td>LES</td>
<td>0.22</td>
<td>4.72</td>
<td>4.5</td>
<td>100</td>
</tr>
<tr>
<td>Launder-Sharma</td>
<td>0.38</td>
<td>2.93</td>
<td>2.55</td>
<td>56.8</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.283</td>
<td>5.63</td>
<td>5.34</td>
<td>118.82</td>
</tr>
<tr>
<td>SST</td>
<td>0.283</td>
<td>7.54</td>
<td>7.257</td>
<td>161.26</td>
</tr>
<tr>
<td>$\overline{v^2} - f$</td>
<td>0.302</td>
<td>5.18</td>
<td>4.878</td>
<td>108.4</td>
</tr>
<tr>
<td>LDM</td>
<td>0.364</td>
<td>4.82</td>
<td>4.45</td>
<td>99.02</td>
</tr>
<tr>
<td>$\varphi - f$</td>
<td>0.38</td>
<td>4.49</td>
<td>4.11</td>
<td>91.33</td>
</tr>
<tr>
<td>SSG SCWF</td>
<td>0.60</td>
<td>3.48</td>
<td>2.88</td>
<td>64</td>
</tr>
</tbody>
</table>

Table 9.1: Separation and reattachment points for various models

it can be seen how the $k - \varepsilon$ model underpredicts the recirculation region and by $x/H = 3$ the flow is already attached. In the next section, the flow has more time to recover therefore giving better predictions but again because of the wrong reasons. The $k - \omega$ gives a better performance compared to the SST, although both give delayed reattachment. The SST profiles show how the fact that the recirculation is too long, affects the shape of the velocity.

The SSG velocity predictions are shown in Figure 9.5. They show how the velocity peak near the wall at $x/H = 0.05$ is correctly predicted, indicating that this is a phenomenon more related to the stress transport than to the wall effects. Even though that profile is correctly predicted, the separation is too late and the reattachment too early. In the second half of the domain it can be seen how the velocity profiles have a double inflection point which is completely wrong.

All eddy viscosity models missed the strain due to the high accelerating layer on the hill and therefore the velocity peak near the lower wall in the first section ($x/H = 0.05$) is not reproduced. This causes the angle of separation to be too low leading to the mismatch in the zone around ($y/H = 1$). This can also be seen by looking at the shear stress profiles in Figures 9.6 and 9.7 where the rapid change in sign at $x/H = 8$ is not captured by any of the models. This in turn means that after the hill, the shear stress is also badly predicted. This erroneous behaviour induces the models to under estimate the peak shear stress in the recirculation zone. Although the SSG model does not capture the change in sign near the
9.3. Results

Figure 9.3: Flow over periodic hills. Velocity profiles for the elliptic relaxation models

wall at $x/H = 8$ (see figure 9.8), after the hill the level of shear stress in close to the LES prediction, with a magnitude much higher than any of the other models. The low levels of turbulence can be seen in the kinetic energy profiles in Figures 9.9, 9.10 and 9.11. All of the eddy viscosity models fail to predict the high peak near the lower wall at the separation region ($x/H = 0.05$) created by previous the acceleration ($x/H = 8$). This is also the case for the SSG model, for which no near wall modification has been used.
Figure 9.4: Flow over periodic hills. Velocity profiles. SST, $k - \omega$ and Launder-Sharma models.
Figure 9.5: Flow over periodic hills. Velocity profiles. SSG model

- LES
- SSG Scalable WF
Figure 9.6: Flow over periodic hills. Shear stress profiles, elliptic relaxation models
Figure 9.7: Flow over periodic hills. Shear stress profiles, SST, $k - \omega$ and Launder-Sharma models.
Figure 9.8: Flow over periodic hills. Shear stress profiles, SSG model
Figure 9.9: Flow over periodic hills. Kinetic energy profiles, elliptic relaxation models
Figure 9.10: Flow over periodic hills. Kinetic energy profiles, SST, $k - \omega$ and Launder-Sharma models.
Figure 9.11: Flow over periodic hills. Kinetic energy profiles, SSG model.
Chapter 10

