The computation of turbulent natural convection flows

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

2011

By
Seyed Ali Omranian
School of Mechanical, Aerospace and Civil Engineering
Contents

List of Tables 
List of Figures 
Abstract 
Declaration 
Copyright 
Acknowledgements 

1 Introduction 
  1.1 Buoyant flows 
  1.2 Turbulent flow 
  1.2.1 Characteristics 
  1.2.2 Turbulence length scale 
  1.2.3 Turbulent buoyancy-driven flow 
  1.3 Turbulence modelling 
  1.4 Objectives 
  1.5 Outline of thesis 

2 Literature Review 
  2.1 Experimental Studies 
  2.2 Numerical Studies 

3 Turbulence modelling 
  3.1 Introductory Remarks 

2
3.2 Mean Flow Equations and their solution .......................... 70
3.3 RANS Turbulence models ............................................. 72
  3.3.1 High-Reynolds Number $k$-$\varepsilon$ Model .................. 72
  3.3.2 Further modification in High-Re $k$-$\varepsilon$ model for 3D simulations .................................................. 74
  3.3.3 Low-Reynolds Number $k$-$\varepsilon$ model ................. 75
  3.3.4 Approximation of turbulent heat flux $u_i\theta$ ............... 76
  3.3.5 Low-Reynolds-Number Three and Four Equations model $k$-$\varepsilon$-$\theta^2$ and $k$-$\varepsilon$-$\theta^2$-$\varepsilon$ .......................... 78
  3.3.6 High-Reynolds-Number Three and Four Equations model $k$-$\varepsilon$-$\theta^2$ and $k$-$\varepsilon$-$\theta^2$-$\varepsilon$ .......................... 80
  3.3.7 Differential Stress Closure ...................................... 80
  3.3.8 Differential Stress Closure Used with $\theta^2$ and $\varepsilon$-$\theta$ Transport Equations ........................................... 91

4 Wall Functions ......................................................... 92
  4.1 Conventional Wall Functions ..................................... 92
    4.1.1 Simple Hydraulic Wall Function ......................... 92
    4.1.2 Standard Wall Function ................................. 95
  4.2 Analytical Wall Functions ...................................... 98
    4.2.1 Derivation of Analytical Wall Functions ............ 98
    4.2.2 Thermal Wall Function When the Near Wall Cell Is Thicker Than the Viscous Sublayer .................. 100
    4.2.3 Thermal Wall Function When the Near Wall Cell Is Thinner Than the Viscous Sublayer .................. 103
    4.2.4 Hydrodynamic Wall Function When the Near Wall Cell Is Thicker Than the Viscous Sublayer .................. 104
    4.2.5 Hydrodynamic Wall Function When Near Wall Cell Is Thinner Than Viscous Sublayer .................. 111
    4.2.6 Convection and pressure gradient terms ............ 112

5 Numerical Implementation .......................................... 116
  5.1 TEAM Code ................................................................ 116
  5.2 STREAM Code ....................................................... 117
  5.3 Approximation of Convective and Diffusive terms .......... 118
5.4 The velocity-pressure linkage scheme .................................. 122
5.5 Sequence of solution steps ............................................... 126
5.6 Time-dependent computations ............................................. 129
5.7 Boundary conditions ......................................................... 130
5.8 Simulation of inclined cavity .............................................. 132
5.9 Implementation of buoyancy related terms in the STREAM code 133
5.10 Closing Remarks ............................................................. 133

6 Inclined Cavity-2D simulations ............................................. 134
   6.1 Grid ........................................................................... 135
   6.2 Test case description ...................................................... 138
   6.3 Overview of 2-dimensional flow in the tall, differentially heated 143
   cavities at different angles of inclination .................................. 143
   6.4 Inclined tall cavity - 60° stable ........................................ 157
      6.4.1 Flow Pattern (including comparison with 90° case) ........ 157
      6.4.2 $k$-$\varepsilon$ predictions .............................................. 161
      6.4.3 RSM predictions ......................................................... 167
      6.4.4 Employment of $\theta^2$ and $\varepsilon_\theta$ transport equations 173
   6.5 Inclined tall cavity - 60° unstable ...................................... 184
      6.5.1 Numerical predictions ............................................... 184
   6.6 Inclined tall cavity - 5° stable .......................................... 191
      6.6.1 Test case description .................................................. 191
      6.6.2 Flow pattern ............................................................ 192
      6.6.3 Comparisons of $k$-$\varepsilon$ using LRN, AWF and SWF with LES 194
      data within the inclined 5° stable tall cavity .............................. 194
   6.7 Closing remarks ............................................................ 199

7 Inclined Cavity-3D time-dependent simulations .......................... 200
   7.1 Validation of STREAM ..................................................... 201
   7.2 3D time dependent simulation of inclined tall cavity - 15° unstable210
      7.2.1 Test case description .................................................. 210
      7.2.2 Flow Pattern ............................................................ 211
      7.2.3 Grid for analytical wall functions .................................. 212
      7.2.4 3D time dependent simulation by $k$-$\varepsilon$-AWF .............. 215
      7.2.5 Time-averaged data by $k$-$\varepsilon$-AWF .......................... 222
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.2.6</td>
<td>FFT analysis</td>
<td>244</td>
</tr>
<tr>
<td>7.2.7</td>
<td>3D time dependent simulation by RSM</td>
<td>250</td>
</tr>
<tr>
<td>7.2.8</td>
<td>Time-averaged data by RSM</td>
<td>257</td>
</tr>
<tr>
<td>7.2.9</td>
<td>FFT analysis-RSM</td>
<td>274</td>
</tr>
<tr>
<td>7.3</td>
<td>3D time-dependent simulation of inclined tall cavity - 15° stable</td>
<td>280</td>
</tr>
<tr>
<td>7.3.1</td>
<td>Flow Pattern</td>
<td>281</td>
</tr>
<tr>
<td>7.3.2</td>
<td>Simulation by $k$-$\varepsilon$-AWF</td>
<td>282</td>
</tr>
<tr>
<td>7.3.3</td>
<td>Simulation by $k$-$\varepsilon$-AWF with turbulent viscosity limitation</td>
<td>298</td>
</tr>
<tr>
<td>7.3.4</td>
<td>Simulation by RSM-Basic</td>
<td>316</td>
</tr>
<tr>
<td>7.4</td>
<td>Closing remarks</td>
<td>337</td>
</tr>
<tr>
<td>8</td>
<td>Annular horizontal penetration</td>
<td>339</td>
</tr>
<tr>
<td>8.1</td>
<td>Test case description</td>
<td>340</td>
</tr>
<tr>
<td>8.2</td>
<td>Grid</td>
<td>341</td>
</tr>
<tr>
<td>8.3</td>
<td>Steady-state simulation ($Ra = 4.5 \times 10^{8}$)</td>
<td>344</td>
</tr>
<tr>
<td>8.4</td>
<td>Time-dependent simulation ($Ra = 4.5 \times 10^{8}$)</td>
<td>361</td>
</tr>
<tr>
<td>8.5</td>
<td>Steady-state simulation ($Ra = 3.1 \times 10^{13}$)</td>
<td>379</td>
</tr>
<tr>
<td>8.6</td>
<td>Time-dependent simulation ($Ra = 3.1 \times 10^{13}$)</td>
<td>392</td>
</tr>
<tr>
<td>8.7</td>
<td>Closing remarks</td>
<td>411</td>
</tr>
<tr>
<td>9</td>
<td>Conclusions and Future Works</td>
<td>413</td>
</tr>
<tr>
<td>9.1</td>
<td>Conclusions</td>
<td>413</td>
</tr>
<tr>
<td>9.2</td>
<td>Suggested Future Work</td>
<td>425</td>
</tr>
<tr>
<td>A</td>
<td>Grid sensitivity - 2D test cases</td>
<td>436</td>
</tr>
<tr>
<td></td>
<td>Word count: 95277</td>
<td></td>
</tr>
</tbody>
</table>
List of Tables
2.1

Correlations Nu-Ra for Aspect ratio = 0.75 and 1.5 at different
angles of inclination and 103 < Ra < 108 [1]. . . . . . . . . . . . .

59

3.1
3.2

Values of constants in High-Re k-ε, model. . . . . . . . . . . . .
the coefficients for the k-ε-θ2 -εθ turbulence model. . . . . . . . .

74
79

3.3

the coefficients for the high-Reynolds-number k-ε-θ2 -εθ turbulence model. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .

80

The coefficients required in the basic Reynolds-stress closure. .

87

3.4

6


### List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Comparison of experimental measurements and correlations[2]</td>
<td>50</td>
</tr>
<tr>
<td>2.2</td>
<td>Schematics of the flow geometry, (a) flat plate, (b) backward-facing step, (c) forward-facing step.</td>
<td>51</td>
</tr>
<tr>
<td>2.3</td>
<td>Velocity profiles under aiding and opposing buoyancy force[3].</td>
<td>52</td>
</tr>
<tr>
<td>2.4</td>
<td>Effect of aiding natural convection on a turbulent flow in a vertical tube. $u_{e}$ is the velocity at the border of the viscous layer, $u_{m}$ is the mean velocity in the core of the flow.[3].</td>
<td>52</td>
</tr>
<tr>
<td>2.5</td>
<td>Heat transfer data and correlations for the Rayleigh-Bénard configuration[4].</td>
<td>53</td>
</tr>
<tr>
<td>2.6</td>
<td>Schematic view of the air filled cavity[5].</td>
<td>56</td>
</tr>
<tr>
<td>2.7</td>
<td>Boundary layer structure at Y=0.5 (vertical velocity profile near the hot wall)[6].</td>
<td>57</td>
</tr>
<tr>
<td>2.8</td>
<td>Vector plot of mean velocity profiles (a) Non-partitioned cavity (b) Partitioned cavity[7].</td>
<td>58</td>
</tr>
<tr>
<td>4.1</td>
<td>A low-Reynolds-number and wall function grids.</td>
<td>93</td>
</tr>
<tr>
<td>4.2</td>
<td>Turbulent velocity boundary layer.</td>
<td>93</td>
</tr>
<tr>
<td>4.3</td>
<td>Schematic of near-wall cell.</td>
<td>102</td>
</tr>
<tr>
<td>5.1</td>
<td>Grid and control volume arrangements for pressure and mean velocity storage in the TEAM code.</td>
<td>117</td>
</tr>
<tr>
<td>5.2</td>
<td>Schematic of control volumes.</td>
<td>119</td>
</tr>
<tr>
<td>5.3</td>
<td>Calculation procedure</td>
<td>127</td>
</tr>
<tr>
<td>5.4</td>
<td>Schematic of the inclined cavity.</td>
<td>132</td>
</tr>
<tr>
<td>6.1</td>
<td>100x270 Grid for low-Reynolds-number-model computation of the tall cavity, (scale of the small side is enlarged by a factor of 7).</td>
<td>136</td>
</tr>
<tr>
<td>6.2</td>
<td>40x220 Grid for high-Reynolds-number-model computation of the tall cavity, (scale of the small side is enlarged by a factor of 7).</td>
<td>136</td>
</tr>
</tbody>
</table>
6.3 80x180 Grid for low-Re-number computation of the inclined tall cavity at 5° stable, (scale of the small side is enlarged by a factor of 7). .............................................................. 138
6.4 40x180 Grid for high-Re-number computation of the inclined tall cavity at 5° stable, (scale of the small side is enlarged by a factor of 7). .............................................................. 138
6.5 Schematic of inclined tall cavity and its thermal boundary condition. .............................................................. 139
6.6 V and $V_{rms}$ profiles at, and off, the centreline of the 60° stable cavity, to illustrate the two-dimensionality of the flow, Cooper et al. [8] .............................................................. 140
6.7 Velocity profile in the 60° stable cavity at the centreline where $Y=0.95$, Cooper et al. [8] .............................................................. 141
6.8 Temperature profiles at, and off, the centreline of the 60° unstable cavity, to illustrate the two-dimensionality of the flow, Cooper et al. [8] .............................................................. 141
6.9 Wall-parallel velocity profiles in the 15° stable cavity along the spanwise direction where $X=0.5$, to illustrate the three-dimensionality of the flow, Cooper et al. [8] .............................................................. 142
6.10 Wall-normal velocity profiles in the 15° unstable cavity along the spanwise direction where $X=0.5$, to illustrate the three-dimensionality of the flow, Cooper et al. [8] .............................................................. 143
6.11 Temperature and stream lines within a 0° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .............................................................. 144
6.12 Temperature and stream lines within a 5° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .............................................................. 145
6.13 Temperature and stream lines within a 10° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .............................................................. 146
6.14 Temperature and stream lines within a 15° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .............................................................. 147

8
6.15 Temperature and stream lines within a 60° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 148

6.16 Temperature and stream lines within a 90° vertical cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 149

6.17 Temperature and stream lines within a 10° unstable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 150

6.18 Temperature and stream lines within a 5° unstable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 151

6.19 Temperature and stream lines within a 0° unstable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 151

6.20 Mean velocity and $k$ profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k$-$\varepsilon$-LRN. ................. 153

6.21 Mean velocity and $k$ profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k$-$\varepsilon$-AWF. ................. 154

6.22 Mean velocity and $k$ profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k$-$\varepsilon$-SWF. ................. 155

6.23 Mean velocity and $k$ profiles at mid-height of the tall cavity at different angles of inclinations resulting from RSM-Basic-AWF. ......... 156

6.24 $k$ profiles along the section at $X=0.5$ at different angle of inclinations resulting from $k$-$\varepsilon$ using LRN, AWF, SWF and RSM-Basic using AWF. ................................................................. 157

6.25 Temperature and turbulent kinetic energy contours in tall cavity resulting from $k$-$\varepsilon$ using LRN ($Ra = 0.8 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 158

6.26 Mean velocity vector plot and stream lines in tall cavity resulting from $k$-$\varepsilon$ using LRN ($Ra = 0.8 \times 10^6$), (scale of the small side is enlarged by a factor of 7). .................................................. 159
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.27</td>
<td>Temperature contours and stream traces within vertical and inclined 60° stable tall cavity resulting from k-$\varepsilon$ using LRN, (scale of the small side is enlarged by a factor of 7).</td>
<td>160</td>
</tr>
<tr>
<td>6.28</td>
<td>Temperature distributions inside vertical and 60° stable inclined tall cavities resulting from k-$\varepsilon$ using LRN, AWF and SWF.</td>
<td>162</td>
</tr>
<tr>
<td>6.29</td>
<td>Nu distribution along cold wall of vertical and 60° stable inclined tall cavity resulting from k-$\varepsilon$ using Low-Re-Number and AWF and SWF.</td>
<td>163</td>
</tr>
<tr>
<td>6.30</td>
<td>Nusselt distribution along the cold wall of 60° stable inclined tall cavity resulting from k-$\varepsilon$-AWF.</td>
<td>164</td>
</tr>
<tr>
<td>6.31</td>
<td>Mean velocity distributions within a 60° stable inclined tall cavity resulting from k-$\varepsilon$ using LRN, AWF and SWF.</td>
<td>164</td>
</tr>
<tr>
<td>6.32</td>
<td>rms velocity fluctuations distributions within vertical and tilted 60° stable tall cavity resulting from k-$\varepsilon$ using LRN, AWF and SWF.</td>
<td>167</td>
</tr>
<tr>
<td>6.33</td>
<td>Temperature distributions resulting from RSM-basic(AWF).</td>
<td>169</td>
</tr>
<tr>
<td>6.34</td>
<td>Temperature distributions resulting from RSM-TCL(AWF).</td>
<td>170</td>
</tr>
<tr>
<td>6.35</td>
<td>Shear stress distributions resulting from RSM-basic(AWF).</td>
<td>171</td>
</tr>
<tr>
<td>6.36</td>
<td>Shear stress distributions resulting from RSM-TCL(AWF).</td>
<td>172</td>
</tr>
<tr>
<td>6.37</td>
<td>Nusselt distribution along the cold wall of 60° stable inclined tall cavity resulting from RSM.</td>
<td>173</td>
</tr>
<tr>
<td>6.38</td>
<td>Temperature distributions resulting from k-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$ - AWF.</td>
<td>175</td>
</tr>
<tr>
<td>6.39</td>
<td>Temperature distributions resulting from RSM-$\theta^2$-$\varepsilon_\theta$ - Basic and TCL.</td>
<td>176</td>
</tr>
<tr>
<td>6.40</td>
<td>Mean velocity distributions within a 60° stable inclined tall cavity resulting from RSM-Basic and RSM-$\theta^2$-$\varepsilon_\theta$ - Basic.</td>
<td>177</td>
</tr>
<tr>
<td>6.41</td>
<td>Mean velocity distributions within a 60° stable inclined tall cavity resulting from RSM-TCL and RSM-$\theta^2$-$\varepsilon_\theta$ - TCL.</td>
<td>177</td>
</tr>
<tr>
<td>6.42</td>
<td>rms velocity fluctuation distributions within a 60° stable inclined tall cavity resulting from RSM-Basic and RSM-$\theta^2$-$\varepsilon_\theta$ - Basic.</td>
<td>178</td>
</tr>
<tr>
<td>6.43</td>
<td>rms velocity fluctuation distributions within a 60° stable inclined tall cavity resulting from RSM-TCL and RSM-$\theta^2$-$\varepsilon_\theta$ - TCL.</td>
<td>178</td>
</tr>
<tr>
<td>6.44</td>
<td>Temperature fluctuations within vertical cavity using RSM-TCL(AWF) and 4-equations EVM.</td>
<td>181</td>
</tr>
</tbody>
</table>
6.45 Temperature fluctuations within 60° stable inclined cavity using RSM-basic(AWF) and 4-equations EVM. 182
6.46 Temperature fluctuations within 60° stable inclined cavity using RSM-TCL(AWF) and 4-equations EVM. 183
6.47 Nusselt distribution along the cold wall of 60° stable inclined tall cavity. 184
6.48 Mean temperature distributions within a 60° unstable inclined tall cavity resulting by $k-\varepsilon$ using LRN, AWF and SWF. 187
6.49 Mean temperature distributions within a 60° unstable inclined tall cavity resulting by RSM-Basic. 187
6.50 Mean temperature distributions within a 60° unstable inclined tall cavity resulting by RSM-TCL. 188
6.51 Mean temperature distributions within a 60° unstable inclined tall cavity resulting by $k-\varepsilon-\theta^2-\varepsilon_\theta$-AWF. 188
6.52 Temperature variance distributions within a 60° unstable inclined tall cavity resulting by $k-\varepsilon-\theta^2-\varepsilon_\theta$-AWF. 189
6.53 Mean temperature distributions within a 60° unstable inclined tall cavity resulting by RSM-$\theta^2-\varepsilon_\theta$-Basic. 189
6.54 Temperature variance distributions within a 60° unstable inclined tall cavity resulting by RSM-$\theta^2-\varepsilon_\theta$-Basic. 190
6.55 Stream lines in tall cavity resulting from RSM-Basic, (scale of the small side is enlarged by a factor of 7). 190
6.56 Nusselt distribution along the cold wall of 60° unstable inclined tall cavity resulting from $k-\varepsilon$-AWF and RSM-Basic. 191
6.57 Schematic of 5° stable inclined tall cavity and its thermal boundary condition. 192
6.58 Stream traces and vector plot resulting from Low-Re-Number $k-\varepsilon$, (scale of the small side is enlarged by a factor of 7). 193
6.59 temperature contours resulting from Low-Re-Number $k-\varepsilon$, (scale of the small side is enlarged by a factor of 7). 193
6.60 Stream traces and vector plot resulting from $k-\varepsilon$ - AWF, (scale of the small side is enlarged by a factor of 7). 193
6.61 temperature contours resulting from $k-\varepsilon$ - AWF, (scale of the small side is enlarged by a factor of 7). 194
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.62</td>
<td>Stream traces and vector plot resulting from $k$-$\varepsilon$ - SWF, (scale of the small side is enlarged by a factor of 7).</td>
</tr>
<tr>
<td>6.63</td>
<td>Temperature contours resulting from $k$-$\varepsilon$ - SWF, (scale of the small side is enlarged by a factor of 7).</td>
</tr>
<tr>
<td>6.64</td>
<td>Mean velocity profiles within $5^\circ$ stable tilted tall cavity.</td>
</tr>
<tr>
<td>6.65</td>
<td>Temperature distributions within $5^\circ$ stable tilted tall cavity.</td>
</tr>
<tr>
<td>6.66</td>
<td>Turbulent kinetic energy distributions within $5^\circ$ stable tilted tall cavity.</td>
</tr>
<tr>
<td>6.67</td>
<td>Local Nusselt numbers along hot wall within $5^\circ$ stable tilted tall cavity.</td>
</tr>
<tr>
<td>7.1</td>
<td>Temperature distributions comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.2</td>
<td>Mean velocity distributions comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.3</td>
<td>rms velocity fluctuations comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.4</td>
<td>Temperature distributions comparisons of results from 2D simulation of $15^\circ$ unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.5</td>
<td>Mean velocity distributions comparisons of results from 2D simulation of $15^\circ$ unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.6</td>
<td>rms velocity fluctuations comparisons of results from 2D simulation of $15^\circ$ unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).</td>
</tr>
<tr>
<td>7.7</td>
<td>Temperature distributions comparisons of results from 2D simulation of $15^\circ$ unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-LRN).</td>
</tr>
<tr>
<td>7.8</td>
<td>Mean velocity distributions comparisons of results from 2D simulation of $15^\circ$ unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-LRN).</td>
</tr>
</tbody>
</table>
7.9 rms velocity fluctuations comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-LRN). .................................................. 210
7.10 Inclined cavity with angle 15° unstable. .............................. 211
7.11 Temperature iso-surfaces in tall 15° unstable tilted cavity resulting from $k$-$\varepsilon$ using AWF at $\tau^* = 440$ (Ra= $0.8 \times 10^6$). .......... 212
7.12 Grid within (a) x-y (b) x-z (c) y-z planes of 15° unstable inclined tall cavity resulting (coarse mesh). ................................. 213
7.13 Grid within (a) x-y (b) x-z (c) y-z planes of 15° unstable inclined tall cavity resulting (fine mesh). ...................................... 214
7.14 Temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Z=0.5). 217
7.15 Vector plots within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Z=0.5). .......... 218
7.16 Temperature contours within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Y=0.5). 219
7.17 Vector plots within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Y=0.5). .......... 220
7.18 Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (X=0.5). 221
7.19 Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (X=0.5). 222
7.20 Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, Z=0.5. ................................................................. 227
7.21 3D-Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, Z=0.5. ............................................................ 228
7.22 Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, Z=0.5. ................................................................. 229
7.23 Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, Z=0.5. ................................................................. 230
7.24 Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, Z=0.5. ................................................................. 231
7.25 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, Z=0.5, $\tau^* = 550.232$ ................................................................. 232
7.26 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, Z=0.5, $\tau^* = 550.233$ ................................................................. 233
7.27 Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. .................................................. 234
7.28 Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. .................................................. 235
7.29 Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $X=0.5$. .................................................. 236
7.30 Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $X=0.5$. .................................................. 237
7.31 Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. .................................................. 238
7.32 Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. .................................................. 239
7.33 Nusselt distributions resulting from $k$-$\varepsilon$-AWF. ................. 240
7.34 Time average vector plots within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Y=0.5). .................................................. 241
7.35 Time average temperature contours within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent $k$-$\varepsilon$-AWF (Y=0.5). .................................................. 242
7.36 Time average vector plots and temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (X=0.5). .................................................. 243
7.37 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. .................................................. 245
7.38 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. .................................................. 246
7.39 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. .................................................. 247
7.40 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. .................................................. 248
7.41 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. .................................................. 249
7.42 Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. .................................................. 250
7.43 Temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5)... 252
7.44 Vector plots within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5)... 253
7.45 Temperature contours within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Y=0.5)... 254
7.46 Vector plots within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Y=0.5)... 255
7.47 Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent numerical simulations (X=0.5)... 256
7.47 Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM, X=0.5 (continued)... 257
7.48 Time average temperature distributions resulting from RSM, Z=0.5... 260
7.49 Time average temperature distributions resulting from RSM, Z=0.25... 261
7.50 Time average temperature distributions resulting from RSM, Z=0.75... 262
7.51 3D-Time average mean velocity distributions resulting from RSM, Z=0.5... 263
7.52 Time average mean velocity distributions resulting from RSM, Z=0.5... 264
7.53 Time average mean velocity distributions resulting from RSM, X=0.5... 265
7.54 Time average mean velocity distributions resulting from RSM, X=0.5... 266
7.55 Time average rms velocity fluctuations resulting from RSM, Z=0.5... 267
7.56 Time average rms velocity fluctuations resulting from RSM, Z=0.5... 268
7.57 Time average rms velocity fluctuations resulting from RSM, X=0.5... 269
7.58 Time average rms velocity fluctuations resulting from RSM, X=0.5... 270
7.59 Nusselt distributions resulting from RSM... 270
7.60 Time average vector plots within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent RSM (Y=0.5)... 271
7.61 Time average temperature contours within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent RSM (Y=0.5). ........................................ 272
7.62 Time average vector plots and temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5). ........................................ 273
7.63 Time average vector plots and temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (X=0.5). ........................................ 274