Hybrid modelling using the $\varphi - f$ model

10.1 The hybrid approach

The use of LES techniques is very attractive for the reason that it requires less modelling than RANS, since the large scales are resolved and the small scales can be modelled in a relatively simple way. The main disadvantage of LES arises in the near wall region, because the latter introduces high anisotropy and therefore the small scales are difficult to capture. This means that to use LES accurately in wall bounded flows, the resolution near the wall has to increase. This makes the use of LES restricted, especially in industrial applications where the near wall layer can be very thin but necessary to capture the nature of the flow. As seen in Section 2.4, there are different ways to approach the near wall problem. The approach presented herein differs from the previous presented here (see Section 2.4) in the sense that it is not only designed to relieve the necessity of number of grid points in the wall normal direction, but also to account for the anisotropy introduced by the wall. This allows the use of coarser meshes in all directions near the wall while maintaining the turbulent characteristics of a resolved field in the whole domain. Instead of using one velocity field to couple RANS and LES, the model attempts to adjust the resolved velocity field with information taken from the averaged velocity field. Therefore making it easier to couple the statistical field with the resolved one. Many sub-grid models assume that the flow contains an inertial subrange and hence the sub-grid motions can be assumed to be isotropic. This is true only if the grid is small enough so that the anisotropy introduced by the mean shear can be neglected. At high
Reynolds numbers, the refinement of the grid becomes too costly, therefore restricting the LES method to low Reynolds numbers flows. As the solid boundary is approached the mean shear becomes high enough to introduce anisotropy. It is therefore desirable that as the wall is approached, there should be a transition towards an ensemble averaged that will account for the decrease of the turbulent scales. Schumann [99] proposed to split the residual stress tensor into two, one "locally isotropic" part and one "inhomogeneous" part. The isotropic part is proportional to the fluctuating strain and does not affect the mean flow equations but determines the rate of energy dissipation. The inhomogeneous part is proportional to the mean strain and controls the shear stress and mean velocity profile:

$$\tau_{ij}^{r} - \frac{2}{3} \tau_{kk} \delta_{ij} = -2\nu_r (\langle S_{ij} \rangle - \langle \langle S_{ij} \rangle \rangle) - 2\nu_a \langle S_{ij} \rangle$$ (10.1)

where $\langle \cdot \rangle$ denotes ensemble averaged of the filtered equations. The viscosities $\nu_r$ and $\nu_a$ are based on fluctuating and mean strains. The isotropic part of the residual stress tensor has a zero time mean value. By refining the grid the residual stresses must tend to zero, therefore the inhomogeneous part must have a grid dependence parameter in the turbulent viscosity $\nu_a$. Schumann [99] used a mixing length model for $\nu_a$ with the length scale computed as $L = \min(\kappa y, C_{10} \Delta)$, where $C_{10}$ is a constant that is difficult to prescribe for all types of flows. Schumann [99] and Grotzbach and Schumann [45] tried to derive a theoretical value for the constant but they were forced to introduced corrective constants to agree with a range of experiments. Moin and Kim [88] used the same principle of splitting the residual stress but in the mixing length model, they use the spanwise size of the cell as the length scale. They argue that for the near wall region in a channel flow, the important structures are streaks that are finely spaced on the spanwise direction. Therefore a coarse resolution in the spanwise direction would lead to larger eddies and a thicker viscous sublayer. Sullivan et al. [105] developed a similar approach for planetary boundary layer flows but chose $\nu_a$ to match the Monin-Obukhov similarity theory [18]. Baggett [9] used a similar approach to compare two hybrid models, one "Schumann-like" and one "DES-like" but found excessive streamwise fluctuations leading to streaks that were much too large.

In the context of hybrid LES-RANS, a blending function can be used to introduce a smooth transition between the resolved and the ensemble averaged turbulence parts. In the
The present study the total residual stress is written as:

\[
\tau_{ij}^r - \frac{2}{3} \tau_{kk} \delta_{ij} = -2 \nu_r f_b (\bar{S}_{ij} - \langle \bar{S}_{ij} \rangle) - 2(1 - f_b) \nu_a \langle \bar{S}_{ij} \rangle
\]  

(10.2)

In this way the averaged stress would be:

\[
\langle \tau_{ij}^r - \frac{2}{3} \tau_{kk} \delta_{ij} \rangle = 2(1 - f_b) \nu_a \langle \bar{S}_{ij} \rangle
\]  

(10.3)

which is just the RANS stress. This way the total stress would be \(2(1 - f_b) \nu_a \langle \bar{S}_{ij} \rangle + \langle u'v' \rangle\).

It is therefore necessary that the blending function \(f_b\) tends to one in the region where \(\langle u'v' \rangle\) is resolved correctly and to zero in the region near the wall where the shear stress is under resolved due to the coarse grid. The total rate of transfer of energy from the filtered motions to the residual scales is given by (assuming that \(\langle \nu_r S_{ij} S_{ij} \rangle \approx \nu_r \langle S_{ij} S_{ij} \rangle\) [84]):

\[
- \langle \tau_{ij} S_{ij} \rangle = 2 \nu_r f_b (\bar{S}_{ij} - \langle \bar{S}_{ij} \rangle) \bar{S}_{ij} + 2(1 - f_b) \nu_a \langle \bar{S}_{ij} \rangle \langle S_{ij} \rangle
\]

\[
= 2 f_b \nu_r (\langle \bar{S}_{ij} \rangle \bar{S}_{ij} - \langle S_{ij} \rangle \langle S_{ij} \rangle) + 2(1 - f_b) \nu_a \langle \bar{S}_{ij} \rangle \langle S_{ij} \rangle
\]  

(10.4)

which shows how the RANS viscosity contributes to dissipation in association with the mean velocity only, i.e. the resolved turbulent stresses are free to develop independently from the RANS viscosity.

## 10.2 The RANS Model

Since the model is based on the idea of including the wall effects via the averaged field, a model that can be integrated all the way down to the wall must be used. In this study, the \(\varphi - f\) model presented in Chapter 4 was chosen, since it has been proved robust and accurate.

The \(\varphi - f\) model represents the near wall effects via elliptic relaxation and without the need for ad-hoc damping functions. The inhomogeneous turbulent viscosity is represented by:

\[
\nu_a = C_\mu \varphi k T
\]  

(10.5)

where \(T\) is the timescale given by equation (3.31). Another advantage of using the \(\varphi - f\) model is that the length scale near the wall can be easily computed as:

\[
L_t = \frac{k^{3/2}}{\varepsilon}
\]  

(10.6)

which corresponds to the integral length scale in the wall normal direction. In Figure 10.1 three different length scales are plotted using DNS data [65].
Chapter 10. Hybrid modelling using the $\varphi - f$ model

10.3 The LES Model

For the isotropic viscosity $\nu_r$, Schumann [99] used a model based on the sub-grid energy. Moin and Kim [88] used the standard Smagorinsky [100] model based on the fluctuating strain. Here the later approach is used:

$$\nu_r = (C_s \Delta)^2 \sqrt{2 s'_{ij} s'_{ij}}$$  \hspace{1cm} (10.7)

$$s'_{ij} = S_{ij} - \langle S_{ij} \rangle$$  \hspace{1cm} (10.8)

In the frame of unstructured codes, the filter width is taken as twice the cell volume ($\Delta = 2Vol$).