7.64 Power spectral diagram for temperature resulting from RSM, Z=0.5. ......................................................... 275
7.65 Power spectral diagram for temperature resulting from RSM, Z=0.5. ......................................................... 276
7.66 Power spectral diagram for temperature resulting from RSM, Z=0.25. ......................................................... 277
7.67 Power spectral diagram for temperature resulting from RSM, Z=0.25. ......................................................... 278
7.68 Power spectral diagram for temperature resulting from RSM, Z=0.75. ......................................................... 279
7.69 Power spectral diagram for temperature resulting from RSM, Z=0.75. ......................................................... 280
7.70 15° stable inclined cavity. ......................................................... 280
7.71 Temperature iso-surfaces in tall 15° stable tilted cavity resulting from $k-\varepsilon$ using AWF at $Ra = 1.4 \times 10^6$. ............................................... 282
7.72 Temperature distributions resulting from $k-\varepsilon$-AWF, Z=0.5. ......................................................... 286
7.73 Mean velocity distributions resulting from $k-\varepsilon$-AWF, Z=0.5. ......................................................... 287
7.74 Mean velocity distributions resulting from $k-\varepsilon$-AWF, Z=0.5. ......................................................... 288
7.75 rms velocity fluctuations resulting from $k-\varepsilon$-AWF, Z=0.5. ......................................................... 289
7.76 rms velocity fluctuations resulting from $k-\varepsilon$-AWF, Z=0.5. ......................................................... 290
7.77 Temperature distributions resulting from $k-\varepsilon$-AWF, Z=0.25. ......................................................... 290
7.78 Temperature distributions resulting from $k-\varepsilon$-AWF, Z=0.655. ......................................................... 291
7.79 Temperature distributions resulting from $k-\varepsilon$-AWF, Z=0.75. ......................................................... 291
7.80 Temperature distributions resulting from $k-\varepsilon$-AWF, Z=0.875. ......................................................... 292
7.81 Mean velocity distributions resulting from $k-\varepsilon$-AWF, Z=0.625. ......................................................... 292

16
7.82 Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. . . 293
7.83 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.625$. . . . 293
7.84 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. . . . 294
7.85 Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $X=0.5$. . . . 295
7.86 Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $X=0.5$. . . . 296
7.87 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. . . . . 297
7.88 rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. . . . . 298
7.89 Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. . . . . . . . . . . . . . . . . . . . . . . . . . . 300
7.90 Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. . . . . . . . . . . . . . . . . . . . . . . . . . . 301
7.91 Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. . . . . . . . . . . . . . . . . . . . . . . . . . . 302
7.92 Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. . . . . . . . . . . . . . . . . . . . . . . . . . . 303
7.93 Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. . . . . . . . . . . . . . . . . . . . . . . . . . . 304
7.94 Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.25$. . . . . . . . . . . . . . . . . . . . . . . . . . . 305
7.95 Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.655$. . . . . . . . . . . . . . . . . . . . . . . . . . . 305
7.96 Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. . . . . . . . . . . . . . . . . . . . . . . . . . . 306
7.97 Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.875$. . . . . . . . . . . . . . . . . . . . . . . . . . . 307
7.98 Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.625$. . . . . . . . . . . . . . . . . . . . . . . . . . . 308
7.99 Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. . . . . . . . . . . . . . . . . . . . . . . . . . . 309
7.100 Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.625$. . . . . . . . . . . . . . . . . . . . . . . . . . . 310
7.101 Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. . . . . . . . . . . . . . . . . . . . . . . . . . . 311
7.102 Time-averaged mean velocity distributions resulting from $k-\varepsilon$-AWF with limiter, $X=0.5$. ................................................. 312
7.103 Time-averaged mean velocity distributions resulting from $k-\varepsilon$-AWF with limiter, $X=0.5$. ................................................. 313
7.104 Time-averaged rms velocity fluctuations resulting from $k-\varepsilon$-AWF with limiter, $X=0.5$. ................................................. 314
7.105 Time-averaged rms velocity fluctuations resulting from $k-\varepsilon$-AWF with limiter, $X=0.5$. ................................................. 315
7.106 Nusselt distributions resulting from $k-\varepsilon$-AWF with limiter. .... 316
7.107 Time-averaged temperature distributions resulting from RSM, $Z=0.5$. ................................................................. 319
7.108 Time-averaged mean velocity distributions resulting from RSM, $Z=0.5$. ................................................................. 320
7.109 Time-averaged mean velocity distributions resulting from RSM, $Z=0.5$. ................................................................. 321
7.110 Time-averaged rms velocity fluctuations resulting from RSM, $Z=0.5$. ................................................................. 322
7.111 Time-averaged rms velocity fluctuations resulting from RSM, $Z=0.5$. ................................................................. 323
7.112 Time-averaged temperature distributions resulting from RSM, $Z=0.25$. ................................................................. 324
7.113 Time-averaged temperature distributions resulting from RSM, $Z=0.65$. ................................................................. 324
7.114 Time-averaged temperature distributions resulting from RSM, $Z=0.75$. ................................................................. 325
7.115 Time-averaged temperature distributions resulting from RSM, $Z=0.875$. ................................................................. 326
7.116 Time-averaged mean velocity distributions resulting from RSM, $Z=0.625$. ................................................................. 327
7.117 Time-averaged mean velocity distributions resulting from RSM, $Z=0.75$. ................................................................. 328
7.118 Time-averaged rms velocity fluctuations resulting from RSM, $Z=0.625$. ................................................................. 329
7.119 Time-averaged rms velocity fluctuations resulting from RSM,
\( Z=0.75 \). .................................................. 330
7.120 Time-averaged mean velocity distributions resulting from RSM,
\( X=0.5 \). .................................................... 331
7.121 Time-averaged mean velocity distributions resulting from RSM,
\( X=0.5 \). .................................................... 332
7.122 Time-averaged rms velocity fluctuations resulting from RSM,
\( X=0.5 \). .................................................... 333
7.123 Time-averaged rms velocity fluctuations resulting from RSM,
\( X=0.5 \). .................................................... 334
7.124 Nusselt distributions resulting from RSM. ..................... 335
7.125 Vector plots within x-z plane of \( 15^\circ \) stable inclined tall cavity
where \( Y=0.5 \) resulting from RSM. ......................... 335
7.126 Temperature contours within x-z plane of \( 15^\circ \) stable inclined tall
cavity where \( Y=0.5 \) resulting from RSM. ..................... 336
7.127 Temperature contours within x-y plane of \( 15^\circ \) stable inclined tall
cavity resulting from RSM. ................................... 336
7.128 Vector plots and temperature contours within y-z plane of \( 15^\circ 
stable inclined tall cavity resulting from RSM (\( X=0.5 \)). ....... 337

8.1 Test case geometry and thermal boundary conditions. ......... 341
8.2 Grid in y-z plane. ........................................... 342
8.3 Grid in x-y plane. ........................................... 343
8.4 Grid inside the tube. ........................................ 343
8.5 Contour plots of \( U \) and \( V \) within longitudinal, y-z, plane, at \( X=0, \)
at \( Ra = 4.5 \times 10^8 \). ......................................... 349
8.6 Contour plots of \( W \) and \( P^* \) within longitudinal, y-z, plane, at
\( X=0, \) at \( Ra = 4.5 \times 10^8 \). ................................... 350
8.7 Contour plots of \( T \) and \( k \) within longitudinal, y-z, plane, at \( X=0, \)
at \( Ra = 4.5 \times 10^8 \). ......................................... 351
8.8 Contour plots of \( \mu_t \) within longitudinal, y-z, plane, at \( X=0, \) at
\( Ra = 4.5 \times 10^8 \). ......................................... 352
8.9 Contour plots of \( U, V \) and \( W \) within cross-sections normal to the
axis of the penetration, at \( Z=0.23, 0.46, 0.69 \) and \( 0.87 \), at \( Ra = 
4.5 \times 10^8 \). ............................................... 353
8.10 Contour plots of $P^r$, $T$ and $\mu_t$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and $0.87$, at $Ra = 4.5 \times 10^8$. ................................................................. 354

8.11 Contour plots of $U$ and $V$ within cross-sections normal to the axis of the penetration, at $Z=0.88, 0.92$ and $1$, at $Ra = 4.5 \times 10^8$. 355

8.12 Contour plots of $W$ and $P^r$ within cross-sections normal to the axis of the penetration, at $Z=0.88, 0.92$ and $1$, at $Ra = 4.5 \times 10^8$. 356

8.13 Contour plots of $T$ and $\mu_t$ within cross-sections normal to the axis of the penetration, at $Z=0.88, 0.92$ and $1$, at $Ra = 4.5 \times 10^8$. 357

8.14 Contour plots of $V$, $T$ and $P^r$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and $0.5$, at $Ra = 4.5 \times 10^8$. ................................................................. 358

8.15 Contour plots of $k$ and $\mu_t$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and $0.5$, at $Ra = 4.5 \times 10^8$. ................................................................. 359

8.16 Contour plots of $V$ and $\mu_t$ within horizontal planes outside the penetration, at locations $Y=0.22$ (inlet), $Y=-0.11$ and $Y=-0.78$ (outlet), at $Ra = 4.5 \times 10^8$. ................................................................. 360

8.17 Contour plots of $V$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ................................................................. 364

8.18 Contour plots of $V$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ................................................................. 365

8.19 Contour plots of $W$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ................................................................. 366

8.20 Contour plots of $W$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ................................................................. 367

8.21 Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$, at $Ra = 4.5 \times 10^8$. ................................................................. 368

8.22 Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.46$, at $Ra = 4.5 \times 10^8$. ................................................................. 369
8.23 Contour plots of $V, W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$, at $Ra = 4.5 \times 10^8$. ................................. 370

8.24 Contour plots of $V, W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.87$, at $Ra = 4.5 \times 10^8$. ................................. 371

8.25 Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$ and 0.46, at $Ra = 4.5 \times 10^8$. 372

8.26 Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$ and 0.87, at $Ra = 4.5 \times 10^8$. 373

8.27 Contour plots of time-averaged and steady-state $V$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ......................... 374

8.28 Contour plots of time-averaged and steady-state $W$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. ......................... 375

8.29 Contour plots of time-averaged and steady-state $V$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and 0.87, at $Ra = 4.5 \times 10^8$. ......................... 376

8.30 Contour plots of time-averaged and steady-state $W$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and 0.87, at $Ra = 4.5 \times 10^8$. ......................... 377

8.31 Contour plots of steady-state, time-averaged and total $k$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and 0.87, at $Ra = 4.5 \times 10^8$. ......................... 378

8.32 Contour plots of $U$ and $V$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. ................................. 382

8.33 Contour plots of $W$ and $P^\prime$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. ................................. 383

8.34 Contour plots of $T$ and $k$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. ................................. 384

8.35 Contour plots of $U$, $V$ and $W$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and 0.87, at $Ra = 3.1 \times 10^{13}$. ................................. 385
8.36 Contour plots of $k$, $P^r$ and $T$ within cross-sections normal to the axis of the penetration, at $Z=0.23$, 0.46, 0.69 and 0.87, at $Ra = 3.1 \times 10^{13}$. .................................................. 386

8.37 Contour plots of $U$ and $V$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$. 387

8.38 Contour plots of $W$ and $P^r$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$. 388

8.39 Contour plots of $T$ and $k$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$. 389

8.40 Contour plots of $V$, $T$ and $P^r$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and 0.5, at $Ra = 3.1 \times 10^{13}$. .................................................. 390

8.41 Contour plots of $V$ and $k$ within horizontal planes outside the penetration, at locations $Y=0.22$(inlet), $Y=-0.11$ and $Y=-0.78$(outlet), at $Ra = 3.1 \times 10^{13}$. .................................................. 391

8.42 Contour plots of $V$ at different time steps within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. .................................................. 396

8.43 Contour plots of $V$ at different time steps within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. .................................................. 397

8.44 Contour plots of $W$ at different time steps within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. .................................................. 398

8.45 Contour plots of $W$ at different time steps within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. .................................................. 399

8.46 Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$, at $Ra = 3.1 \times 10^{13}$. .................................................. 400

8.47 Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.46$, at $Ra = 3.1 \times 10^{13}$. .................................................. 401

8.48 Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$, at $Ra = 3.1 \times 10^{13}$. .................................................. 402
8.49 Contour plots of V, W and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.87$, at $Ra = 3.1 \times 10^{13}$.

8.50 Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$ and 0.46, at $Ra = 3.1 \times 10^{13}$.

8.51 Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$ and 0.87, at $Ra = 3.1 \times 10^{13}$.

8.52 Contour plots of time-averaged and steady-state V within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$.

8.53 Contour plots of time-averaged and steady-state W within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$.

8.54 Contour plots of time-averaged and steady-state V within cross-sections normal to the axis of the penetration, at $Z=0.23$, 0.46, 0.69 and 0.87, at $Ra = 3.1 \times 10^{13}$.

8.55 Contour plots of time-averaged and steady-state W within cross-sections normal to the axis of the penetration, at $Z=0.23$, 0.46, 0.69 and 0.87, at $Ra = 3.1 \times 10^{13}$.

8.56 Contour plots of steady-state, time-averaged and total $k$ within cross-sections normal to the axis of the penetration, at $Z=0.23$, 0.46, 0.69 and 0.87, at $Ra = 3.1 \times 10^{13}$.

A.1 Mean temperature, velocity and velocity fluctuations in the 60° stable cavity using $k$-$\varepsilon$-AWF with different number of grids.
Abstract

This study is focused on the efficient computation of natural convection flows. Numerical investigations have been carried out of turbulent natural convection flow in cavities, using different strategies for the modelling of the near-wall turbulence and also different levels of modelling of the turbulent stresses and the turbulent heat fluxes. More specifically, for the modelling of near-wall turbulence the low-Reynolds-number modelling approach, the standard wall function (SWF) and the more recently developed analytical wall function (AWF) strategies have been tested. For the turbulent stresses, the effective viscosity approximation (EVM), the basic Reynolds stress closure (RSM) and a more recent variant, the two-component-limit version (TCL-RSM) of the Reynolds stress closure have been tested. The turbulent heat fluxes have been modelled through the effective diffusivity approximation when the EVM was used for the Reynolds stresses, while for the RSM closures, the generalised gradient diffusion hypothesis (GGDH) and a more elaborate algebraic expression for the fluxes that involved the temperature variance $\overline{t^2}$ and its dissipation rate, $\varepsilon_t$, obtained from additional transport equations, have been tested. The studied cases are flow in high-aspect-ratio rectangular cavities inclined at angles of 60° (with stable and unstable heating configurations, both at $Ra = 0.8 \times 10^6$), 15° (with stable and unstable heating configurations, at $Ra = 1.4 \times 10^6$ and $Ra = 0.8 \times 10^6$, respectively), 5° stable ($Ra = 4.16 \times 10^8$). The cavities have the longitudinal aspect ratio of 28.7 and spanwise aspect ratio of 6.8. All the inclined cavities are computed as 2D while the 15° stable and unstable configurations are computed as 3D time-dependent as well. The final case studied was a horizontal annular penetration ($Ra = 4.5 \times 10^8$ and $Ra = 3.1 \times 10^{13}$). The horizontal annular penetration case is computed as 3D steady-state and time-dependent.
For the inclined tall cavities, in the 60° stable case, most of the models tested perform reasonably well over much of the domain. The \( k-\varepsilon \) model using LRN, AWF and SWF produce reasonable predictions of Nusselt number which were slightly lower than the experimental data. The stress transport schemes tend to over-predict the mixing near the end walls, although this can be improved by adopting a more complex model for the turbulent heat fluxes. This more elaborate thermal model has little effect on the predicted local Nusselt number, which is already in close agreement with the measurements. In the 60° unstable case all of the models tested produced reasonable predictions of the thermal field compared with the experimental data. The Nusselt numbers that were predicted using the \( k-\varepsilon \)-AWF are in close agreement with the data. In contrast to what is indicated by the data, the RSM predicts multiple circulation cells in the 60° unstable cavity. When the cavity is further inclined to an angle of 5°, with stable heating configuration, the turbulence levels are quite low and, as a result, most of the models predict fairly similar mean flow and Nu profiles, in reasonable agreement with available LES data. In the LES simulation, repeating flow conditions are imposed in the spanwise directions, which results in a time-averaged 2D flow.