10.4 The blending function $f_b$

The blending function has been parametrised by the ratio of the turbulent length scale to the filter width:

$$f_b = \tanh \left( C_l \frac{L_t}{\Delta} \right)^n$$  \hspace{1cm} (10.9)

Here $C_l = 1$ and $n = 1.5$ are empirical constants. These values were chosen to match the shear stress profile based on channel flow results at $Re_\tau = 395$ with DNS data. The function
10.5 The averaging procedure

In (10.2) the averaged velocity can be calculated as a standard time averaged or a running average. Although it is possible also to have plane averaged in the case of the channel flow, this was not done in order to keep the formulation useful for 3D flows where no plane averaging is possible. The running averaged is calculated as:

\[
\langle U \rangle^{n+1} = \alpha U^{n+1} + (1 - \alpha) \langle U \rangle^n
\]  

(10.10)
where $\alpha$ is a constant designed to impose the size of the averaging window. The time averaged is calculated as:

$$\langle U \rangle = \frac{1}{N} \sum_{1}^{N} u$$

(10.11)

where $N$ is the number of samples taken. A typical history plot of the velocity at a point in the middle of the channel flow can be seen in Figure [10.4]. Here the two types of averaging can be seen, along with the instantaneous velocity. The running average is calculated using a

$$\langle U \rangle = \frac{1}{N} \sum_{1}^{N} u$$

(10.11)
window of ten seconds (about 2000 time steps, or $10\delta/u_τ$), making it respond more quickly to changes. The time average is calculated once the flow has converged. Tests were carried out using both types of averaging in equation (10.2) and there was no difference in the final averaged profiles. The only difference was the CPU time for the calculation, with the time average taking longer. Varying the size of the window does not seem to have an influence as long as it is much larger than the eddy turnover time ($h/u_τ$). In different tests carried out, once the window size was lower than 2 times the eddy turnover time, the moving average velocity has significant fluctuations and therefore the mean strain becomes larger in the region far from the wall. For all the results presented here, the window size has been set to approximately 10 times the eddy turnover time.

10.6 Channel Flow Results

Channel flow computations have been carried out at different Reynolds numbers on the same domain, a box of dimensions [0,6.4],[0,2],[0,3.2]. Three different grids have been used in order to have the first cell centre at approximately $y^+ = 1$ for $Re_τ =395$, 1100 and 4000 respectively. The Reynolds numbers (based on friction velocity) that have been considered are 395, 590, 1100, 2000 and 4000 (See Table 10.1).

10.6.1 Channel flow at $Re_τ = 395$

Mean profiles

In Figures 10.5 and 10.6 the results of the model are shown compared with the DNS of Kim et al. [65] and with the results of a standard Smagorinsky LES with the same mesh (case C1). The additive constant of the logarithmic layer is corrected by the hybrid model whereas for a standard LES on the same mesh is overpredicted. The resolved normal stresses can be seen in Figure 10.7. The coarse LES over prediction of the streamwise normal stress ($\langle u'u' \rangle$) is corrected by the hybrid model. In comparison, the LES results overpredict it by a factor of 1.5 approximately. For the other stresses, the improvement is not as obvious. The spanwise stress ($\langle w'w' \rangle$) has a higher peak but still underpredicted in log-layer. The normal
stress ($\langle v'v' \rangle$) is underpredicted in all the domain, almost as bad as in the coarse LES. This implies that the resolution in the wall normal direction is still too coarse to capture the correct damping produced by pressure fluctuations.

### Structures

Although the idea of the model is not to reproduce the structures of a fine LES, some improvements can be observed when compared to a coarse LES. The two point correlations of streamwise velocity ($U$) in the streamwise ($x$) and spanwise ($z$) directions

\[
R_{11}(x, t) = \langle U(x + r, t)U(x) \rangle \quad R_{11}(z, t) = \langle U(z + r, t)U(z) \rangle
\]  

are plotted in Figures [10.8] and [10.9] at $y^+ = 5$. It can be seen how the instantaneous streamwise velocity fluctuations are more realistic that the standard LES. The streamwise correlation provides a measure of how long the streamwise streaks are. For the coarse LES the streaks are extremely long, they are longer than the domain. The lowest value of the correlation is about 30%. The hybrid model reproduces streaks that are still larger than the real ones but much smaller than the LES. The integral length scale computed from the two point correlation (area under the curve in Figure [10.8]) is 1.34 times larger for the LES than for the hybrid model.
Figure 10.5: Velocity profile for case C1. $Re = 395$

Figure 10.6: Shear stress for case C1. $Re = 395$

model. Although the streaks are shorter with the hybrid model, the smallest structures still depend on the grid size. This can be seen from Figure 10.9 where the spanwise correlation is shown at $y^+ = 5$. The smallest structures are still only two points wide due to the coarse resolution in the spanwise direction.

The instantaneous streamwise velocity contours at $y^+ = 5$ are shown in Figure 10.10 for LES and in Figure 10.11 for the hybrid model (C1). From these figures it can be seen that the standard LES has long streaks and weak fluctuations. Both simulations seem to have streaks
that are the same width on average, this is because they are limited by the mesh resolution in the spanwise direction.

**Mesh refinement**

The blending function allows for mesh refinement, so as $\Delta$ becomes smaller, the function reaches a value of 1 closer to the wall. This is in accordance with the idea that as the mesh is refined, the resolved structures are smaller and closer to an isotropic state. The blending function for all meshes used in this study are shown in Figure 10.12 always with the same
10.6. Channel Flow Results

Figure 10.10: Contours of $u'$ at $y^+ = 5$. Figure 10.11: Contours of $u'$ at $y^+ = 5$. Hybrid model, case C1.

coefficients. In Figure 10.13 the velocity profiles for cases C1, C2 and C3 are shown. The shear stresses profiles are plotted in Figure 10.14 where it can be seen how the modelled contribution reduces as the mesh is refined, but since the resolved contribution increases, the total stress remains almost constant. The effect of the mesh refinement can be seen in Figures 10.15 and 10.16 where the two point correlations are plotted. By refining the mesh, smaller structures are obtained. Another effect of the mesh refinement can be seen in Figure 10.17 where the normal stresses are plotted for cases C1 and C3. Although $\langle u'u' \rangle$ and $\langle w'w' \rangle$ are not greatly affected, the normal stress $\langle v'v' \rangle$ is closer to the DNS data. This indicates that the
resolution in the wall normal direction might not be fine enough to get the correct pressure fluctuations therefore damping the normal stress.

\subsection{Higher Reynolds numbers}

For the higher Reynolds number cases, the results of mean velocity profiles follow the log-law (see Figure 10.18). The model seems able to take into account the anisotropy introduced
10.6. Channel Flow Results

Figure 10.15: $R_{11}$ streamwise correlation. $y^+ = 5$, cases C1 and C3

Figure 10.16: $R_{11}$ spanwise correlation. $y^+ = 5$, cases C1 and C3

Figure 10.17: Normal stresses for cases C1 and C3

by the wall for different Reynolds numbers and different grids.

For $Re_\tau = 2000$ DNS data is available [54] and some comparison can be made at this much higher Reynolds number. The normal stresses are shown in Figure 10.19 and the shear stress in Figure 10.20 for case C8 (i.e. 100000 cells). At this much higher Reynolds number the streamwise component is slightly underpredicted as opposed as what would be expected from a coarse LES but the wall normal stress is not in good agreement near the wall. Streamwise instantaneous velocity for $Re_\tau$ 2000 and 4000 at $y^+ = 1$ are shown in Figures 10.21 and 10.22. It can be seen that the streaks are not excessively long.
Chapter 10. Hybrid modelling using the $\varphi - f$ model

Figure 10.18: Velocity profiles for cases C1, C4, C6, C8, and C10

Figure 10.19: Normal stresses for case C8.

Figure 10.20: Shear stress for case C8
Figure 10.21: Instantaneous streamwise velocity contours at $y^+ = 1$ for case C8.