For the 15° stable and unstable test cases, the measurements show that the flow within these cavities becomes three-dimensional, with large-scale structures both in the spanwise and the lengthwise planes. For the 15° unstable test case, the two-dimensional RANS computations return wall-parallel velocities an order of magnitude higher than those measured and also a temperature gradient across the cavity core. A three-dimensional time-dependent simulation with the \( k-\varepsilon \)-AWF model reproduces the four-vortex structure over the entire length of the cavity. The RSM, on the other hand, produced 3 longitudinal vortices, the sizes and strength of which were almost constant from the bottom to top of the cavity, a feature that is closer to experimental findings. The RSM’s Nu predictions, like the \( k-\varepsilon \) ones, are in close agreement with most of the available data, even at the central, Y=0.5, location where the \( k-\varepsilon \) over-predicted the Nu data. The spectral distribution of the temperature fluctuations is well resolved by the \( k-\varepsilon \)-AWF and RSM unsteady flow computations. For the 15°
stable test case, the computed three-dimensional structures are weaker. In contrast to the experimental evidence, the time-dependent three-dimensional $k$-$\varepsilon$-AWF simulations returned a mainly stable and largely two-dimensional flow field. With the introduction of the RSM model, on the other hand, the time-dependent RANS are able to reproduce the three-dimensional flow features, though not to the same degree as the experimental data. The Nu variation predicted by the RSM is in reasonable agreement with the measurements in terms of the Nu levels and variations, while the EVM returned the correct levels for the Nusselt number, but not the correct variation.

The horizontal annular penetration case involves a horizontal annular passage, closed at one end, with the inner core maintained at a lower temperature. The inner core cools the fluid around it, which makes it heavier. The heavier fluid drops out of the penetration, from the open end, and new warm fluid enters to take its place. In the lower Ra case, the time-dependent simulation showed that downward flow which separates from the cold tube was not uniform all along the tube and over most of the penetration there were strong oscillations. In the higher Ra case, the contours of the instantaneous vertical velocity, in common with those at the lower Rayleigh number, showed flow oscillations within the penetration, but with smaller amplitude.
Declaration

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

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

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

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

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

I would like to express my appreciation to my supervisors, Professor Hector Iacovides and Dr. Tim Craft who have spent considerable amount of time to carry out the study. They have substantially contributed to my professional development.

I have to say thank to my supportive family for bringing me comfort and relief especially at the difficult times during the study.
Dedicate to Maryam and Annahita...
Nomenclature

Acronyms

AWF Analytical Wall Function
CFD Computational Fluid Dynamics
CV Control Volume
DNS Direct Numerical Simulation
EVM Eddy-Viscosity Model
GGDH Generalized Gradient Diffusion Hypothesis
HRN High Reynolds Number
LES Large-Eddy Simulation
LHS Left-Hand Side
LRN Low Reynolds Number
NLEVM Non-Linear Eddy-Viscosity Model
PDE Partial Differential Equation
PLDS Power Law Differencing Scheme
QUICK Quadratic Upwind Interpolation for Convection Kinematics
RANS Reynolds Averaged Navier-Stokes
RHS Right-Hand Side
RSM  Reynolds-Stress Model
SGS  Sub-Grid-Scale
SIMPLE  Semi-Implicit Method for Pressure-Linked Equations
TCL  Two-Component-Limit
TEAM  Turbulent Elliptic Algorithm - Manchester
UMIST  University of Manchester Institute of Science and Technology

**Greek symbols**

- \( \alpha \): Prescription constant for the turbulent viscosity in AWF, \( \equiv c_\mu c_l \)
- \( \alpha \): Thermal diffusivity, \( \equiv \frac{\lambda}{\rho c_p} \)
- \( \alpha, \gamma \): Terms used in the scaling function \( F_\varepsilon \)
- \( \alpha_t \): Term used in evaluation of the AWF, \( \equiv \frac{Pr_\alpha}{Pr_t} \)
- \( \beta \): Coefficient of volumetric expansion, \( \equiv -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right) \)
- \( \chi \): Von Karman thermal constant, \( \equiv \kappa/Pr_t \)
- \( \delta \): Thickness of the hydrodynamic boundary layer
- \( \delta_{ij} \): Kronecker delta
- \( \delta_T \): Thickness of the thermal boundary layer
- \( \eta \): Kolmogorov length scale, \( \equiv (\nu^3/\varepsilon)^{1/4} \)
- \( \Gamma \): Molecular thermal diffusion, \( \equiv \frac{\lambda}{c_p} \)
- \( \Gamma_{th} \): Effective thermal diffusion, \( \equiv \left( \frac{\mu}{Pr_t} + \frac{\mu_t}{Pr_{tt}} \right) \)
- \( \kappa \): Von Karman constant, \( \equiv 0.41 \)
- \( \lambda \): Flow parameter indicating changes in the viscous sublayer thickness
- \( \lambda \): Function used in Johnson-Launder wall function
\[ \lambda \] Thermal conductivity

\[ \mu \] Molecular dynamic viscosity

\[ \mu_t \] Turbulent dynamic viscosity

\[ \nu \] Kinematic viscosity

\[ \nu_t \] Turbulent kinematic viscosity

\[ \Omega \] Vorticity invariant

\[ \omega \] Dissipation rate per unit of turbulent kinetic energy

\[ \Omega_{ij} \] Vorticity, \( \equiv \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \)

\[ \overline{\theta^2} \] Scalar invariance

\[ \overline{\theta} \] Dissipation rate of \( \overline{k} \) averaged over near-wall cell

\[ \phi \] General variable

\[ \phi_{\theta} \] Pressure-scalar gradient term of \( u_i \theta \)

\[ \phi_{ij} \] Pressure-correlation term in Reynolds-stress models

\[ \rho \] Density

\[ \sigma_k, \sigma_\varepsilon \] Empirical constants in \( k \) and \( \varepsilon \) transport equations

\[ \tau^* \] Normalised time

\[ \tau_{\eta} \] Kolmogorov time scale, \( \equiv (\nu/\varepsilon)^{1/2} \)

\[ \varepsilon \] Dissipation rate of turbulent kinetic energy \( k \)

\[ \varepsilon_{i\theta} \] Dissipation rate of scalar flux \( u_i \theta \)

\[ \varepsilon_{i\theta} \] Dissipation rate of scalar invariance \( \overline{\theta^2} \)

\[ \varepsilon_{ij} \] Dissipation rate of turbulent stress component

\[ \varepsilon_{ij}', \varepsilon_{ij}'' \] Kronecker delta

\[ t \] Fluctuating temperature component
Mean temperature
\( \theta \)
Temperature fluctuation
\( \Delta \)

**Symbols**

- \( \overline{P}_k \) Average production of turbulent kinetic energy
- \( \overline{u_i}\theta \) Turbulent scalar flux
- \( \overline{u_iu_j} \) Kinematic Reynolds stress
- \( A \) Cell-face area
- \( \hat{A} \) Flatness parameter
- \( A_1, A_2 \) Integration constants in hydrodynamic AWF
- \( A_2, A_3 \) Stress invariants
- \( a_{ij} \) Anisotropy tensor
- \( A_{th} \) Integration constant in thermal AWF
- \( b \) Coefficient in the mean buoyant term of AWF, \( \equiv -\frac{\mu^2}{\rho c k_p} \rho_{ref} g \beta \)
- \( B_2, M, N, R, R_\mu, Y, Y_T \) Terms used in evaluation of AWF
- \( b_\mu \) Parameter defining the variation of molecular viscosity across viscous sublayer
- \( B_T \) Mean buoyant term in the momentum equation
- \( C \) Wall-parallel convective mass flux
- \( C_1, C_2 \) Discretised convective and pressure-gradient terms in hydrodynamic AWF
- \( c_1, c_2, c_3, c'_1, c'_2, c_{\varepsilon_1}, c_{\varepsilon_2}, c_{\theta}, c_{1w}, c_{2w}, f_A \) Constants and functions in second-moment-closure models
- \( c_\mu \) Eddy-diffusivity coefficient
- \( c_{\varepsilon_1}, c_{\varepsilon_2} \) Constants of the modelled \( \varepsilon \) transport equation
Equilibrium length scale constant

Specific heat capacity at constant pressure

Convective terms in thermal AWF

Additional term in k equation

Part of $\varepsilon_{ij}$

Diffusive transport of scalar invariance $\overline{\theta^2}$

diffusion of the turbulent kinetic energy dissipation rate

Diffusive transport of scalar flux $u_i \overline{\theta}$

Tensor in $\phi_{ij}$ model

Diffusive transport of Reynolds stress

Indicator of length-scale gradient direction

Additional dissipation source term in $\varepsilon$ equation

Damping functions used in the low-Reynolds-number k-$\varepsilon$ model

Scaling function

Part of $\phi_{ij}$ model

Buoyant constants in the AWF

External force

Acceleration due to gravity

Generation rate of scalar invariance $\overline{\theta^2}$

Buoyant generation rate of $u_i \overline{\theta}$

Buoyant production of Reynolds stress

Buoyant production of turbulent kinetic energy

Local Grashof number, $\equiv \frac{g \Delta \Theta x^3}{\nu^2}$
\( k \) Turbulent kinetic energy

\( k_{i	ext{r}} \) Turbulent kinetic energy at the edge of the viscous sublayer

\( k_P \) Turbulent kinetic energy at the near wall node

\( l \) Turbulent length scale, \( \equiv \frac{k^{3/2}}{\varepsilon} \)

\( l_m \) Characteristic length scale

\( N_i \) Length scale gradient vector

\( P \) Jayatilleke pee-function in thermal log-law

\( P'' \) Deviation from the hydrostatic pressure

\( P_{i\theta} \) Generation rate of \( \overline{u_i\theta} \) by mean gradient of \( U_i \) and \( \Theta \)

\( P_{ij} \) Mean-strain production of Reynolds stress

\( P_k \) Production of turbulent kinetic energy

\( Pr_t \) Turbulent Prandtl number

\( S_{ij} \) Strain-rate tensor, \( \equiv \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \)

\( S_I \) Dimensionless third invariant of the stress tensor

\( S_i \) Heat source

\( S_U, S_P \) Source terms in discretised transport equations

\( U, V \) Mean velocities

\( U_\tau \) Friction velocity

\( u_i \) Fluctuating velocity component

\( x, y, z \) Coordinate directions

\( y^* \) Dimensionless distance to the wall, \( \equiv \frac{y \sqrt{k}}{\nu} \)

\( y^+ \) Dimensionless distance to the wall, \( \equiv \frac{y U_\tau}{\nu} \)

\( y_w \) Viscous sublayer thickness
$y_d$ Thickness of the dissipative layer

$y_n$ Thickness of the near-wall control volume

$y_{Pr}$ Distance from the near-wall node to the wall

$y_t$ Thermal viscous sublayer thickness

Gr Grashof number, $\equiv \frac{g\beta \Delta \theta L^3}{\nu^2}$

$n$ Unit vector in the direction normal to the wall

Nu Nusselt number

$P$ Mean pressure

Pr Molecular Prandtl number, $\frac{\mu c_p}{\lambda}$

Re Reynolds number, $\equiv \frac{UL}{\nu}$

$S$ Strain rate invariant

$T$ Normalised mean temperature

$t$ time

**Subscripts**

$v$ Value at the edge of the viscous sublayer

$0$ Initial value

$1$ Region 1: the viscous sublayer in derivation of AWF

$2$ Region 2: the fully turbulent region in derivation of AWF

$an$ Analytical value

$E, W, N, S, P, EE, WW, NN, SS, e, w, n, s$ Node and face values of variables

$eff$ Effective value

$n$ Value at the outer edge of the near-wall cell

$ref$ Reference value

37
rms  Root mean square value

wall  Wall values

**Superscripts**

ω  Wall-reflection terms

′  Correction values in SIMPLE algorithm

*  Dimensionless value scaled by $k$

*  Guessed values in SIMPLE algorithm

**  Dimensionless value for natural convection flows

+  Dimensionless value scaled by $U_\tau$
Chapter 1

Introduction

In this research, various RANS turbulence modelling strategies are employed to predict buoyant flows in enclosed cavities with different angles of inclination and an annular horizontal penetration with an open-end. Some of the specific aspects of turbulence modelling examined are appropriate levels of modelling to employ for the turbulent stresses and turbulent heat fluxes in the fully turbulent region, and the modelling of near-wall turbulence across the viscosity-affected layer.

1.1 Buoyant flows

Most flows that take place in our surroundings are induced, or affected, by buoyancy. Buoyant flows can be seen as air circulation around our bodies, in the atmosphere, oceanic circulation, etc. Buoyant flows originate from density differences, often resulting from temperature gradients. The buoyant flows can be subdivided into two major broad categories. One is that of buoyant flows induced by temperature differences or heat transfer between a fluid and a surface immersed in it, which are called external buoyancy induced flows. The other category is that of buoyant flows that occur inside enclosures bounded by surfaces which are at different temperatures, referred to as internal buoyancy-driven flows.

In the past years, there has been intensive research on buoyant flows. These
research studies have been carried out because of a growing demand for detailed quantitative knowledge regarding buoyancy induced flows in a wide range of applications, including the atmosphere, bodies of water, enclosures, electronic devices, process equipment. One area of relevance to the present study is the cooling of nuclear power plants. The numerical prediction of such flows can be very challenging, due to the onset of turbulence and complex flow structures at high enough temperature differences.

In this research, the main focus is to determine reliable turbulence modelling strategies for the computation of complex buoyant flows. Several RANS turbulence models have been tested in the simulation of buoyancy-driven flows with 2D/3D and steady-state/time-dependent calculations and many comparisons have been made in order to identify the turbulence model which produces the best prediction of buoyant flows.

In turbulent buoyancy-driven flows, even at relatively high Rayleigh numbers it is possible to encounter regions with stagnant fluid, laminar circulation, the transitional regime, and fully turbulent regions, all in one flow. This emphasizes the role of molecular effects both close to a solid boundary and away from walls (at the edge of turbulent flow regions). On the other hand, strong variation of all flow properties usually found in very thin boundary layers along the walls, where the buoyancy exhibits the strongest effects on turbulence. This requires a fine numerical resolution of the near-wall region and adequate modelling of both the viscous and conduction effects.

One the important targets of this study is to find approaches to the reliable, cost-effective numerical simulation of complicated buoyancy-driven flows. As mentioned earlier, capturing the complex near-wall region plays an important role in the simulation result for the entire computational domain. Two approaches have been employed in this study to account for the near-wall flow. One is to adopt a fine numerical resolution near the walls, with appropriate viscous influences in the turbulence models. This is a relatively expensive method. The other approach is to employ simplified functions to compute the near-wall variables. This method is called a Wall Function (WF) and does not
need fine numerical resolution near the walls. There are several WF strategies. In this study, two of them have been investigated. One is a widely used conventional method which is called the Standard Wall Function (SWF) and the other one is a more recently developed method which is called the Analytical Wall Function (AWF).

Several test cases have been investigated to test how different turbulence models and near wall strategies perform in the simulation of a range of flows driven by buoyancy forces. The test cases have been selected in order to cover a wide range of physical phenomena.

The first family of test cases are tall cavities having different angles of inclination. The angles of inclination range from a vertical and moderate angle of $90^\circ, 60^\circ$, respectively, to highly inclined ($15^\circ$ and $5^\circ$). For all cases apart from the $90^\circ$ and $5^\circ$, two configurations for the thermal boundary conditions are examined. One configuration is where the hot wall is located above the cold wall. In this configuration, due to the existence of stable thermal stratification, it is expected to see only weak turbulence mixing and heat transfer. In the other configuration, with the hot wall located below the cold wall, due to existence of unstable thermal stratification, it is expected to see stronger turbulence mixing heat transfer.

The final test case concerns buoyancy-driven flow in an annular horizontal penetration. One end of the annulus is closed and the other side is open to a large volume of hot fluid. The surface of the inner annulus cylinder is set to a cold temperature, and the aim of simulation is to explore how hot fluid penetrates inside the horizontal annulus from its open side. This test case provides further challenges compared to the closed cavities test cases, due to the fact that no inlet flow rate is imposed to the computational domain.
1.2 Turbulent flow

1.2.1 Characteristics

Most flows found in industrial applications are turbulent flows. Among these industrial applications, we can identify boundary layers growing on aircraft wings, combustion processes, natural gas and oil flow in pipelines and the wakes of ships, cars, submarines and aircraft.

It is difficult to state a definition for turbulent flow. The best way is to explain different characteristics of the turbulence. The first feature of turbulent flow is irregularity. That is why analytical methods are not suitable for turbulent flow and statistical and numerical methods are employed to resolve the turbulent flow. Another characteristic of turbulence is its diffusivity. The turbulence diffusivity is used to denote the mixing of momentum, mass and thermal energy by the turbulent eddies. The diffusivity of turbulence is the most important effect of turbulence in industrial applications. For example it will delay separation in flows over airfoils at high angles of attack, or will increase heat transfer rates etc.

Two major dimensionless numbers in forced and mixed convection are the Rayleigh and Reynolds numbers. They are defined as:

\[
Ra = \frac{\beta g \Delta \Theta L^3}{\nu \alpha}
\]

\[
Re = \frac{UL}{\nu}
\]

where the symbols are defined as: \(\beta\): volumetric expansion coefficient, \(g\): gravity acceleration, \(\Delta \Theta\): temperature difference, \(L\): length scale, \(\nu\): kinetic viscosity, \(\alpha\): thermal diffusivity and \(U\): velocity scale.

Turbulent flows occur at high Reynolds numbers in the case of forced convection and at high Rayleigh numbers in the case of natural convection. Because of the inherent apparent randomness and the nonlinearity of turbulent flows, the phenomenon is almost intractable. Both mathematical models and
nonlinear discretization methods have to be developed further than current methods in order to obtain accurate answers to many turbulent flow problems.

One of the important characteristics of turbulence is its dissipation. The viscous shear stresses, through the small-scale turbulent fluctuations, dissipate the kinetic energy of the turbulent fluctuations into thermal energy. That is why turbulent flow needs a continuous energy input to compensate for the viscous losses, otherwise the turbulence vanishes very quickly[9].

1.2.2 Turbulence length scale

In the dynamics of turbulent flows, a wide variety of length scales exist. These length scales range from the physical dimension of flow field to the dissipative action of turbulent motion. The larger flow length scales are representative of the size of the larger turbulent eddies. These contain most of the energy and also interact most strongly with the mean motion, removing energy from the mean motion and transferring it into turbulent eddies.

It is important to find the smallest length scale in turbulent flows in order to analyze the dissipation of turbulence through the smallest scale eddies. Viscosity causes dissipation of turbulence in the very small eddies. The small scale fluctuations originate from nonlinear terms in the momentum equations. Viscosity decays the very small eddies and dissipates the small scale energy into heat. It is often assumed that the rate of energy supplied is to be equal to the rate of dissipation. In the present study, it is important to obtain an accurate smallest length scale especially in the viscous affected near-wall region in order to calculate an accurate amount of heat transferred across the walls. Kolmogorov’s universal equilibrium theory has been based upon the above assumption.

Parameters characterising the small-scale motion can be obtained, based on the dissipation rate per unit mass $\varepsilon (\frac{m^2}{s \cdot kg})$ and the kinematic viscosity $\nu (\frac{m^2}{s})$. Using these variables, length, time and velocity scales can be obtained as follows:
These scales are called the Kolmogorov microscales of length, time and velocity\[9\].

1.2.3 Turbulent buoyancy-driven flow

Turbulent flows can be greatly influenced by body forces. One of the most common examples is the effect of gravity on flows with density fluctuations. This can occur in flows in which there are variations of mean density. One example of such a case is flow over heated or cooled horizontal plates. Another case is a flow which is driven by mean density difference, such as buoyant cavity flows, or buoyant plumes in still air. In a situation that an unstable thermal stratification, which means the hot surface is located below the cold surface, is imposed to a volume of fluid, mean density fluctuates due to mean temperature fluctuations. In turn, the mean density fluctuation gives rise to buoyancy force and velocity fluctuations which increases the amount of turbulence in the flow.

In situations where density increases in the vertical direction, the flow is unstable and consequently the potential energy is converted into turbulent kinetic energy. In this case, the buoyancy force leads to an increase in the level of turbulence. On the other hand, when the density decreases vertically in a way that its gradient is steeper than would be the case for hydrostatic equilibrium, the turbulent kinetic energy is converted into potential energy\[10\]. In this case, the buoyancy force leads to a decrease in the level of turbulence in the flow.