Figure 10.22: Instantaneous streamwise velocity contours at $y^+ = 1$ for case C10.
Chapter 11

Conclusions and recommendations for future work

11.1 Final remarks

In the present work, the effects of the wall on turbulence have been studied from the industrial point of view, based on the finite-volume method designed for unstructured grids with a segregated solver. The aim of the work has been to improve the existing treatment of wall bounded flows by looking at three different types of solutions in the near wall region. A simple modification to the standard wall function approach, the scalable wall function, has been implemented and tested, showing to be independent of the mesh refinement. Focusing on the resolution of the near wall region, a new model has been derived to implement a robust formulation of the $v^2 - f$ model. The original $v^2 - f$ model uses the basis of elliptic relaxation in an eddy viscosity model. This makes it attractive from the industrial point of view, but the stiffness of the boundary condition translates into the requirement of using a coupled solver, which is barely used in industrial software. The new model, called $\varphi - f$, has the advantage that it can be implemented in a segregated solver while retaining most of the qualities of the original model. Several other models have been also implemented in Code_Saturne an industrial code, in order to test the new model and compare it with models that have similar characteristics in terms of CPU requirements and robustness. The models have been tested in different two-dimensional cases and their capability to reproduce experimental, LES or
DNS data has been assessed. To improve 3D industrial simulations, a new approach to hybrid RANS-LES has been derived using the $\varphi - f$ model as this RANS model was shown to give good results in two-dimensional studies. The hybrid approach correctly takes into account the effect of the mean shear produced by the wall and blends it with an existing subgrid scale model (i.e. the Smagorinsky model). The hybrid model has been tested in channel flow with grids too coarse for a standard LES computation and at high Reynolds numbers.

Although in every chapter the performance of the models has been analysed, some important conclusions are summarised below.

**Fully developed channel flow**

In a plane channel flow, the wall normal velocity fluctuations are suppressed creating an increase in the pressure fluctuations. This effect can be modelled by using the elliptic relaxation method of Durbin [33] but the original $\overline{v^2} - f$ model, when solved segregated, requires a timestep 10000 times smaller than other eddy viscosity models which makes it unusable in industry. On the contrary, the $\varphi - f$ model developed in this thesis allows a segregated solver to produce the same benefits from the original model. The time step required is the same as for the two equation models and the results are similar to the original $\overline{v^2} - f$ model. The $\varphi - f$ model also proved to have a closer performance to the original $\overline{v^2} - f$ model than the other Code friendly variation, LDM. The results for the other models implemented, $k - \omega$, SST and $k - \varepsilon$ Launder-Sharma agree well with the results from literature indicating a correct implementation. These last three models provide good results in terms of mean velocity profiles but the turbulent variables are not as satisfactory as with the elliptic relaxation models.

The use of a scalable wall function circumvents the problem of the restriction on the distance of the first computational point off the wall. The implementation of the SSG model has been also tested and the velocity profiles and Reynolds stresses results are in accordance with the literature. The main advantage of this approach being that it is able to predict the spanwise and the wall normal stress difference.
Turbulent heated cavity

The low Reynolds number characteristics of the flow make it a good case for testing the models in the near wall region. Here the eddy viscosity models give good predictions for the temperature profiles in all sections of the cavity. In terms of velocity profiles, the $\varphi - f$ model is clearly superior compared to the LDM. The $k - \omega$ and the SST models have similar behaviour but are not as accurate as the $\varphi - f$ model. This is mainly due to good resolution of the turbulence properties in the near wall region and the ability to mimic the kinematic blocking effects that arise from the solid boundaries.

Plane asymmetric diffuser

In the flow through an asymmetric diffuser, the results are highly model dependent. The smooth separation is predicted too early by the SST model and too late by the $k - \omega$ model, which indicates that the difference may come from the empiricism in the viscosity limiter of the SST model. The elliptic relaxation models have a closer separation point compared with the experiments indicating that the shear stress is correctly predicted. The SSG model with scalable wall function has a spurious behaviour in the recirculation region, with a double inflection point near the inclined wall. This problem has been noted in other second moment closures. In terms of pressure coefficient, the $\varphi - f$ provides the best prediction among the eddy viscosity models with the SSG model being the best one. Overall in the separated region, the $\varphi - f$ velocity predictions are closer to the experimental data, but all models fail to recover fast enough after reattachment.