1.3 Turbulence modelling

The main priority in many studies of turbulence is to contribute to the development of mathematical models that can reliably and efficiently compute relevant quantities in practical applications. Past decades of research on turbulent flows have proved that it is impossible to obtain a simple analytical solution\[9\]. Instead the application of computers is increasingly growing as
the only way to achieve computation of turbulent flows.

Most mathematical models for simulation of turbulent flows lead to a set of partial differential equations. Then in a specific flow, this set of equations with appropriate initial and boundary conditions can be solved numerically. The most common mathematical modelling approaches for computing engineering turbulent flows are Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds Average Navier-Stokes (RANS) methods.

In DNS, the filtered Navier-Stokes equations are directly solved to obtain the full velocity and pressure fields $U(x_i, t)$ and $P(x_i, t)$. In this method all the time and length scales should be resolved. This makes the method computationally very expensive. The computational cost is proportional to $Re^3$ [9], limiting the application of this method to low and moderate Reynolds number flows.

In LES, the Navier-Stokes equations are solved to obtain a velocity field $\overline{U}(x_i, t)$. This filtered velocity field resolves only the larger scale turbulent motions. The effect of smaller scale motions are included in the set of equations by using sub-grid scale models.

In RANS, the set of Reynolds averaged equations are solved to obtain the mean velocity field. In these equations, there are terms called the Reynolds stresses. The Reynolds stresses form a tensor that represents the effect of the turbulent fluctuations on the mean hydraulic field. The RANS equations only solve the quantities related to the mean flow. Therefore for the Reynolds stresses, which represent correlations of the smaller scale turbulence need to be modelled. Two broad ways are employed in this study to model the Reynolds stresses. The first method employs a turbulent viscosity to model the Reynolds stresses via algebraic equations. The second way (Reynolds Stress Model) solves partial differential equation for the stresses and thus avoids using the turbulent viscosity. Different methods are also used for calculation of the turbulent heat fluxes. One is eddy-diffusivity model which simply approximate the turbulent heat fluxes by mean temperature gradient along the direction of
the component of the turbulent heat flux. More elaborate algebraic expressions are also implemented to take into account effects of mean velocity strains and gravity on the turbulent heat fluxes.

1.4 Objectives

It is generally recognized that the two-equation $k-\varepsilon$ model (and, in fact, most eddy-viscosity models with a linear stress-strain relation and eddy-diffusivity model for the turbulent heat fluxes) cannot reproduce flows with significant non-equilibrium effects due to buoyancy. The first obvious deficiency of eddy-viscosity and eddy-diffusivity models (for calculation of Reynolds stresses and turbulent heat fluxes) in flows driven by thermal buoyancy is the absence of buoyancy related production terms, which appear in the full transport equations of the Reynolds stresses and turbulent heat fluxes. In the eddy-diffusivity model, components of the turbulent heat flux vector are aligned with the corresponding components of the mean temperature-gradient. The alignment of the turbulent heat flux with the mean temperature-gradient vector leads to inaccurate modelling in some cases. For instance, the natural convection flow moves along a heated vertical wall. The vertical component of turbulent heat flux is almost zero due to its relation with the almost zero temperature gradient in the vertical direction, as implied by the isotropic eddy-diffusivity model.

In a convective boundary layer along a heated vertical wall, the major buoyancy source of turbulence kinetic energy is associated with the vertical heat flux. With the eddy-diffusivity model, the magnitude of the wall-parallel heat flux is significantly under-predicted, since the scheme is tuned for calculation of the wall-normal heat flux in forced convection flows. Associating the wall-parallel turbulent heat flux with the usually negligible (zero for an infinite plate) mean temperature gradient in the vertical direction will eliminate the buoyancy effect on turbulence, contrary to the basic physics of the buoyancy-driven turbulence. Since, in this study a number of flows are considered where there are heated or cooled vertical, or near-vertical walls, more
elaborate expressions for modelling of the turbulent heat flux have been examined to establish some of these strengths and weaknesses.

As mentioned earlier, the eddy-viscosity model is not accurate for calculation of turbulent stresses in buoyancy-driven flows due to lack of buoyancy-force sources in its expression for calculation of Reynolds stresses. Therefore, in this study, Reynolds Stress Transport Models (RSM) have also been tested, since these directly account for the effect of buoyancy of the turbulent stresses and thus offer the potential for more accurate prediction of the turbulent stresses.

Several LRN near wall treatments are already applied for buoyancy-driven flows with some success but their deficiency is that they need fine near-wall mesh which is computationally not economic. To account for the near-wall modelling in this study a low-Re model, and two different wall-function treatments were examined. The standard wall function is based on the assumption of near-wall equilibrium which leads to logarithmic velocity and temperature distributions. However, in a buoyancy-driven flow, there is not local equilibrium in the near-wall region. Even if the logarithmic distribution is correct for a particular flow, it is only valid inside the fully turbulent region. Therefore, if the near-wall node is located inside the viscous affected layer, the SWF produces inaccurate predictions. It is common in complex turbulent buoyant flows that laminar, transition and fully turbulent regimes may exist in a single case. Therefore, it is necessary to employ a near wall treatment that can account for the effects of buoyancy forces (which cause deviation from near-wall equilibrium and log law distribution) and which is also valid in both laminar (or viscous-affected) and fully turbulent regimes. The AWF was derived to achieve at least some of these targets. In this study, the performance of AWF was therefore examined in the simulation of buoyancy-driven flows.
1.5 Outline of thesis

In this thesis, after the introduction Chapter, in Chapter 2 major experimental and numerical studies which have been carried out before are discussed to gain more understanding of the physics of buoyant flows and also of the advantages and disadvantages of numerical models available. Chapter 3 discusses the mathematical models employed to represent turbulence and presents all the turbulence models employed in this research. In Chapter 4, different wall functions are discussed in detail. The analytical wall function is explained more extensively and some proposed amendments to the original analytical wall function are presented. In Chapter 5, the numerical solution methods which are employed in this study are explained. In Chapter 6, the results of 2D numerical simulations of the flow in tall cavities at different angles of inclination are compared with experimental data. It was concluded that the highly inclined cavities (15° stable and unstable) need to be simulated by 3D, time-dependent computations. In Chapter 7, the three dimensional time dependent numerical simulations of flow in a tall cavity are discussed in detail. In chapter 8, the numerical simulation of buoyancy-driven flow inside a horizontal annulus with one open-end is presented. Chapter 9 is allocated to conclusions and suggestions for further study.
Chapter 2

Literature Review

In this chapter major research efforts that have been carried out so far in cases of natural or mixed convection will be reviewed. In Section 2.1, experimental researches and their findings regarding the physics of natural or mixed convection are discussed. In Section 2.2, the major studies in the numerical modelling of buoyancy-induced flows are reviewed and the intended contribution of this work to the progress of numerical simulation of natural convection is explained.

2.1 Experimental Studies

Flat Plates

Warner and Arpaci[2] carried out an experimental study of turbulent natural convection in air over a vertical heated flat plate. Their results showed good agreement with the analytical correlation of Bayley[11] \( (Nu = 0.1Ra^{1/3}) \) for Rayleigh numbers up to \( 10^{12} \). Those measurements and correlations are compared in the Figure 2.1. The temperature measurements showed that the thermal boundary layer gets thicker as \( x \) increases (\( x \) is the distance from bottom of the vertical heated wall). The results also showed that temperature profiles retain the same basic shape over the length of the vertical plate.
Kutateladze et al.[12] carried out an experimental investigation on the hydrodynamics of a turbulent free convection boundary layer on a vertical plate while a film of ethyl alcohol covered the hot plate with constant temperature. Electronic stroboscopic flow visualization was employed in this experiment. Based on their measurements, they found that along the middle of the vertical cross section of the turbulent boundary layer, the temperature gradient along the vertical plate was zero \( (d\Theta/dx = 0, \ x \text{ is in the vertical direction}) \). Therefore they reported the existence of a quasi-stationary (weak flow) wall layer. In the region of the quasi-stationary wall layer, the maximum velocity and wall distance at which maximum velocity occurred \( (\delta) \), satisfied the condition:

\[
\text{RePr}^{1/2} = \frac{U_{\text{max}}\delta}{(\nu\alpha)^{1/2}} = \text{const.}
\]

Rough Surfaces

Abu-Mulaweh[13] carried out an experimental investigation of the effect of backward and forward-facing steps on turbulent natural convection flow along a vertical flat plate (Figure 2.2). Laser-Doppler velocimetry and cold wire anemometry were employed to simultaneously measure mean velocity and temperature distributions and their turbulent fluctuation intensities. The heights of the backward and forward steps were 22mm. The temperature difference between the hot wall and ambient air was \( \Theta = 30^\circ C \) and the Grashof number was \( Gr = 6.45 \times 10^{10} \). Based on the experimental data, the maximum

Figure 2.1 – Comparison of experimental measurements and correlations[2]
local Nusselt number occurs in the reattachment region. For the backward-facing step, the maximum local Nusselt number was 2 times higher than the maximum Nusselt number over a flat plate. For the forward-facing step, the maximum local Nusselt number was 2.5 times more than for a flat plate.

![Figure 2.2 – Schematics of the flow geometry, (a) flat plate, (b) backward-facing step, (c) forward-facing step.]

**Cylindrical and rectangular passages**

Aicher and Martin[3] experimentally studied mixed turbulent natural convection and forced convection heat transfer in vertical tubes. They summarized the experimental data available in the literature for both aiding and opposing flow conditions. The aiding buoyant flow means that the buoyancy force is in the direction of the forced convection flow and the opposing buoyant flow means the buoyancy force is in the opposite direction to the forced convection flow. Figures 2.3 and 2.4 show the effect the buoyancy force has on velocity profiles in mixed convection flows. The turbulent heat transfer is dependent on two mechanisms. One is heat conduction across the viscous sublayer and the other is diffusion of energy from the edge of viscous sublayer to the core of turbulent flow. The rate of turbulent diffusion of thermal energy is determined by the production of turbulent energy near the viscous sublayer. The production of turbulence energy is proportional to the mean velocity gradients. In Figure 2.4 the velocity profile for the aiding situation is depicted.
Based on the above discussion, heat transfer is reduced. In the buoyancy opposed situation, the effect of the buoyant force is to increase heat transfer.

**Figure 2.3** – Velocity profiles under aiding and opposing buoyancy force[3].

They (Aicher and Martin) then presented their own set of experimental data which focused on the influence of length-to-diameter ratio and of heat and mass flux directions on heat transfer in vertical tubes. Based on their measurements, they showed that in the case of aiding flow, heat transfer would be reduced with increasing length-to-diameter ratio. For opposing mixed convection, a change of length-to-diameter ratio had almost negligible effect on heat transfer.

**Figure 2.4** – Effect of aiding natural convection on a turbulent flow in a vertical tube. $u_e$ is the velocity at the border of the viscous layer, $u_m$ is the mean velocity in the core of the flow.[3]
Cavities and Enclosures

Kirkpatrick and Bohn[4] experimentally studied natural convection at a high Rayleigh number in a cubical enclosure with various thermal boundary conditions. They ran experiments using four different configurations of differentially heated and cooled vertical and horizontal walls. They made measurements of mean and fluctuating values of temperature and observed fluid flow patterns in the enclosure. It was concluded that heating from below would cause more mixing in the enclosure and consequently cause a reduction of temperature stratification. As the Rayleigh number was increased, the thermal stratification at the core of the cavity decreased. They observed that the thermal stratification changed suddenly at $Ra \approx 0.65 \times 10^{10}$. The thermal stratification was also not symmetric around the mid-plane. Increasing the bottom wall temperature, which increases the Ra number, lead to an increase of the temperature difference between the bottom wall and the core and produces stronger thermal plumes which in turn caused greater mixing of the core. The range of Rayleigh numbers covered in the experiment was from $0.1 \times 10^{10}$ to $3 \times 10^{10}$. The working fluid was water. Comparisons were made between average Nusselt numbers over the bottom wall of the cavity at various thermal boundary conditions from the experiment and some available correlations of Nusselt and Rayleigh number (such as Figure 2.5). This Figure corresponds to the thermal boundary condition in which the bottom wall is hot and the top wall is cold and the side walls are adiabatic. The experimental measurements show that the averaged Nu number has a $1/3$ power dependence on the Ra number.

![Figure 2.5](image)

**Figure 2.5** – Heat transfer data and correlations for the Rayleigh-Bénard configuration[4].

Cheesewright et al.[14], Cheesewright and Ziai[15], Bowles and Cheesewright[16]
and Cheesewright and King[17] carried out experimental investigations on two-dimensional buoyant cavity flows. They studied rectangular cavities to produce experimental data for validation of two-dimensional computer codes. The working fluid was air and the Rayleigh number developed based on cavity height was \(10^{10}\). They produced a large amount of measurements of mean and fluctuation values of velocity and temperature in the cavity. LDA and thermocouples were employed to carry out the measurements of velocity and temperature respectively. They observed that the flow on the hot and cold walls were not symmetric to each other. They also reported re-laminaris ation on the floor wall and then transition to turbulence at 20% of the way up the hot wall. The measurements of velocity fluctuations showed a reduction at the bottom of the cavity, which reconfirmed the re-laminarisation phenomena they reported here.

DafaAlla and Betts[18] experimentally studied turbulent natural convection in a tall air cavity. They analyzed turbulent natural convection in an air cavity with an aspect ratio of 28.6. The experiment resulted in data for velocities and temperatures at a Rayleigh number based on cavity width of \(0.83 \times 10^6\). The experimental data is useful for validating computational and theoretical studies. Velocity measurements were carried out by a Laser Doppler Anemometer. Fine thermocouples were employed for temperature measurements.

Betts and Bokhari[19] experimentally studied turbulent natural convection in an enclosed tall cavity. The turbulent natural convection flow developed in a tall differentially heated rectangular cavity (2.18 m high by 0.076 m wide by 0.52 m in depth). The temperature difference was imposed between the vertical walls which were maintained at isothermal temperatures of \(19.6^\circ C\) and \(39.9^\circ C\). This temperature difference resulted in a Rayleigh number of \(1.43 \times 10^6\) based on the width of the cavity as length scale. In the above mentioned situation, the flow in the core of the cavity was fully turbulent and variation of fluid properties with temperature was negligible. They modified the experimental rig previously used by Dafa’Alla and Betts[18] by fitting partially conducting top and bottom walls and outer guard channels, to provide better adiabatic
boundary conditions which avoid the inadequately defined sharp changes in
temperature gradient and other problems associated with insufficient insula-
tion on nominally adiabatic walls. Mean and turbulent temperature and ve-
locity variations inside the cavity were measured. Heat fluxes and turbulent
shear stresses were also measured at various locations. The temperature and
velocity distributions were shown to be nearly two-dimensional, except close
to the front and back walls, and anti-symmetric across the diagonal of the cav-
ity. The partially conducting top and bottom walls created locally unstable
thermal stratification in the wall impinging flows there, which increased the
turbulence as the flow moved toward the isothermal vertical plates. The im-
pinging jets are located at the top of the cold wall and the bottom of the hot
wall. The impinging jets are created due to acceleration of buoyant flow to-
wards the top of the hot wall and change of direction of the accelerated flow
at the top of the hot wall towards the cold wall. The same phenomena takes
place at the bottom of the cavity as well.

Tian and Karayiannis[5] carried out an experimental investigation on low
turbulence natural convection in an air filled square cavity (Figure 2.6). Ther-
mocouples and 2-D LDA were employed to accomplish the measurements.
The experimental data showed that fluid flow inside the cavity was two di-
mensional. The low level turbulent natural convection flow developed a Rayleigh
number of $1.58 \times 10^9$. The main focus in their work was analyzing turbulence
quantities. The buoyancy-driven flow was created from temperature differ-
ence between vertical walls of the cavity where one side was hot and the other
side was cold. In this case, the turbulence regions are located near the hot
and the cold walls while the core region remains relatively laminar. They
also reported the flow was anisotropic in the turbulence regions. Turbulent
frequencies of 0.1-0.2Hz were identified in the investigation. Turbulent inten-
sity increased downstream of flow along the hot and cold walls where the
downstream for the hot wall is toward the top of the hot wall and for the cold
wall is toward the bottom of the cold wall. Therefore the peak values of the
power spectral densities move to higher frequencies downstream of the flow.
The experimental results revealed that the temperature and velocity fluctua-
tions were confined to the boundary layers along the solid walls and were not
in Gaussian distribution. It was concluded that the temperature and velocity components were fluctuating independently of each other. The turbulence quantities were calculated based on the time-average of measurements within a specific period of time.

Ampofo and Karayiannis[6] experimentally investigated turbulent natural convection in an air filled square cavity. In the experiment the temperature difference caused low-level turbulent natural convection in the air filled vertical square cavity. It produced a two dimensional flow inside the cavity. The cavity dimensions were 0.75 m high, 0.75 m wide, 1.5 m deep. The hot and cold vertical walls of the cavity were isothermal at $50^\circ C$ and $10^\circ C$ respectively giving a Rayleigh number of $1.58 \times 10^9$. It was shown that the temperature distribution inside the near-wall viscous sublayer was linear. They observed that the thickness of the inner boundary layers along the hot and cold walls were 7% of the thickness of the outer boundary layer where the inner boundary layer is the distance from the wall to the position of maximum velocity and the outer boundary layer is the distance from the position of maximum velocity to the position that velocity becomes zero (Figure 2.7). It was reported that the thickness of the viscous sublayer was 3mm and the thickness of the conduction region was 2mm. In this case the maximum Nusselt number occurred at the bottom of the hot wall and top of the cold wall. This is because of the thinner boundary layer at these locations. Based on the measurements, the turbulent heat flux was nearly zero at the bottom of hot wall and the top

![Figure 2.6 – Schematic view of the air filled cavity[5].](image)
of cold wall. They showed that the flow in those regions was almost laminar.

Ampofo[7] carried out an experimental investigation of turbulent natural convection of air in non-partitioned and partitioned cavities with differentially heated vertical and conducting horizontal walls. The experiments resulted in two-dimensional flow in the mid-plane of the cavity with dimensions of 0.75 m high, 0.75 m wide, 1.5 m deep. The thermal boundary conditions of the experiment were isothermal vertical walls at 50°C and 10°C. This temperature difference created a Rayleigh number of $1.58 \times 10^9$. He installed five partitions made of higher thermal conductivity material than that of the cavity hot wall. The partitions' dimensions were 150 mm long, 3 mm thick and the same depth as the depth of the cavity. The local velocity and temperature were measured simultaneously at various locations in the cavities. A laser Doppler anemometer and a microdiameter thermocouple were employed to carry out the measurements. Both mean values and fluctuating values were measured and presented in the experimental results. The local and average Nusselt numbers, the wall shear stress and the turbulent kinetic energy were also recorded in this experiment. The experiments were accomplished with very high accuracy devices and the results can be used as experimental benchmark data and are useful for validation of computational fluid dynamics codes. Based on the experimental measurements, it was concluded that the average and local Nusselt numbers decreased when the partitions were placed over the hot wall. The

Figure 2.7 – Boundary layer structure at $Y=0.5$ (vertical velocity profile near the hot wall)[6].
existence of partitions substantially modified the dynamic and thermal fields compared to those in the empty cavity with the same dimensions and boundary conditions. The partitions on the hot wall also affected the flow near the cold wall and made the boundary layer thicker and gave higher peak velocity values (Figure 2.8).

![Figure 2.8](image)

**Figure 2.8** – Vector plot of mean velocity profiles (a) Non-partitioned cavity (b) Partitioned cavity[7].

Valencia et. al.[20] experimentally and numerically studied turbulent natural convection in a cubical cavity heated from below and cooled from above. In their research, water was the working fluid (Pr = 6.0). They measured the velocity field by the particle image velocimetry technique for Ra numbers $Ra = 10^7$, $Ra = 7 \times 10^7$ and $Ra = 10^8$. They also carried out DNS analysis of the test case for the range of Ra numbers $10^7$ to $10^8$. In the numerical calculations, they tested three types of thermal boundary conditions for the vertical walls of the cavity. They imposed perfectly adiabatic, partially and perfectly conductive conditions for those walls. The perfectly conductive wall was imposed by a setting linear temperature variation from the bottom hot wall to the top cold wall while the partially conductive condition was identified in the experimental data which means there was non-linear temperature distribution along the side walls. They concluded that the averaged heat transfer rate and overall flow structure stayed almost the same between partially conducting walls and perfectly conducting lateral walls, but they found that the difference between velocities reached up to 110 percent. They achieved better agreement between
their predictions and their experiments when they took into account the walls’ thermal conductivity by imposing the non-linear temperature distribution resulting from the experimental measurements along the side walls.

Baïri et. al.[1] investigated steady-state natural convection within rectangular cavities. The working fluid in their study was air. They considered different angles of inclination from 0° to 360° for cavities with aspect ratios $L/H = 0.75$ and 1.5, where $L$ is the distance between the active walls. In this investigation, the active walls are set to isothermal hot and cold thermal boundary conditions and the other walls are adiabatic. They included the numerical simulation of cavities with configurations corresponding to inclinations of 0° (vertical active walls), 90° (hot wall down, Rayleigh-Bénard convection) and 270° (hot wall up, pure conductive mode). The $Ra$ number ranged from $10^1$ to $10^8$. The temperature measurements carried out for heat transfer analysis using thermocouples. They developed empirical relations between $Nu$ and $Ra$ for cavities with the above mentioned configurations (Table 2.1). These correlations are useful for design of thermal structures similar to these cavities.

<table>
<thead>
<tr>
<th>$\alpha$ (Degree)</th>
<th>Correlation</th>
</tr>
</thead>
<tbody>
<tr>
<td>0, 30, 360</td>
<td>$Nu = 0.147Ra^{0.287}$</td>
</tr>
<tr>
<td>45, 135, 315</td>
<td>$Nu = 0.130Ra^{0.305}$</td>
</tr>
<tr>
<td>60, 90</td>
<td>$Nu = 0.133Ra^{0.304}$</td>
</tr>
<tr>
<td>270</td>
<td>$Nu = 0.058Ra^{0.058}$</td>
</tr>
</tbody>
</table>

Table 2.1 – Correlations $Nu$-$Ra$ for Aspect ratio = 0.75 and 1.5 at different angles of inclination and $10^3 < Ra < 10^8$ [1].

Cooper et al. [8] experimentally studied natural convection flows inside inclined tall cavities. He provided reliable temperature and velocity measurements in a tall rectangular cavity with different inclination angles. A tall rectangular differentially heated air cavity with aspect ratio of 28.6 was employed. The tall walls were kept at constant temperature while there was a specified temperature difference between two walls. The rest of the surfaces
were insulated from the surroundings. The experimental rig was set up so that the angle between the tall walls and horizontal line could be varied from 0 to 180 degrees. Both temperature and velocity measurements were obtained at different heights and different angles of inclination at a Rayleigh number $Ra = 0.86 \times 10^6$. Laser Doppler Anemometry (LDA) and a Chromel-Alumel thermocouple were employed for velocity and temperature measurements respectively. For the 90 degree (vertical) and 60 degree (hot wall at the top) configurations, he concluded that the flow was fully turbulent and the temperature field was almost 2 dimensional. He also found that the temperature profiles in the vertical and 60 degree inclined tall cavity were similar. He also investigated highly inclined cavities in thermally stable and unstable configurations. Thermally stable configuration refers to the arrangement where the hot wall is located above the cold wall and the thermally unstable configuration is the other way round. In these cases, the tall walls of the cavity made an angle of 15 degrees with the horizontal axis. Based on the velocity measurements, he reported complicated 3D hydraulic fields consisting of multiple circulation cells for both stable and unstable cases. However, the thermal field stayed almost 2D.

Some of the important experimental studies on natural convection which have been carried out in the past have been discussed in this section. It has been shown that numerous experimental studies have been carried out to discover the characteristics of buoyancy induced flows. Natural convection flows typically originate from a thermal instability. The warmer and lighter fluid moves upward relative to cooler and heavier fluid which moves downward. As in forced convection, disturbances inside the natural convection flows might increase and transition from laminar to turbulent flow occur. The transition to turbulence for natural convection flow depends on the Rayleigh number which is the ratio of buoyancy and viscous forces. For natural convection over a vertical plate, transition to turbulent flow occurs at around $Ra = 10^9$.

In a vertical tall cavity with heated side walls, the fluid near the vertical hot wall moves up due to the increase of temperature and reduction of density
and the fluid near the vertical cold wall moves down due to the temperature reduction and increase of density. The greater the temperature difference between the hot and the cold wall is, the higher the velocity and velocity gradient will be near the hot and the cold walls. The turbulence is generated by means of mean velocity gradients and buoyancy forces. Therefore as the temperature difference increases, the turbulence generation increases until the flow starts to undergo transition to turbulence. In the tall cavity, due to the short distance between the active walls, there is no zero velocity region in the core of the cavity. The velocity simply changes sign across the center line. There are relatively high gradients of velocity parallel to the hot and cold walls at the core of the cavity and in turn the maximum turbulence levels occur in the core as well.

As the tall cavity is tilted, the flow pattern changes, depending on the relative positions of the hot and the cold walls. In the case that the tall cavity is tilted so that the hot wall is positioned above the cold wall, the buoyancy force acting on the fluid near the hot and cold walls due to the temperature difference decreases compared with buoyancy force within a vertical cavity. Therefore the velocity and velocity gradients decrease and in turn turbulence levels decrease as the cavity is tilted more. The fluid will be fully stagnant in the case that the hot and cold walls are horizontal and the hot wall is located at the top. In the case that the tall cavity is tilted so that the cold wall is positioned above the hot wall, the flow will be unstably stratified. The fluid near the bottom hot wall always tends to move up, due to the buoyancy force, and the fluid near the top wall tends to move down. As the the cavity is tilted more, the buoyancy force increases. At a certain angle of inclination this will give rise to the formation of multiple circulation cells within the tall cavity. That is why the flow is expected to have higher turbulence levels in this case. In summary, as the tall cavity is tilted toward the stable flow configuration (the hot wall at the top), the Nusselt number reduces because convective flow is less strong and turbulence levels is lower. In the case that the tall cavity is tilted so that the hot wall is at the bottom, the Nusselt numbers increase because there is stronger convective flow and higher turbulence levels.

In square cavities, the flow pattern is different to that described above. Due
to the larger distance between the active walls of the square cavity, there is an almost zero velocity region at the core of the square cavity. In this case, the velocity near the hot and cold walls reaches a maximum and then reduces to almost zero in the core region. Therefore, due to lack of velocity gradients in the core, there is almost laminar flow in that region and turbulent flow is located only near the hot and cold walls. That is why a clear temperature stratification is usually observed at the core of the square cavities heated from the side walls.

### 2.2 Numerical Studies

The other major body of studies related to buoyancy-driven turbulent flows are those exploring numerical simulation techniques. The numerical prediction of natural or mixed convection can be an economical method to use for design purposes in industrial applications. However, because of the complex nature of such convection flows, their numerical simulation is still a challenge for researchers.

Ince and Launder[21] numerically studied buoyancy-driven flows in rectangular enclosures. In the numerical study, air was the working fluid. They investigated two different aspect ratios, 30:1 and 5:1, which showed different flow structures. The study is covered the range of $Ra$ from $10^6$ to $10^{10}$. Isothermal hot and cold temperatures were imposed to the vertical walls of the cavities while the horizontal walls were adiabatic. The generalized gradient diffusion hypothesis (GGDH) was employed to compute turbulent heat fluxes, which then appear in the buoyancy turbulence generation term. In the GGDH, cross flow gradients are taken into account which might be important in buoyancy driven flows, whereas in the simpler eddy-diffusivity scheme these make no contribution. The other important point in their research was the application of a near wall source term in the $\varepsilon$ equation. The Jones-Launder [22] form of the k-$\varepsilon$ predicts length scales too large at locations near to separation points and consequently gives wrong prediction of wall heat transfer. It was concluded that the Jones-Launder low-Reynolds-number k-$\varepsilon$ model with amendment of GGDH for calculation of turbulent heat fluxes and a near wall source
term in the $\varepsilon$ equation performed satisfactorily in these cases since there was good agreement with experimental data of the thermal and hydraulic fields [14].

Henkes et al.[23] performed numerical investigations on natural convection flow in a square cavity using low-Reynolds-number turbulence models. They carried out two-dimensional calculations in a square cavity which was heated from its vertical walls. The computations covered both laminar and turbulent flow, Rayleigh numbers, ranging up to $10^{14}$ for air and $10^{15}$ for water. Three turbulence models were used in the computations. Those turbulence models were the standard k-$\varepsilon$ model with logarithmic wall function, and the low-Reynolds-number models of Chein [24] and of Jones and Launder [22]. The position of transition from laminar flow to turbulent flow was shown to vary between turbulence models at a fixed Rayleigh number. From comparisons of numerical results to experimental data, it was found that the standard wall function gave too high predictions regarding the average Nusselt number over the hot vertical wall. The low-Reynolds-number models gave Nusselt number results in better agreement with the experimental data. They concluded that the predicted $Ra_{cr}$, which is the point where the flow undergoes transition, is dependent on the turbulence model used. $Ra_{cr} \approx 10^{10}$ was the transition point for the Chein model when air was the working fluid, although $Ra_{cr} \approx 10^{11}$ was the transition point for the Jones and Launder model again when air was the working fluid. When water was the working fluid, giving a higher Prandl number, there was a delay in transition. $Ra_{cr} \approx 10^{12}$ and $Ra_{cr} \approx 10^{13}$ were then the predicted transition points for the Chein and Jones and Launder models respectively. The k-$\varepsilon$ model with standard wall function gave the transition point to turbulence at $Ra_{cr} \approx 10^{9}$ for air and $Ra_{cr} \approx 10^{11}$ for water. They also reported multiple partly turbulent solutions for the low-turbulence cases [14] which were examined in their study. In this context a partly turbulent solution refers to a solution in which there exists turbulent flow in some portion of the computational domain while there is essentially laminar flow in the rest of the domain. Those multiple solutions were from different versions of low-Reynolds-number k-$\varepsilon$ models.
Hanjalić et al.[25] carried out numerical investigations on natural convection in two-dimensional empty and partitioned rectangular enclosures at high Rayleigh numbers. The range of Rayleigh numbers covered in this study was from $10^{10}$ to $10^{12}$. Three and four-equation models, $k$-$\varepsilon$-$\theta^2$ and $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$, were applied to compute the thermo-fluid flow. The computations were performed using a low-Reynolds-number near-wall treatment to resolve changes in the buffer layer. They reported improvements in the turbulence field results by using an extended algebraic model for the turbulent heat fluxes $\overline{u_i\theta}$, as compared with the eddy diffusivity hypothesis. They presented some results regarding mean values and turbulent field and Nusselt number. They concluded that, unlike the eddy diffusivity model, such as typically employed in low-Re-number $k$-$\varepsilon$ models, which predicted the turbulent regime erratically and underpredicted turbulent kinetic energy, the algebraic flux model was capable of predicting turbulence at Ra numbers consistent with experimental findings. They compared the computed Nusselt number results with experimental and heat transfer correlations. Those comparisons proved that the three and four equation model resulted in good agreement with experimental data and correlations, always within 15%.

Dol et al[26] numerically studied turbulent natural convection. They presented a comparison between differential and algebraic second moment closures in the case of natural convection. DNS data of turbulent natural convection between two differentially heated infinite vertical plates at a Rayleigh number $Ra = 5.4 \times 10^5$ (Versteegh and Nieuwstadt [27], Boudjemadi et al. [28]) were employed to carry out the comparisons. They converted the differential equations for $\overline{u_i\theta}$ and $\overline{\theta^2}$ into algebraic forms using relevant hypotheses. Then they carried out computations using the full differential form and a four-equation $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$ algebraic model. They reported reasonable agreement between the algebraic model predictions and available experimental and DNS data. It was shown that the algebraic models predicted well the mean flow and turbulence properties in the tall cavity case [18]. The algebraic model was tested at different levels of truncation. They initially developed the algebraic equation for $\overline{u_i\theta}$ based on the assumption of weak equilibrium, which implies that the anisotropy of the thermal flow $u_i\theta / \left( \overline{\theta^2} k \right)^{1/2}$ is approximately constant in
space and time. Then the full algebraic equation was derived by considering source terms of $k$ and $\overline{\theta^2}$. It was not found necessary to employ full algebraic expression for turbulent heat fluxes which is computationally difficult to use. The reduced expression also captured the major phenomena of turbulent flow by neglecting the source terms of $k$ and $\overline{\theta^2}$.

Kenjereš and Hanjalić [29] numerically studied transient Rayleigh-Bénard convection with a RANS model. They investigated Rayleigh-Bénard (RB) convection using a transient Reynolds-average-Navier-Stokes approach (TRANS) at high Rayleigh numbers. Their aim was to test the RANS method in predicting the structure and large-scale unsteadiness in buoyant-driven turbulent flows. An algebraic low-Reynolds-number $k$-$\varepsilon$-$\overline{\theta^2}$ stress/flux model was employed as numerical model. It showed good performance regarding the near-wall turbulent heat flux and wall heat transfer. The mean flow and turbulence values and wall heat transfer were in good agreement with experimental and DNS data for different Rayleigh numbers and cavity aspect ratios.

Liu and Wen [30] numerically studied buoyancy driven flows in enclosures. They developed and validated an advanced turbulence model for natural convection flows in cavities. A low-Reynolds-number four equation $k$-$\varepsilon$-$\overline{\theta^2}$-$\varepsilon_{\theta}$ model was used to represent the fluid flow and heat transfer in buoyant-flow cavities. To approximate the Reynolds stresses, they employed an eddy viscosity model (EVM) to estimate the mean strain effect on the Reynolds stresses and then they added up a correction in the form of algebraic stress model (ASM) due to buoyancy and near wall effect as:

$$u_i u_j = (u_i u_j)_{EVM} + (u_i u_j)_{ASM}$$

In buoyancy driven flows, there is a strong anisotropy in the Reynolds stresses. That is why the “return to isotropy” term of the pressure strain correlation was considered in the ASM. Then they compared the results of the modified turbulence model with experimental data and the results of the other numerical model [21]. They reported that the modification led to improvement of the results in both tall [14] and square [6] cavities, especially there was significant improvement for vertical velocity fluctuations near the hot and cold
walls. For the Reynolds shear stress, there was improvement in the square cavity case but there was no improvement for the tall cavity case.

Hanjali[31] numerically studied one-point closure models for buoyancy-driven turbulent flows. The thermal buoyancy-driven flows are considered as a challenge for one-point-closure models. The buoyancy-driven flows are difficult to model because of inherent unsteadiness, energy non-equilibrium, counter gradient diffusion and high pressure fluctuations. He investigated some specific turbulence modelling issues regarding buoyant flows within the realm of one-point closures. Firstly, the disadvantages of isotropic eddy-diffusivity models were discussed in the work. A major disadvantage is that they are not capable of capturing interactions of turbulent heat flux and the effects of buoyancy. Another disadvantage is that the alignment of the turbulent heat flux with the mean temperature-gradient vector leads to model failure in many cases such as Rayleigh-Bénard convection. However, 3D and time-dependent simulation using EVM is able to avoid this drawback. Then it was shown that Algebraic models based on a rational truncation of the differential second-moment closure is the minimum closure level for complex flows. The algebraic second moment closure was found not to be adequate to capture the structure of buoyancy driven flows but it was capable of predicting more phenomena than the eddy-diffusivity models. He presented results regarding two generic flows of the side heated vertical channel and Rayleigh-Bénard convection. It was concluded that in these cases the isotropic eddy-diffusivity model leads to wrong results. The algebraic second-moment closure was found to improve the results to some extent but still there was huge discrepancy with DNS data. It was possible to modify the algebraic second-moment closure term by term to re-produce the DNS data but, because of inherent nonlinearity, the generalization to complex flows would be doubtful. Finally he proposed using algebraic truncation of the second moment closure to determine an algebraic equation for the turbulent heat fluxes. It was shown that this will capture a number of internal buoyancy-driven flows.

Hsieh and Lien[32] investigated the numerical modelling of buoyancy-driven turbulent flows in enclosures. They carried out computations for enclosures...
with heated vertical walls. They investigated the tall cavity of Betts and Bokhari [19] at \( Ra = 1.43 \times 10^6 \) and the square cavity of Tian and Karayiannis [5] at \( Ra = 1.58 \times 10^9 \). They realized that the low turbulence buoyancy-driven flow is numerically challenging especially when low-Reynolds-number models were employed. In the cases where the turbulence level in the core of the cavity was low, the low-Re-number models tended to relaminarize the flow near the heated walls. Another difficulty encountered in low turbulence buoyancy driven flows is the inherent unsteadiness. This difficulty would prevent the Reynolds-averaged Navier-Stokes (RANS) solution from converging in a steady state calculation. They used the low-Reynolds-number \( k-\varepsilon \) model of Lien and Leschziner [33], which explicitly included the wall distance. In order to overcome the relaminarization problem, it was found that the two-layer approach (viz, combining the \( k-\varepsilon \) model in the core region with a one-equation \( k-\ell \) model [34] in the near-wall layers) with the interface located approximately at the peaks of velocities parallel to walls was able to trigger the onset of transition and promote the generation of turbulence.

Craft et al.[35], Gerasimov [36], Craft et al.[37], Craft et al.[38] developed new wall-function treatments for turbulent forced and mixed convection flows. There are two major methods to resolve the very complex near wall viscosity affected sub-layer. One method is to employ a very fine near-wall mesh to capture the sharp near-wall changes. The other method is the wall function strategy, which computes the near-wall region from analytical solution of simplified momentum and temperature transport equations. The wall function method is popular in industrial computations because it is an economical method. What will here be referred to as the Standard wall function is widely used for numerical simulation of different test cases. this Standard wall function is based on the assumption that the near wall velocity and temperature distributions match a logarithmic function. In the case of buoyant flows, due to existence of gravity force, logarithmic near-wall variations of velocity and temperature are not generally found in reality. Therefore, the above authors developed two new wall-function strategies based on either analytical or numerical solution of simplified momentum and enthalpy transport equations across the near-wall control volumes. In the analytical approach (AWF), the
equations are not based on local equilibrium and the effects of buoyancy are included. The AWF has been successfully used in the computation of internal mixed convection flows and also for simple external natural convection flows. The numerical wall function is based on the numerical integration of a one-dimensional simplified low-Reynolds-number model over the near-wall control volume. This method produced accurate results in cases including channel flow, impinging jets, rotating disks, backward steps etc, although the numerical wall function has not yet been applied to any buoyant flow.

Omranian et al. [39, 40] explored the potential of the analytical wall function strategy for the economical and reliable prediction of natural convection flows. Three types of test cases were computed; a square cavity with differentially heated vertical walls, a tall cavity with similar heating arrangements and a square cavity with unstable stratification. The simpler, standard wall function strategy, based on the log law and numerically expensive low-Reynolds-number near wall strategy were also employed for baseline comparisons. These near wall modeling strategies were combined with different high-Reynolds-number turbulence models, which included the $k-\varepsilon$, a basic form of second-moment closure, and a more elaborate second-moment closure which satisfies a number of physical realisability constraints, including the 2-component limit. In the second-moment computations, in addition to the generalized gradient diffusion hypothesis, more complex algebraic expressions were also employed for the modeling of the turbulent heat fluxes. These involved the solution of transport equations for the temperature variance and its dissipation rate. The resulting comparisons showed that the more elaborate wall function displayed distinctive predictive advantages. In comparison to the use of low-Re-models it was highly cost effective and in some cases also resulted in superior predictions. The $k-\varepsilon$ model, when used with the new near wall approach, was satisfactory in most cases. Of the second-moment closures, the realizable version, used with the additional transport equations, yielded the most satisfactory flow and thermal predictions.

Addad et al.[41] carried out LES and RANS computations of an inclined tall cavity oriented so that its longest axis made an angle of 5 degrees with the
horizontal axis. This tall cavity was heated from the top wall and cooled from the lower wall. The test case that they studied had \( Ra = 4.16 \times 10^8 \). Then they compared the RANS results to the LES data. They concluded that the RANS models tested all produced Nu numbers very close to the LES data because there is only very weak turbulence in this case and most of the heat is transferred by convective mode rather than turbulence mixing.

Pallares et al.[42] numerically and experimentally studied laminar and turbulent Rayleigh-Bénard convection in a cubical cavity for a Prandtl number of 0.7. LES method was implemented for the numerical investigation. The computations were carried out with Rayleigh numbers as high as \( 10^6 \) and \( 10^8 \). They reported, based on LES and flow visualisation, that there are vortex rings near the horizontal walls and vertical flow exists near the side walls.

It has been shown that the numerical modelling of natural convection flows has been extensively studied during the past decades. Even simple 2D natural convection in a cavity, which seems a simple case geometrically, is a great numerical challenge. As described in the numerical section of the literature review, great progress has been achieved so far. This progress has, however, been largely based on the use of low-Reynolds-number models, which in the case of three-dimensional flows can be prohibitively expensive.
Chapter 3

Turbulence modelling

3.1 Introductory Remarks

This chapter first briefly explains some of the numerical approaches for computation of turbulent flows. Then the main focus will be on the Reynolds-averaged Navier-Stokes (RANS) method which is adopted in this research to resolve natural convection cavity flows. Different RANS models, and some of the latest modifications in these related to thermo-fluids applications used in the present work, are explained in detail.