Flow over periodic hills

The flow over periodic hills represents a difficult test case for linear eddy viscosity models since the stress transport plays an important role. The $\omega$ based models give a larger recirculation compared to the highly resolved LES, with the SST model having an excessively late reattachment point. The effect of the viscosity limiter seems to increase the separated region as it did in the diffuser case. The $k - \varepsilon$ Launder-Sharma model gives a too small recirculation with late separation and early reattachment. The introduction of the elliptic relaxation
improves the $k-\varepsilon$ model results by giving a recirculation bubble of length similar to the LES but with a later separation and reattachment. The models are not able to capture the correct behaviour at the top of the hill, and therefore the separation angle is too low. Convergence with the original $\nu^2 - f$ model is obtained in this case but only with a extremely low CFL number. The use of SSG model and scalable wall function produces similar behaviour as in the diffuser case, namely shorter recirculation and a double inflection point near the wall in the separated region. This implies that the near wall effects need to be modelled into the second moment closure taking into account the physics of that region.

Hybrid model

The hybrid model presented in this work based on LES and the $\phi-f$ model, gives results for a channel flow where predictions are obtained at different Reynolds numbers up to $Re_\tau = 4000$ with less than 330000 cells. The model takes into account the effect of the wall in the mean flow by inserting the mean shear into the instantaneous equations. It blends both effects into one momentum equation. The model resolves the filtered velocity field in all the domain which is different from other approaches like DES where there is still debate whether the momentum equations are filtered or averaged. The formulation proposed does not present the "kink" or acceleration in the velocity profile observed in other hybrid approaches and it does not require any type of artificial turbulence addition. The effects of the wall are introduced into the formulation using the $\phi-f$ turbulence model which is solved in the code with the averaged velocity calculated from the instantaneous field. In this way the coupling between both velocity fields is straightforward. The use of a blending function is introduced to avoid double counting of the stresses and to ensure that as the mesh is refined the RANS contribution tends to zero. The accuracy of the formulation is not very sensitive to the blending function, at least for the channel flow calculations. Another advantage of the formulation is that it allows to have fluctuations in the whole domain. The structures are still limited by the mesh size. For the cases treated here the meshes are very coarse, yet all the Reynolds stresses have realistic values in view of the mesh quality. This is believed to result from the formulation which removes any influence of the RANS viscosity on the resolved dissipation.
11.2 Recommendations for future work

Several recommendations arise naturally from the conclusions. The \( \phi - f \) model has been proven to behave reasonably well in two dimensional attached and separated flows. Although it is not derived to model the anisotropy, it should be tested in three dimensions (some preliminary cases have been considered with unstructured meshes but found to be inconclusive, see [14] and [3]). Flows with complex geometries in which the wall has a strong influence are good candidates for this model. Improvements proposed for the original \( v^2 - f \) can also be tested using the \( \phi - f \) model. This includes using a rescaled \( f \) equation [74] or using a non-linear constitutive relation [91]. The hybrid model is probably where most of the future work should be focused. This would make the method suitable to predict the unsteady physics of the flow reasonably well without the high cost of CPU required by LES for wall bounded flows. Although the model presented here has performed well in channel flows, it is necessary to test it on more complex flows. This will require more computational resources. Since computer technology development is making powerful computers available at reasonable price, the future of the model can be considered effective for industrial applications. In order to make the model less empirical, a blending function based on other parameters has to be defined. It would be useful to be able to express the function in terms of some parameter such as the ratio of LES to RANS turbulent kinetic energy strain or dissipation that will test if the LES field is sufficiently developed to switch from RANS to LES.
Bibliography


[40] S.E. Gant. *Development and Application of a New Wall Function for Complex Turbu-


[43] M Gibson and B. Launder. Ground effects on the pressure fluctuations in the atmo-


[48] F. Hamba. A hybrid RANS LES simulation of turbulent flows. *Theoretical and com-