3.2 Mean Flow Equations and their solution

There are three major numerical methods for predicting turbulent flows.

- Direct Numerical Simulation (DNS): this is the most accurate method for computing turbulent flows by directly solving the Navier-Stokes equations. It is also conceptually the simplest method. In the DNS method, the Navier-Stokes equations are solved with very fine spatial and temporal resolution in order to capture all the scales of the turbulent eddies. Based on current computer resources, only flows at low Reynolds numbers can be simulated by this method. The DNS results are valid like measurements from experimental data and it is capable of resolving all motions in the flow. DNS produces very detailed data of flows but it is computationally too expensive an approach for use in design purposes.
• Large eddy simulation (LES): in this approach, the largest scale turbulent motions of the flow are resolved but approximations are used to account for the effect of the small scale motions. LES is less accurate than DNS but it is numerically more economic. In the cases of low Reynolds number and availability of enough computational resources, DNS can be used but in many cases such as high Reynolds number or complex geometry LES is a more cost efficient method.

• Reynolds averaged Navier-Stokes (RANS): This method is based on averaging the equations of motion. This approach is also called one-point closure. A set of differential equations results from averaging the equations of motion. This set of differential equations is called the Reynolds averaged Navier-Stokes (RANS) equations. This method is based on decomposition of the turbulent flow quantities into mean and fluctuating parts. The mean flow is calculated using the RANS equations. The terms involving correlations of the fluctuating parts have to be modelled. The RANS approach is computationally economical and is popular in industry for design purposes.

In the RANS approach, the set of equations is as follows:

The continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U_j)}{\partial x_j} = 0 \quad (3.1)$$

momentum equations:

$$\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \overline{u_i u_j} \right] + B_{Ti} \quad (3.2)$$

In equation (2.1), $B_{Ti}$ is the buoyancy term and is equal to $g_i (\rho - \rho_{ref})$. In flows that contain heat transfer, the fluid properties change with temperature. In cases where density changes with temperature are not high, it is possible to neglect the variation of density in the unsteady and convection terms, and retain its effect only in the gravitational term. This is called the Boussinesq approximation. Then the gravitational term $(\rho - \rho_{ref})$ can be written as $-\rho_{ref} g_i \beta (\Theta - \Theta_0)$ where $\beta$ is the coefficient of volumetric expansion. In the
test cases considered in the present study, the temperature differences are 2°C, 18°C and 34°C and the working fluid is air. Based on the gas law, \( \rho \) is proportional to \( 1/\Theta \), and even a 34°C temperature difference would cause the air density change by only approximately 15 percent. Therefore molecular transport properties are taken as uniform in the computations, following the above Boussinesq approximation.

The mean temperature \( \Theta \) is obtained from the enthalpy transport equation:

\[
\frac{\partial (\rho \Theta)}{\partial t} + \frac{\partial (\rho U_j \Theta)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial \Theta}{\partial x_j} - \rho u_j \theta \right]
\] (3.3)

However, this set of differential equations is not a closed set. In order to close the set of equations, it is required to use turbulence models to approximate the Reynolds stresses \( \overline{u_i u_j} \) and turbulent heat fluxes \( \overline{u_j \theta} \).

### 3.3 RANS Turbulence models

As mentioned in the previous sections, the RANS turbulence modelling is the approach which is adopted to resolve the buoyant cavity flows in the present research. As noted above, the RANS equations are not a closed set of equations. The turbulence models which have been used in this research to close the set of equations are explained in detail in this section.

#### 3.3.1 High-Reynolds Number \( k-\epsilon \) Model

The \( k-\epsilon \) model is among the category of two-equation models in which two properties of turbulence, in this case \( k \) and \( \epsilon \), are solved for. Then the other variables such as length scale \( (l = k^{3/2}/\epsilon) \) and timescale \( (\tau = k/\epsilon) \) and turbulent viscosity \( \nu_t \propto (k^2/\epsilon) \) can be calculated.

In laminar flows, energy dissipation and transport of mass, momentum and energy normal to the streamlines take place by viscosity. Therefore, it might be a reasonable assumption that the mixing effect of turbulence would increase the diffusive transport of momentum and thermal energy. This leads to the eddy-viscosity model for the Reynolds stress as:
\[ \rho \overline{u_i u_j} = \frac{2}{3} \rho k \delta_{ij} - \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3.4) \]

and the effective eddy-diffusivity model for temperature:

\[ \rho u_i \theta = -\frac{\mu_t}{Pr_t} \frac{\partial \Theta}{\partial x_i} \quad (3.5) \]

where \( Pr_t = 0.9 \). In the \( k-\varepsilon \) model, the eddy viscosity is expressed as:

\[ \mu_t = \rho c_{\mu} k^2 \varepsilon \quad (3.6) \]

where \( k \) and \( \varepsilon \) are obtained from solving the transport equation for turbulent kinetic energy, \( k \):

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + G_k - \rho \varepsilon \quad (3.7) \]

and the modelled transport equation for the dissipation rate, \( \varepsilon \):

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + c_{\varepsilon 1} \frac{\varepsilon}{k} (P_k + G_k) - c_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (3.8) \]

The \( P_k \) term in the \( k \) and \( \varepsilon \) equations represents the rate of production of turbulent kinetic energy. In other words, the transfer of kinetic energy from the mean flow to the turbulence due to mean strains:

\[ P_k = -\rho \overline{u_i u_j} \frac{\partial U_i}{\partial x_j} \quad (3.9) \]

\( G_k \) is the term that accounts for the generation, or destruction, of turbulence by means of body-force fluctuations. In the present work this is due to buoyancy force fluctuations in a variable density fluid in a gravitational field:

\[ G_k = \rho g_i \overline{u_i u_j} = -\rho \beta g_i u_i \theta \quad (3.10) \]

In the case that an effective eddy-diffusivity model(equation 3.5) is employed, the buoyancy generation term \( G_k \) becomes:

\[ G_k = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial \Theta}{\partial x_i} \quad (3.11) \]
The turbulent kinetic energy, $k$, is related to the normal Reynolds stresses by

$$k = \frac{u^2 + v^2 + w^2}{2} \quad (3.12)$$

In table 3.1, all the constants and functions of the high-Reynolds-number $k-\varepsilon$ model are listed.

<table>
<thead>
<tr>
<th>$c_\mu$</th>
<th>$c_{\varepsilon 1}$</th>
<th>$c_{\varepsilon 2}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_{\varepsilon}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 3.1 – Values of constants in High-Re $k-\varepsilon$ model.

Effect of molecular viscosity is not considered in the high-Reynolds-number $k-\varepsilon$ model therefore can not be integrated right across the wall sub-layer. The wall-function method is used to provide wall boundary conditions and include the effect of molecular viscosity. These will be discussed in the next Chapter in detail.

### 3.3.2 Further modification in High-Re $k-\varepsilon$ model for 3D simulations

In cases where there are large strain rates equation 3.4 can become non-realizable, for example producing negative value for normal stresses. It can also return a stress field that violates the Schwarz inequality of equation 3.13:

$$u_i u_j^2 \leq \overline{u_i^2 u_j^2} \quad (3.13)$$

when $i \neq j$. To prevent the High-Re $k-\varepsilon$ from producing non-realizable results in circumstances that strong shear flow exists, the following constraint can be used for calculation of the turbulent viscosity in 3D simulations[43]:

$$\mu_t = \min \left\{ \frac{k^2}{c_\mu \varepsilon}, \frac{2/3k}{\max \left( \left| \frac{\partial u}{\partial y} + \frac{\partial V}{\partial z} \right|, \left| \frac{\partial u}{\partial z} + \frac{\partial W}{\partial x} \right|, \left| \frac{\partial V}{\partial z} + \frac{\partial W}{\partial y} \right| \right)} \right\} \quad (3.14)$$
3.3.3 Low-Reynolds Number $k\text{-}\epsilon$ model

In the low-Reynolds-number $k\text{-}\epsilon$ model, the transport equations can be integrated up to the wall. The constants and coefficients are adjusted from those in the high Reynolds number $k\text{-}\epsilon$ model in order to account for the influence of viscosity and other near-wall effects.

In the Launder-Sharma \[44\] low-Reynolds-number $k\text{-}\epsilon$ model, the transport equation for turbulent kinetic energy, $k$, is written as:

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial x_j}\right] + P_k + G_k - \rho \tilde{\varepsilon} + D \tag{3.15}
$$

and the transport equation for the dissipation rate, $\tilde{\varepsilon}$:

$$
\frac{\partial (\rho \tilde{\varepsilon})}{\partial t} + \frac{\partial (\rho U_j \tilde{\varepsilon})}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right) \frac{\partial \tilde{\varepsilon}}{\partial x_j}\right] + c_\varepsilon f_1 \tilde{\varepsilon} \frac{k^2}{\tilde{\varepsilon}} (P_k + G_k) - c_\varepsilon f_2 \rho \tilde{\varepsilon}^2 k + E + S_\varepsilon \tag{3.16}
$$

In this model, the eddy viscosity is expressed as:

$$
\mu_t = \rho c_\mu f_\mu \frac{k^2}{\tilde{\varepsilon}} \tag{3.17}
$$

The constants and functions of the model are given below:

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

$$
f_1 = 1.0
$$

$$
f_2 = 1 - 0.3 \exp\left(-Re_t^2\right)
$$

The constants $C_\mu$, $C_{\varepsilon_1}$ and $C_{\varepsilon_2}$ in the high-Reynolds-number $k\text{-}\epsilon$ model are respectively multiplied by $f_\mu$, $f_1$ and $f_2$. These wall damping functions are in turn functions of turbulent Reynolds number $Re_t$: 
The $D$ term in the $k$ transport equation is designed so that the homogeneous dissipation rate $\tilde{\varepsilon}$ on the wall is zero, which makes the boundary condition more convenient than that for $\varepsilon$.

\[
D = -2\mu \left( \frac{\partial \sqrt{k}}{\partial x_j} \right)^2
\]

then

\[
\varepsilon = \tilde{\varepsilon} + D
\]

Therefore the $k$-$\varepsilon$ model (Launder and Sharma[44]) solves an equation for “isotropic” dissipation rate instead of the true dissipation rate. Hence the wall boundary condition for $\tilde{\varepsilon}$ is $\tilde{\varepsilon} = 0$.

The $E$ term is a source term in the transport equation of dissipation rate $\tilde{\varepsilon}$ that is influential only in the viscous sub layer:

\[
E = 2\mu t \rho \left( \frac{\partial^2 U_i}{\partial x_j \partial x_k} \right)^2
\]

Another source term is also added to $\tilde{\varepsilon}$ transport equation. This source term corrects the length scales near the wall where low-Reynolds-number $k$-$\varepsilon$ models tend to produce too large values in flows involving separation and re-attachment. Yap [45] proposed an equation for the correction of the length scale near walls of the form ($y$ is distance from the walls):

\[
S_{\varepsilon} = 0.83 \left( \frac{k^{3/2}}{\tilde{\varepsilon} c_l y} - 1 \right) \left( \frac{k^{3/2}}{\tilde{\varepsilon} c_l y} \right)^2 \frac{\varepsilon^2}{k}
\]

and this is adopted in the present work.

### 3.3.4 Approximation of turbulent heat flux $\overline{u_i \theta}$

In the two equation turbulence models, $\overline{u_i \theta}$ is usually calculated by algebraic equations instead of solving transport equations for $\overline{u_i \theta}$. In an EVM, the eddy diffusivity equation (equation 3.5) is often used to calculate $\overline{u_i \theta}$. The eddy
The diffusivity model is rather simple scheme which calculates $\overline{u_i}\theta$ based on the temperature gradient in the same direction as the turbulent heat flux. When the effective eddy-diffusivity is used to approximate the turbulent heat fluxes, the buoyancy generation vanishes if there is no vertical temperature gradient. In simple shear flows with only a cross flow temperature gradient, the streamwise heat flux is, in fact, significantly larger in magnitude than cross stream heat flux. However, in this case the eddy diffusivity model produces zero turbulent heat flux in the streamwise direction in a fully developed flow in which the streamwise temperature gradient is zero. This error in modeling the streamwise heat flux is not influential in the computation of the mean temperature because it is the gradient of the heat flux that appears in the energy transport equation (and the streamwise contribution is therefore zero). In the turbulence generation due to buoyancy, $G_k$, however the turbulent heat flux appears directly, not its gradient. Depending on the orientation of the flow and heating, the streamwise component of the turbulent heat flux can make a significant contribution to $G_k$.

The generalized gradient diffusion hypothesis (GGDH) can be employed to give a better approximation of the streamwise component of the turbulent heat flux. The GGDH is formulated as:

$$\overline{u_i}\theta = -c_\theta \frac{k\kappa}{\varepsilon} u_i u_k \frac{\partial \Theta}{\partial x_k}$$  \hspace{1cm} (3.19)

The GGDH is developed to take into account more influential parameters in the calculation of $\overline{u_i}\theta$ such as the Reynolds stresses. But in a buoyancy-driven flow, the GGDH does not take into account all of the important parameters on which $\overline{u_i}\theta$ depends, such as gravity and mean velocity gradients.

In order to include some influence of buoyancy effects on the turbulent heat fluxes, an extended algebraic model was proposed in [25]:

$$\overline{u_i}\theta = -c_\theta \frac{k\kappa}{\varepsilon} \left( u_i u_j \frac{\partial \Theta}{\partial x_j} + \xi u_i \theta \frac{\partial U_i}{\partial x_j} + \eta \beta g_i \theta^2 \right)$$  \hspace{1cm} (3.20)

This equation is derived from simplification of the transport equation of
The temperature variance $\theta^2$ appears inside this equation. Thus, it is necessary to solve a transport equation for $\theta^2$ (equation 3.21). Then there are two methods to calculate dissipation rate of temperature variance, $\varepsilon_\theta$ which are described in the section 3.3.5.

### 3.3.5 Low-Reynolds-Number Three and Four Equations model

$k$-$\varepsilon$-$\theta^2$ and $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$

It is worth noting that even flow in undivided two-dimensional enclosures shows some characteristics related to complex buoyant flows. There are two distinct regions in buoyant cavity flows. One is the boundary layer region along the walls and the other is the circulation region of fluid in the core. They interact at their interface. They possess different turbulent structures and turbulent scales. The circulating flow in the core can be a single cell or a multi-cell structure, depending on thermal boundary conditions, enclosure aspect ratio and $Ra$ value.

Another complexity of buoyant cavity flows is the transition to turbulence. In practical cases, such as flow in building structures and the cooling of nuclear reactors, etc., there is significant turbulent flow only in some parts of the domain. Therefore prediction of transition locations, and also the decay of turbulence away from the wall, is another challenge of turbulence modelling for buoyant flows.

Three- and four-equation models, $k$-$\varepsilon$-$\theta^2$ and $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$ schemes, have been proposed by Hanjalic et al.[25] which are presented in this section. The equations for $k$ and $\varepsilon$ are the same as those presented in Section 3.3.3, whilst transport equations were also introduced for $\theta^2$ and $\varepsilon_\theta$.

\[
\frac{\partial (\rho \theta^2)}{\partial t} + \frac{\partial (\rho U_j \theta^2)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \theta^2}{\partial x_j} \right] + 2 P_\theta - 2 \rho \varepsilon_\theta \quad (3.21)
\]
\[
\frac{\partial (\rho \varepsilon_\theta)}{\partial t} + \frac{\partial (\rho U_j \varepsilon_\theta)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon_\theta}{\partial x_j} \right] \\
+ c_{\varepsilon 1}^\theta \frac{\varepsilon_\theta}{k} P_k + c_{\varepsilon 3}^\theta \frac{\varepsilon_\theta}{\theta^2} P_\theta - c_{\varepsilon 2}^\theta \frac{\varepsilon_\theta}{\theta^2} - c_{\varepsilon 4}^\theta \frac{\varepsilon_\theta}{k} + E_\theta
\]

(3.22)

For the simpler three-equation model \( \varepsilon_\theta \) is obtained algebraically, instead of solving equation 3.23:

\[ \varepsilon_\theta = \frac{\theta^2 \varepsilon}{k} \]  

(3.23)

The term \( P_\theta \) is the production of temperature variance \( \overline{\theta^2} \). The term \( E_\theta \) is related to molecular effects. The turbulent diffusion part of the total diffusion is modelled based on the effective eddy diffusivity hypothesis.

\[ P_\theta = -\rho u_j \theta \frac{\partial \Theta}{\partial x_j} \]  

(3.24)

\[ E_\theta = 2 \rho \alpha_t \left( \frac{\partial^2 \Theta}{\partial x_j \partial x_k} \right)^2 \]  

(3.25)

\( \varepsilon_\theta \) is the total dissipation rate:

\[ \varepsilon_\theta = \overline{\varepsilon_\theta} + \alpha \left( \frac{\partial \left( \theta^2 \right)^{1/2}}{\partial x_n} \right)^2 \]  

(3.26)

The turbulent heat fluxes \( \overline{u_i \theta} \), are calculated by the equation 3.20 and the remaining coefficients employed in the turbulence model are shown in the table 3.2.

<table>
<thead>
<tr>
<th>( c_\theta )</th>
<th>( \xi )</th>
<th>( \eta )</th>
<th>( c_\varepsilon )</th>
<th>( c_{\varepsilon 1} )</th>
<th>( c_{\varepsilon 2} )</th>
<th>( c_{\varepsilon 3} )</th>
<th>( c_{\varepsilon 4} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>0.6</td>
<td>0.6</td>
<td>0.07</td>
<td>1.44</td>
<td>1.92</td>
<td>0.07</td>
<td>0.72</td>
</tr>
<tr>
<td>2.2</td>
<td>1.3</td>
<td>0.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3.2 – the coefficients for the \( k-\varepsilon-\overline{\theta^2}-\varepsilon_\theta \) turbulence model.
### 3.3.6 High-Reynolds-Number Three and Four Equations model

\[ k-\varepsilon-\theta^2 \text{ and } k-\varepsilon-\theta^2-\theta \]

In this research, a high-Reynolds-number version of the model presented in the previous section has been employed. The equations are similar to those shown in previous section apart from the terms and functions directly involving molecular viscosity, which are simplified to the forms shown in the table 3.3.

\[
\begin{array}{cccc}
 f_\mu & f_2 & E & E_\theta \\
 1 & 1 & 0 & 0 \\
\end{array}
\]

Table 3.3 – the coefficients for the high-Reynolds-number \( k-\varepsilon-\theta^2-\varepsilon_\theta \) turbulence model.

The AWF has been adopted to provide appropriate wall boundary conditions with these high Reynolds number models. All the wall function equations used in the high-Reynolds-number models are presented in detail in the next chapter.

### 3.3.7 Differential Stress Closure

In the Reynolds-stress transport models, transport equations are solved for the Reynolds stresses \( \overline{u_i u_j} \) and the dissipation rate \( \varepsilon \). Therefore one of the main drawbacks of EVMs, which is the turbulent-viscosity hypothesis, is removed in the differential stress closure.

Eddy-viscosity models have significant disadvantages which originate from the eddy-viscosity assumption. This is mainly designed to give the shear stress in a shear dominated flow (normal stresses are still wrong). But usually in three-dimensional cases, the relation between Reynolds stress and strain rate is not as simple as implied by the eddy-viscosity hypothesis. One route to overcome these problems is by solving transport equations for each element of the Reynolds stress tensor \( \rho \overline{u_i u_j} \). The transport equations for the Reynolds
stresses can be derived from the Navier-Stokes equations and are written in the form:

\[
\frac{D (\rho u_i u_j)}{Dt} = \rho P_{ij} + \rho G_{ij} + \rho \phi_{ij} + d_{ij} - \rho \varepsilon_{ij}
\]  

(3.27)

The first two terms are production terms due to the mean strain rate, \( P_{ij} \), and buoyancy force, \( G_{ij} \). These two terms do not need any modelling:

\[
P_{ij} = - \left( \frac{u_i u_k}{u_j} \frac{\partial U_j}{\partial x_k} + \frac{u_j u_k}{u_i} \frac{\partial U_i}{\partial x_k} \right)
\]  

(3.28)

\[
G_{ij} = - \beta \left( g_i u_j \theta + g_j u_i \theta \right)
\]  

(3.29)

the next term in equation 3.27 is:

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

(3.30)

The \( \phi_{ij} \) term is usually called the pressure-strain term. It is responsible for the redistribution of turbulent kinetic energy, \( k \), among the components of the Reynolds stress tensor. This term does not change the total kinetic energy value directly. The pressure-strain term needs modelling, which will be explained in detail later.

The next term is:

\[
d_{ij} = - \frac{\partial}{\partial x_k} \left( \rho u_i u_j u_k + p u_j \delta_{ik} + p u_i \delta_{jk} - \mu \frac{\partial u_i u_j}{\partial x_k} \right)
\]  

(3.31)

This term is diffusion. The last term is:

\[
\varepsilon_{ij} = -2\nu \frac{\partial u_i}{\partial x_k} \frac{\partial u_j}{\partial x_k}
\]  

(3.32)

This is the dissipation rate tensor. The turbulent diffusion and dissipation tensor both need to be modelled.

In the diffusion term, there are three parts. The viscous diffusion term is exact. The pressure diffusion terms are often assumed to be negligible, other than very close to walls. The remaining term is the triple moment correlation \( u_i u_j u_k \). In the present work, this correlation is modelled using the generalized
gradient diffusion hypothesis (GGDH) of Daly and Harlow. In this approximation, the flux of quantity $\phi$ is modelled through:

$$u_k \phi \propto -\frac{k}{\varepsilon} u_k u_l \frac{\partial \phi}{\partial x_l}$$  (3.33)

Applying this approximation to the triple moment correlation results in:

$$u_k u_i u_j \propto -\frac{k}{\varepsilon} u_k u_l \frac{\partial u_i u_j}{\partial x_l}$$  (3.34)

Introduction of the above approximation to (3.31) and canceling the second and third terms (assuming them to be negligible compared to the others) results in the following equation for the diffusion process:

$$d_{ij} = \frac{\partial}{\partial x_k} \left( c_s \rho \frac{k}{\varepsilon} u_k u_l \frac{\partial u_i u_j}{\partial x_l} + \mu \frac{\partial u_i u_j}{\partial x_k} \right)$$  (3.35)

the coefficient $c_s$ is taken equal to 0.22.

There are two major modelling practices for the pressure-strain term $\phi_{ij}$ and the dissipation tensor $\varepsilon_{ij}$ used in the present work that will be discussed later.

To close the set of equations, it is required to solve a transport equation for the energy dissipation rate $\varepsilon$, in addition to the transport equations for the Reynolds stresses. The transport equation for the energy dissipation rate, $\varepsilon$, is of the same general form as the one employed in the $k-\varepsilon$ closure:

$$\frac{D (\rho \varepsilon)}{Dt} = \frac{\rho \varepsilon^2}{k} \left( c_{\varepsilon 1} \frac{P}{\varepsilon} - c_{\varepsilon 2} \right) + d_{\varepsilon}$$  (3.36)

Its diffusion term is modelled using the generalized gradient diffusion hypothesis (GGDH):

$$d_{\varepsilon} = \frac{\partial}{\partial x_k} \left( c_{\varepsilon} \rho \frac{k}{\varepsilon} u_k u_l \frac{\partial \varepsilon}{\partial x_l} + \mu \frac{\partial \varepsilon}{\partial x_k} \right)$$  (3.37)

where $c_{\varepsilon}$, $c_{\varepsilon 1}$ and $c_{\varepsilon 2}$ are taken 0.16, 1.45 and 1.90, respectively.

In order to model the turbulent heat fluxes, $u_i \theta$, there are two main options.
One option is to derive and solve modelled transport equations for the turbulent heat fluxes, in a similar way as outlined above for the Reynolds stresses. The other way is to employ algebraic relations for the turbulent heat fluxes, similar to those adopted within an eddy-viscosity based scheme. These two alternatives are presented below.

The correlation $u_i\theta$ represents the heat flux in the direction of $x_i$ by turbulent fluctuations. The exact transport equation of $u_i\theta$ can be derived by multiplying the transport equation for the temperature fluctuations by the $u_i$ velocity component adding it to the $x_i$-component of the Navier-Stokes equations multiplied by $\theta$, and then averaging. The exact form of the $u_i\theta$ transport equation is then obtained as:

$$\frac{D (\rho u_i \theta)}{Dt} = \rho P_{i\theta} + \rho G_{i\theta} + \rho \phi_{i\theta} + d_{i\theta} - \rho \varepsilon_{i\theta} \quad (3.38)$$

$P_{i\theta}$ expresses the rate of generation of $u_i\theta$ by mean velocity and mean scalar gradients. The mean velocity gradient tends to increase velocity fluctuations and the mean scalar gradient increases the magnitude of the temperature fluctuations:

$$P_{i\theta} = - \left( \frac{1}{2} u_i u_k \frac{\partial \Theta}{\partial x_k} + u_k \theta \frac{\partial U_i}{\partial x_k} \right) \quad (3.39)$$

The $G_{i\theta}$ term represents the generation of $u_i\theta$ as a result of buoyancy forces:

$$G_{i\theta} = - \beta g_i \theta^2 \quad (3.40)$$

The pressure temperature gradient correlation, $\phi_{i\theta}$, is similar to the pressure strain term in the stress equations:

$$\phi_{i\theta} = \frac{\rho \partial \theta}{\rho \partial x_i} \quad (3.41)$$

The diffusion term is:

$$d_{i\theta} = - \frac{\partial}{\partial x_k} \left( \rho u_k u_i \theta + \rho t \delta_{ik} - \Gamma u_i \frac{\partial \theta}{\partial x_k} - \mu \theta \frac{\partial u_i}{\partial x_k} \right) \quad (3.42)$$

The dissipation rate of $u_i\theta$ is:
Turbulence modelling

\[ \varepsilon_{i\theta} = (\alpha + \nu) \frac{\partial \theta}{\partial x_k} \frac{\partial u_i}{\partial x_k} \quad (3.43) \]

The dissipation rate, \( \varepsilon_{i\theta} \), is zero in isotropic turbulence and it is generally assumed to be negligible if the Reynolds number is high, even in non-isotropic turbulence.

With suitable models for \( \phi_{i\theta} \), \( d_{i\theta} \) and \( \varepsilon_{i\theta} \) equation 3.38 can be solved for the heat fluxes. However, in the present work the simpler, and computationally cheaper, route of employing algebraic forms such as those introduced in section 3.3.4 has been adopted instead. Nevertheless, equation 3.38 highlights the physical generation process affecting \( u_i \theta \), allowing to see the simplification involved in the algebraic models of equations 3.19 and 3.20 in section 3.3.4.

In stratified flows, \( \overline{\theta^2} \) appears in the buoyant source term \( G_{i\theta} \). A transport equation can be obtained for the scalar variance \( \theta^2 \) by multiplying the equation of the temperature fluctuations by \( \theta \) and then averaging. The resulting equation is:

\[ \frac{D \left( \rho \overline{\theta^2} \right)}{D t} = \rho P_{\theta} + d_{\theta} - \rho \varepsilon_{\theta} \quad (3.44) \]

The change in \( \overline{\theta^2} \) is thus seen to be governed by the production term \( P_{\theta} \), dissipation rate term \( \varepsilon_{\theta} \) and turbulent diffusion term \( d_{\theta} \).

\( P_{\theta} \) is the generation rate of temperature fluctuations by means of temperature gradients:

\[ P_{\theta} = -2 \overline{u_k \theta} \frac{\partial \theta}{\partial x_k} \quad (3.45) \]

\( d_{\theta} \) is the diffusion term and represents diffusive transport caused by velocity fluctuations and molecular diffusion:

\[ d_{\theta} = -\frac{\partial}{\partial x_k} \left( \rho u_k \overline{\theta^2} - \Gamma \frac{\partial \overline{\theta^2}}{\partial x_k} \right) \quad (3.46) \]

\( \varepsilon_{\theta} \) is the dissipation rate of temperature variance:
\[
\varepsilon_\theta = 2\alpha \frac{\partial \theta}{\partial x_k} \frac{\partial \theta}{\partial x_k} \quad (3.47)
\]

**Basic Differential Stress Model Closure**

As mentioned earlier the pressure-strain \( \phi_{ij} \) and dissipation rate \( \varepsilon_{ij} \) require modelling. There are two different modelling alternatives for these terms used in this study. One is called the basic second-moment-closure model and the other one is called the two-component-limit model. First, we discuss the basic model.

Modelling of the pressure-strain term is very important. This term can be split into three parts:

\[
\phi_{ij} = \phi_{ij1} + \phi_{ij2} + \phi_{ij3} \quad (3.48)
\]

\( \phi_{ij1} \) represents turbulence-turbulence interactions and is called the slow pressure-strain term. \( \phi_{ij2} \) is associated with mean strain terms and is called the rapid pressure-strain term. \( \phi_{ij3} \) is associated with the force fields such as buoyancy.

The \( \phi_{ij1} \) term is also called the “return to isotropy” term and it is responsible for the redistribution of the energy in order to decrease the anisotropy of the Reynolds stresses. Its modelling has been suggested by Rotta [46]:

\[
\phi_{ij1} = -c_1 \varepsilon a_{ij} \quad (3.49)
\]

where \( a_{ij} \) is:

\[
a_{ij} = \frac{u_i u_j}{k} - \frac{2}{3} \delta_{ij} \quad (3.50)
\]

The \( \phi_{ij2} \) term is often modelled as acting to reduce the anisotropy of the stress production. Its modelling has been suggested by Naot et al [47] as:

\[
\phi_{ij2} = -c_2 \left( P_{ij} - \frac{1}{3} P_{kk} \delta_{ij} \right) \quad (3.51)
\]
In buoyancy driven flows, there is a buoyancy generation term and therefore an extra term proposed by Launder[48] similar to $\phi_{ij2}$ in order to reduce anisotropy of the buoyant stress production:

$$
\phi_{ij3} = -c_3 \left( G_{ij} - \frac{1}{3} G_{kk} \delta_{ij} \right) \quad (3.52)
$$

The three terms of $\phi_{ij}$ are re-distributive among the components of the stress tensor.

The above model forms do not account for some of the important effects found due to the proximity of a wall. Near a wall, turbulent pressure fluctuations are reflected and this leads to a reduction of the stress component normal to the wall. The standard form of $\phi_{ij}$, as given above, does not reproduce this effect. Therefore another term is added to $\phi_{ij}$ to account for the wall reflection process:

$$
\phi^w_{ij} = \phi^w_{ij1} + \phi^w_{ij2} \quad (3.53)
$$

The $\phi^w_{ij1}$ term was suggested by Shir[49] and it modifies the slow part of the pressure-strain correlation:

$$
\phi^w_{ij1} = c_{1w} \frac{\varepsilon}{k} \left( u_{il} u_{j} u_{lm} n_{m} \delta_{ij} - \frac{3}{2} u_{ij} n_{l} n_{l} - \frac{3}{2} u_{il} u_{jn} n_{l} \right) \left( \frac{l}{2.5y} \right) \quad (3.54)
$$

The $\phi^w_{ij2}$ term was suggested by Gibson and Launder[50] and it modifies the rapid part of the pressure-strain correlation:

$$
\phi^w_{ij2} = c_{2w} \left( \phi_{lj2} n_{l} n_{j} \delta_{ij} - \frac{3}{2} \phi_{lj2} n_{l} n_{l} - \frac{3}{2} \phi_{lj2} n_{l} n_{l} \right) \left( \frac{l}{2.5y} \right) \quad (3.55)
$$

In $\phi^w_{ij}$, $n_i$ is the $i^{th}$ component of the unit vector normal to the wall, $l$ is the turbulent length scale ($k^{3/2}/\varepsilon$) and $y$ is the distance from the wall.

The model coefficients required in the basic Reynolds-stress closure are shown in the table 3.4.
Table 3.4 – The coefficients required in the basic Reynolds-stress closure.

<table>
<thead>
<tr>
<th></th>
<th>$c_1$</th>
<th>$c_2$</th>
<th>$c_3$</th>
<th>$c_{1w}$</th>
<th>$c_{2w}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.8</td>
<td>0.6</td>
<td>0.5</td>
<td>0.5</td>
<td>0.3</td>
</tr>
</tbody>
</table>

It has been reported [51] that $\phi_{ij}^w$ in the form shown above does not perform well in all situations. It gives satisfactory results for flows parallel to single walls, but in an impinging jet flow, for example, it does not work well. Its unsatisfactory behavior can be explained as follows. In a near wall shear flow, $\phi_{ij2}$ tends to decrease a component of production of the stress dependent on anisotropy of $P_{ij}$ and to redistribute it between the other components of stress. The $\phi_{ij2}^w$ acts against this process and tends to reduce stress normal to the wall. In an impinging flow, the production term of the stress equation tends to increase the component of stress normal to the wall and $\phi_{ij2}$ is designed to redistribute energy, so tends to reduce the stress normal to the wall. As mentioned before, $\phi_{ij2}^w$ acts against this process but this time tends to increase stress normal to the wall which is against the concept of the wall reflection terms.

Another form of $\phi_{ij2}^w$ was suggested by Craft and Launder[52] to work well in impinging flow as well as simple shear:

$$
\phi_{ij2}^w = -0.08 \frac{\partial U_l}{\partial x_m} u_l u_m (\delta_{ij} - 3n_in_j) \left( \frac{l}{2.5y} \right) \\
-0.1k a_{lm} \left( \frac{\partial U_k}{\partial x_m} n_l n_k \delta_{ij} - 3 \frac{\partial U_i}{\partial x_m} n_l n_j - 3 \frac{\partial U_j}{\partial x_m} n_l n_i \right) \left( \frac{l}{2.5y} \right) \\
+0.4k \frac{\partial U_i}{\partial x_m} n_l n_m \left( n_i n_j - \frac{1}{3} \delta_{ij} \right) \left( \frac{l}{2.5y} \right) 
$$

(3.56)

then this form of wall reflection term is used in the present study.

The other term which needs modelling is $\varepsilon_{ij}$. In high Reynolds number flows, because of local isotropy, $\varepsilon_{ij}$ can be modelled as follows:

$$
\varepsilon_{ij} = -2\nu \frac{\partial u_i}{\partial x_i} \frac{\partial u_j}{\partial x_k} \approx \frac{2}{3} \varepsilon \delta_{ij} 
$$

(3.57)
For moderate to low Reynolds-number flows, this is not an accurate representation of dissipation rate. To some extent, because the anisotropic component, $\varepsilon_{ij} - \frac{1}{2}\varepsilon\delta_{ij}$ has some of the same mathematical properties as the pressure-strain term, it can be argued that the modelling of the pressure-strain term can be interpreted as compensating for the lack of accuracy in the local isotropy assumption.

**Two-Component-Limit Version**

The basic Reynolds-stress closure can capture many important physical processes and is more general than the EVM, but it has some disadvantages as well.

One important weakness of the basic Reynolds-stress closure is that it does not work properly in the situation of the two component limit. The two component limit refers to a situation where one of the normal stress components becomes zero. For instance, this may occur in free shear flows at the region between the stagnant fluid and the jet of fluid. The other disadvantage of the basic Reynolds-stress closure is that $\phi_{ij}$ requires corrections in near-wall regions to reduce the stress normal to the wall. The wall reflection terms developed for this correction are calibrated for plane surfaces and are dependent on the ratio of the turbulent length scale to the normal distance from the wall. In the case of complex geometries, it may be difficult to define the normal distance to the wall.

An alternative method is to develop a more general pressure-strain term so that it can comply with the two component limit situation. An added benefit may be that there is no need to apply a wall correction term.

The $\phi_{ij1}$ term of the TCL pressure-strain model was presented by Craft et al. [53] as:

$$
\phi_{ij1} = -c_1 \varepsilon \left[ a_{ij} + c_1' \left( a_{ik} a_{jk} - \frac{1}{3} A_2 \delta_{ij} \right) \right] - A^{0.5} \varepsilon a_{ij}
$$

(3.58)

where $A$ is termed the flatness factor and $A_2$ and $A_3$ are stress invariants. They
are defined as:

\[ A = 1 - \frac{9}{8} (A_2 - A_3) \]

\[ A_2 = a_{ij} a_{ij} \]

\[ A_3 = a_{ij} a_{jk} a_{ki} \]

\( a_{ij} \) in the above equations is the stress anisotropy tensor and it is defined as:

\[ a_{ij} = \frac{u_i u_j}{k} - \frac{2}{3} \delta_{ij} \]

The \( \phi_{ij2} \) formulation was proposed by Fu [54] and he reported that the model produces satisfactory results when one of the fluctuating velocity components goes to zero:

\[ \phi_{ij2} = -0.6 \left( P_{ij} - \frac{1}{3} P_{kk} \delta_{ij} \right) + 0.3 \varepsilon a_{ij} \left( \frac{P_{kk}}{\varepsilon} \right) - 0.2 \left[ \frac{u_i u_j u_l u_l}{k} \left( \frac{\partial U_k}{\partial x_l} + \frac{\partial U_l}{\partial x_k} \right) - \frac{u_i u_k}{k} \left( \frac{\partial U_j}{\partial x_l} + \frac{\partial U_l}{\partial x_k} \right) \right] \]

\[ -c_2 \left( A_2 (P_{ij} - D_{ij}) + 3 a_{mi} a_{nj} (P_{mn} - D_{mn}) \right) \]

\[ +c_2' \left\{ \left( \frac{7}{15} - \frac{A_2}{4} \right) \left( P_{ij} - \frac{1}{3} \delta_{ij} P_{kk} \right) \right\} \]

\[ +c_2' \left\{ 0.1 \varepsilon \left[ a_{ij} - \frac{1}{2} \left( a_{ik} a_{kj} - \frac{1}{3} \delta_{ij} A_2 \right) \right] \left( \frac{P_{kk}}{\varepsilon} \right) - 0.05 a_{ij} a_{ik} P_{kl} \right\} \]

\[ +c_2' \left\{ 0.1 \left[ \frac{u_i u_m}{k} P_{mj} + \frac{u_j u_m}{k} P_{mi} \right] - \frac{2}{3} \delta_{ij} \frac{u_l u_m}{k^2} P_{ml} \right\} \]

\[ +c_2' \left\{ 0.1 \left[ \frac{u_k u_j u_l u_i}{k^2} - \frac{1}{3} \delta_{ij} \frac{u_k u_m u_l u_i}{k^2} \right] \left[ 6 D_{lk} + 13 k \left( \frac{\partial U_k}{\partial x_l} + \frac{\partial U_l}{\partial x_k} \right) \right] \right\} \]

\[ +c_2' \left\{ 0.2 \frac{u_k u_j u_l u_i}{k^2} (D_{lk} - P_{lk}) \right\} \]

The coefficients and functions required in the TCL closure were proposed by Craft[55] and are:

<table>
<thead>
<tr>
<th>( c_1 )</th>
<th>( c'_1 )</th>
<th>( c_2 )</th>
<th>( c'_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 f_{Amin} [A_2; 0.5]</td>
<td>1.1</td>
<td>min \left[ \frac{3.24}{1 + S} \right]</td>
<td>min \left[ \frac{0.55; 2S}{1 + 0.6; A} \right] + \frac{2(S - \Omega)}{(3 + S + \Omega)} - 1.5 S_I</td>
</tr>
</tbody>
</table>
The contribution to the pressure strain resulting from buoyancy is modelled through the term $\phi_{ij3}$ which is developed with the same strategy as used for $\phi_{ij2}$:

$$
\phi_{ij3} = -\left(\frac{4}{10} - \frac{3}{80}A_2\right) \left(G_{ij} - \frac{1}{3}\delta_{ij}G_{kk}\right) + \frac{1}{4}u_{ij}G_{kk}
+ \frac{3}{20} \left(\beta_i \frac{u_m u_j}{k} + \beta_j \frac{u_m u_i}{k}\right) u_m \theta
- \frac{1}{10} \delta_{ij} \beta_k \frac{u_m u_k}{k} - u_m \theta
- \frac{1}{8} \left(\frac{u_m u_j}{k} u_i \theta + \frac{u_m u_i}{k} u_j \theta\right) \frac{u_m u_k}{k} \beta_k
+ \frac{1}{8} \left(\frac{u_k u_i u_m u_j}{k^2} + \frac{u_k u_j u_m u_i}{k^2}\right) \beta_k u_m \theta
+ \frac{3}{40} \left(\beta_i \frac{u_m u_j}{k} + \beta_j \frac{u_m u_i}{k}\right) \frac{u_m u_n}{k} \beta_k
+ \frac{1}{4} \beta_k \frac{u_m u_k}{k} - u_m \theta
+ \frac{1}{4} \beta_k \frac{u_m u_k}{k} - u_m \theta
$$

The $\varepsilon_{ij}$ is the other term which needs modelling. In high Reynolds number models, dissipative eddies are considered isotropic and $\varepsilon_{ij}$ is taken equal to
At moderate Reynolds numbers it is proposed that $\varepsilon_{ij} = \varepsilon_{ij}/k$. But this model is not valid immediately adjacent to a wall. Therefore Craft and Launder[56] suggested more complex modelling for $\varepsilon_{ij}$.

$$
\varepsilon_{ij} = \frac{(1-A)}{D} \left( \varepsilon'_{ij} + \varepsilon''_{ij} \right) + A\delta_{ij} \varepsilon
$$

(3.68)

$$
\varepsilon'_{ij} = \frac{\varepsilon}{k} u_i u_j + 2\nu \frac{u_i u_n}{k} \frac{\partial \sqrt{k}}{\partial x_l} \frac{\partial \sqrt{k}}{\partial x_n} \delta_{ij} + 2\nu \frac{u_i u_j}{k} \frac{\partial \sqrt{k}}{\partial x_l} \frac{\partial \sqrt{k}}{\partial x_l} + 2\nu \frac{u_i u_j}{k} \frac{\partial \sqrt{k}}{\partial x_i} \frac{\partial \sqrt{k}}{\partial x_l}
$$

(3.69)

$$
\varepsilon''_{ij} = \varepsilon \left[ 2 \frac{u_i u_k}{k} d_i^a d_k^a \delta_{ij} - \frac{u_i u_j}{k} d_i^a d^a_j - \frac{u_i u_j}{k} d_i^a d_i^a \right] (1-A)
$$

(3.70)

$$
D = \frac{\left( \varepsilon_{kk} + \varepsilon''_{kk} \right)}{2\varepsilon}
$$

(3.71)

$$
d_i^a = N_i \frac{1}{0.5 + (N_k N_k)^{0.5}}
$$

(3.72)

$$
N_i = \frac{\partial}{\partial x_i} \left[ \frac{k^{3/2} A^{1/2}}{\varepsilon} \right]
$$

(3.73)

### 3.3.8 Differential Stress Closure Used with $\overline{\theta^2}$ and $\varepsilon_\theta$ Transport Equations

In this research, both versions of the differential stress closure (Basic and TCL) have been employed, which calculates turbulent heat fluxes based on the equation 3.20. Approximation of turbulent heat flux $\overline{u_i \theta}$ was explained in detail in Section 3.3.4. In order to calculate the equation 3.20, it is required to solve differential transport equations of $\overline{\theta^2}$ and $\varepsilon_\theta$. Another approach is to solve a differential transport equation for $\overline{\theta^2}$ and calculate $\varepsilon_\theta$ using an assumed constant ratio of $\tau_\theta/\tau \approx 0.5$. The equations are the same as those shown in the previous sections apart from $c_\theta$ in the equation for $\overline{u_i \theta}$ which is now taken as 0.3, and the turbulent diffusion terms in the $\overline{\theta^2}$ and $\varepsilon_\theta$ differential transport equations which are formulated in terms of the Generalized Gradient Diffusivity Hypothesis (GGDH) instead of the effective eddy diffusivity hypothesis.
Chapter 4

Wall Functions

In the previous Chapter, the mathematical models of turbulence used in this study have been reviewed. In this Chapter the modifications needed to consider near wall effects will be discussed. The wall function Chapter is divided into two major sections, one on standard wall functions and one on analytical wall functions which have recently been developed at UMIST, now the University of Manchester.

4.1 Conventional Wall Functions

4.1.1 Simple Hydraulic Wall Function

When the Reynolds-number is high, the viscous sub-layer part of the boundary layer is very thin. Therefore, in order to resolve the important changes in the viscous sub-layer, extremely fine grid resolution is needed (see Figure 4.1). As a result, the numerical method is computationally expensive. An alternative method is introduced through the wall functions. In the wall function strategy, the near-wall control volume is large enough for the near-wall node to be located in the fully turbulent region. Knowledge of (and assumptions regarding) the near-wall flow behaviour are then used to provide wall boundary conditions for the momentum and turbulence equations from the nodal values of the near-wall control volumes.
The standard wall function is based on the assumption that a part of velocity profile matches to a logarithm function, and this is called the logarithmic region. The velocity profile and logarithmic region in a typical simple near-wall flow are shown in Figure 4.2. As is shown, the velocity profile is divided into three regions. In the region from the wall up to $y^+$ about 5, the flow is fully laminar and velocity varies linearly with the distance from the wall. From $y^+$ about 5 to 30, there is transition from laminar to turbulent regime. In the region beyond $y^+$ about 30, the flow is fully turbulent. The equation for the velocity profile in this latter logarithmic region is:

$$u^+ = \frac{\tau}{\mu} = \frac{1}{\kappa} \ln y^+ + B$$  \hspace{1cm} (4.1)

In the above equation, $u_\tau$ is the wall shear velocity and it is equal to $\sqrt{\tau_w/\rho}$. $\tau$ is the mean velocity parallel to the wall. $\tau_w$ is the shear stress on the wall. $\kappa$ is a constant which is called the von Karman constant and it is usually taken as $\kappa = 0.41$. $B$ is an empirical constant related to the thickness of the boundary layer. Its value is approximately $B \approx 5.5$ for a boundary layer on a smooth flat plate. In the case of a rough plate the value for $B$ is smaller.
than 5.5. $y^+$ is the dimensionless distance from the wall and it is defined as:

$$y^+ = \frac{\rho u_{\tau} y}{\mu}$$  \hspace{1cm} (4.2)

where $y$ is the dimensional distance to the wall.

In the standard wall function, one of the main assumptions is that the near-wall flow resembles a simple zero-pressure gradient boundary layer in local equilibrium, meaning that the production and dissipation of turbulent energy are approximately equal. From this assumption, it can be shown that:

$$u_{\tau} = C_{\mu}^{1/4} \sqrt{k}$$  \hspace{1cm} (4.3)

From equations 4.3 and 4.1, an equation relating wall shear stress and velocity at the node next to the wall can be derived as:

$$\tau_w = \rho u_{\tau}^2 = \rho C_{\mu}^{1/4} k_{\kappa} \sqrt{k_{\kappa}} \frac{\tau_t}{\ln (y^+ E)}$$  \hspace{1cm} (4.4)

where $E = c^{\kappa B}$.

The overall production of turbulent kinetic energy due to shear in the control volume next to the wall can be approximated as:

$$P_k \approx \tau_w \frac{\partial \tau_t}{\partial y}$$  \hspace{1cm} (4.5)

The velocity gradient can be derived from the logarithmic velocity profile:

$$\left( \frac{\partial \tau_t}{\partial y} \right)_p = \frac{u_{\tau}}{\kappa y_p} = \frac{C_{\mu}^{1/4} \sqrt{k_{\kappa p}}}{\kappa y_p}$$ \hspace{1cm} (4.6)

where the subscript P denotes the value of the variable at the center of the near-wall control volume.

After applying the above equation for the production of turbulent kinetic
energy over the near-wall control volume, $\varepsilon$ at the center of the near-wall control volume is prescribed through:

$$\varepsilon_P = \frac{C_\mu^{3/4} k_P^{3/2}}{\kappa y_P}$$  \hspace{1cm} (4.7)

Therefore the value of $\varepsilon$ is used as boundary condition for $\varepsilon$ equation. The above equation is consistent with the assumption that length scale is:

$$L = \frac{k}{C_\mu^{3/4} y} \approx 2.5 y$$  \hspace{1cm} (4.8)

then we replace this relation to $\varepsilon \approx \frac{k^{3/2}}{L}$.

This set of equations for near wall control volumes are valid only in the logarithmic region. Therefore the non-dimensional distance of near-wall grid point to the wall is $y^+ > 30$. The value of $y^+$ must not exceed few hundreds as well.

### 4.1.2 Standard Wall Function

What will here be referred to as the standard wall function is similar to the wall function introduced in the previous section. The wall shear stress, $\tau_w$, is calculated based on equation 4.4, as before. But the difference lies in the approximation for $P_k$ and $\varepsilon$ in solving the $k$ equation in the near wall cell. Instead of the expressions given earlier, cell-averaged values of $P_k$ and $\varepsilon$ are obtained for use in the $k$ transport equation. In this wall function, it is assumed that the turbulent shear stress and turbulent kinetic energy are constant outside the viscous sublayer. In the viscous sublayer, the turbulent shear stress is equal to zero and kinetic energy increases quadratically with distance from the wall. The cell averaged generation rate of $k$ is then obtained as:

$$\Bar{P}_k = \frac{1}{y_n} \int_0^{y_n} P_k dy$$  \hspace{1cm} (4.9)

where $P_k$ is

$$P_k = \bar{u}\bar{v} \frac{\partial \bar{u}}{\partial y}$$
by assuming that $\overline{uw}$ is equal to wall shear stress ($\tau_w$) and velocity gradient is obtained from the derivative of equation 4.1 as:

$$\frac{\partial \overline{u_t}}{\partial y} = \frac{\tau_w}{\kappa C_\mu^{1/4} \rho k_{p}^{1/2} y}$$

then

$$\overline{P_k} = \frac{1}{y_n} \int_{y_w}^{y_n} \frac{\tau_w^2}{\kappa C_\mu^{1/4} \rho k_{p}^{1/2} y} dy = \frac{\tau_w^2}{\kappa C_\mu^{1/4} \rho k_{p}^{1/2} y_n} \ln \left( \frac{y_n}{y_v} \right)$$

A cell-averaged value for the dissipation rate is also used in the $k$ transport equation for the near-wall cell. To obtain it, the assumption is made that outside the viscous sublayer, epsilon is given from a linearly increasing length-scale:

$$\varepsilon = \frac{k^{3/2}}{c_l y}$$

and in the viscous sublayer, $\varepsilon$ is:

$$\varepsilon = \frac{2\nu k_{p}}{y_v^2}$$

this is obtained by assuming constant distribution of $\varepsilon$ within viscous sublayer and equal to $\varepsilon_w$ which is:

$$\varepsilon_w = 2\nu \left( \frac{\partial k^{1/2}}{\partial x_j} \right)^2$$

The cell-averaged dissipation rate over the near wall control volume is then given by:

$$\bar{\varepsilon} = \frac{1}{y_n} \left[ \int_{y_w}^{y_n} \frac{2\mu k_p}{\rho y_v^2} dy + \int_{y_w}^{y_n} \frac{k_p^{3/2}}{c_l y} dy \right]$$

$$= \frac{1}{y_n} \left[ \frac{2\mu k_p}{\rho y_v} + \frac{k_p^{3/2}}{c_l} \ln \left( \frac{y_n}{y_v} \right) \right]$$

$$= \frac{1}{y_n} \left[ \frac{2k_p^{3/2}}{y_v^2} + \frac{k_p^{3/2}}{c_l} \ln \left( \frac{y_n}{y_v} \right) \right]$$

In a similar way to the hydraulic wall functions, the wall temperature or
wall heat flux can be obtained from a prescribed logarithmic temperature profile. The temperature distribution shape is similar to the velocity distribution shown in Figure 4.2 when the Prandtl number is unity. As the Prandtl number increases, the thickness of the molecular dominated regime in the thermal field increases, whilst the opposite happens when the Prandtl number is less than unity. The logarithmic temperature profile can be written as:

$$\Theta^* = \frac{1}{\chi^*} \ln \left( E' y^+ \right)$$  \hspace{1cm} (4.14)

where

$$\Theta^* = \frac{\rho c_p \left( \Theta_{wall} - \Theta \right) k^1/2}{q_{wall}}$$  \hspace{1cm} (4.15)

In the above equation, $E' y^+$ is a function of molecular Prandtl number and the universal constant $\chi^*$ is $\chi^* = \chi C^{1/4}_\mu \equiv 0.25$. An alternative form of writing this logarithmic temperature profile is:

$$\Theta^* = Pr_t \left( U^+ + \frac{P}{C^{1/4}_\mu} \right)$$  \hspace{1cm} (4.16)

$P$ is called the pee-function and it is derived by Jayatilleke [57] from pipe flow measurements.

$$P = 9.24 \left[ \left( \frac{Pr_t}{Pr_t^1} \right)^{0.75} - 1 \right] \left[ 1 + 0.28 \exp \left( -0.007 \frac{Pr_t}{Pr_t^1} \right) \right]$$  \hspace{1cm} (4.17)

In the case of prescribed wall heat flux, wall temperature can be calculated by:

$$\Theta_{wall} = \frac{q_{wall} \Theta^*}{\rho c_p k^1/2} + \Theta_P$$  \hspace{1cm} (4.18)

In the case of prescribed wall temperature, wall heat flux can be obtained from:

$$q_{wall} = \frac{\rho c_p \left( \Theta_{wall} - \Theta \right) k^1/2}{\Theta^*}$$  \hspace{1cm} (4.19)

The above standard wall function has only been used in combination with the high-Reynolds-number $k-\varepsilon$ model, whilst the more advanced schemes have used the AWF, described in the following section.
4.2 Analytical Wall Functions

In complex flows subjected to body forces, flow acceleration, or adverse pressure gradient, velocity and temperature variations no longer obey the log-law and local equilibrium no longer applies. The analytical wall function is an attempt to produce a more general, but equally economical, wall function by removing some of these limiting assumptions.

4.2.1 Derivation of Analytical Wall Functions

The equations of the analytical wall function are directly derived from simplified forms of the Reynolds averaged momentum and temperature equations. To derive the equations, first the assumptions shown below are introduced:

- The diffusion of momentum parallel to the wall is considerably smaller than the diffusion of momentum in the direction normal to the wall. Therefore, the diffusion parallel to the wall is neglected from the differential momentum transport equation.

- A piecewise linear variation of turbulent viscosity is assumed within the near wall control volumes. The turbulent viscosity is taken as zero within a thin sub-layer \(y < y_v\) and beyond it is given by:

\[
\mu_t = \mu_v c_l (y^* - y_v^*) = \mu_v c_l y_v^*(y^* - y_v^*)
\]  

where \(c_l = 2.55\) and \(c_\mu = 0.09\). The dimensionless wall distance is defined as:

\[
y^* = \frac{\rho_v y k_P^{1/2}}{\mu_v}
\]  

where subscript P denotes the value of the variables at the centre of the near wall control volume, and \(v\) denotes the value of properties at the edge of viscous sublayer. The non-dimensional thickness of this viscous layer, \(y_v^*\), is taken as 10.8.

- The pressure gradient parallel to the wall is assumed to be constant over the near-wall control volume and is obtained from the pressure nodal
values.

- The convection terms normal and parallel to the wall are approximated, either directly from the nodal values, or using averaged values across the near-wall control volumes, as will be shown in more detail later.

With the above assumptions the simplified momentum (parallel to the wall) and temperature differential transport equations can be written as:

\[
\left[ \frac{\partial (\rho U U)}{\partial x} + \frac{\partial (\rho V U)}{\partial y} + \frac{dP}{dx} \right] - g_x (\rho - \rho_{ref}) = \frac{\partial}{\partial y}\left[ (\mu + \mu_t) \frac{\partial U}{\partial y} \right]
\]

(4.22)

\[
\left[ \frac{\partial (\rho U \Theta)}{\partial x} + \frac{\partial (\rho V \Theta)}{\partial y} \right] = \frac{\partial}{\partial y}\left[ \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial \Theta}{\partial y} \right]
\]

(4.23)

With the above approximations for \(\mu_t\), convection and pressure gradient terms, these partial differential equations can be solved analytically, as detailed in the sections below.

In buoyancy driven flows, there may be large gradients of temperature and subsequently variation of fluid properties such as dynamic viscosity across the viscous sublayer. In order to take this important physical effect into account, a non-uniform dynamic viscosity was introduced into the equations for the viscous sublayer region. For such cases a hyperbolic variation of molecular dynamic viscosity is assumed within the zero \(\mu_t\) sub-layer (ie. for \(0 < y^* < y^*_{\mu}\)).

\[
\mu = \frac{\mu_{\mu}}{1 + b_{\mu} (y^* - y^*_{\mu})}
\]

(4.24)

where

\[
b_{\mu} = \frac{\mu_{wall} - \mu_{\mu}}{\mu_{wall} y^*_{\mu}}
\]

(4.25)

The molecular viscosity at the wall temperature (\(\mu_{wall}\)) and molecular viscosity at the edge of viscous sublayer (\(\mu_{\mu}\)) feed into the above equations to define the variation of molecular viscosity across the viscous sublayer. As noted earlier in section 3.2 density variations are expected to be relatively small in the cases considered here. Consequently, although the above variation of \(\mu\) was employed in the wall function expressions, fluid property variations with
temperature were neglected in the main flow transport equations (since temperature difference in the outer flow field will be even smaller than across the near-wall viscous layer).

4.2.2 Thermal Wall Function When the Near Wall Cell Is Thicker Than the Viscous Sublayer

First, we convert the simplified temperature equation into $y^*$ coordinates:

$$\frac{\partial}{\partial y^*} \left[ \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial \Theta}{\partial y^*} \right] = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{\partial (\rho U \Theta)}{\partial x} + \frac{\partial (\rho V \Theta)}{\partial y} \right]_P = C_{th} \quad (4.26)$$

In the viscous sub-layer region ($y^* < y_{v}^*$), where $\mu_t = 0$, this gives

$$\frac{\partial}{\partial y^*} \left[ \frac{\mu}{Pr} \frac{\partial \Theta}{\partial y^*} \right] = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{\partial (\rho U \Theta)}{\partial x} + \frac{\partial (\rho V \Theta)}{\partial y} \right]_P = C_{th1} \quad (4.27)$$

After a first integration:

$$\frac{\mu}{Pr} \frac{\partial \Theta_1}{\partial y^*} = C_{th1} y^* + A_{th1} \quad (4.28)$$

for some constant of integration $A_{th1}$. Subscript 1 denotes parameters inside the viscous sublayer region.

After substituting for $\mu$ from equations 4.24 and 4.25, in the viscous sub-layer, the gradient of temperature is:

$$\frac{\partial \Theta_1}{\partial y^*} = \frac{Pr_v C_{th1} \mu_v}{\mu_v} y^* \left[ 1 + b \mu (y^* - y_{v}^*) \right] + \frac{Pr_v A_{th1}}{\mu_v} \left[ 1 + b \mu (y^* - y_{v}^*) \right] \quad (4.29)$$

Then after a second integration, imposing the boundary condition that at $y^* = 0, \Theta_1 = \Theta_{wall}$, the temperature profile within the viscous sublayer is:
\[ \Theta_1 = \frac{Pr_v}{\mu_v} \left[ \frac{C_{th1} y^*}{2} + A_{th1} y^* \right] + \frac{Pr_v b_\mu}{\mu_v} \left[ \frac{C_{th1} y^*}{3} - \frac{C_{th1} y^*}{2} y^* - A_{th1} y^* y^* \right] + \Theta_{wall} \quad (4.30) \]

In the fully turbulent region \((y^* > y^*_\nu)\), the simplified temperature equation is:

\[
\frac{\partial}{\partial y^*} \left[ \left( \frac{\mu_v}{Pr_v} + \frac{\mu_t}{Pr_t} \right) \frac{\partial \Theta_2}{\partial y^*} \right] = \frac{\mu_v^2}{\rho_v k_p} \left[ \frac{\partial (\rho U \Theta)}{\partial x} + \frac{\partial (\rho V \Theta)}{\partial y} \right] = C_{th2} \quad (4.31)
\]

where subscript 2 is used to denote parameters inside the fully turbulent region. After a first integration:

\[
\mu_v \left[ 1 + \frac{Pr_v \alpha}{Pr_t} (y^* - y^*_\nu) \right] \frac{\partial \Theta_2}{\partial y^*} = Pr_v C_{th2} y^* + Pr_v A_{th2} \quad (4.32)
\]

where \(A_{th2}\) is a constant of integration.

In the fully turbulent region of the near wall control volume the gradient of temperature is thus:

\[
\frac{\partial \Theta_2}{\partial y^*} = \frac{Pr_v C_{th2}}{\mu_v \alpha_t} + \frac{Pr_v A_{th2} + C_{th2} \left( y^*_\nu - \frac{1}{\alpha_t} \right)}{\mu_v \alpha_t \left[ 1 + \alpha_t (y^* - y^*_\nu) \right]} \quad (4.33)
\]

where

\[
\alpha_t = \frac{Pr_v \alpha}{Pr_t} \quad (4.34)
\]

Let

\[
Y_T = \left[ 1 + \alpha_t (y^* - y^*_\nu) \right] \quad (4.35)
\]

\[
Y_T n = \left[ 1 + \alpha_t (y^*_n - y^*_\nu) \right] \quad (4.36)
\]

where subscript \(n\) denotes values at the top face of the near wall control volume which can be calculated by interpolation of the nodal values (Figure 4.3).
Then after a second integration:

\[
\Theta_2 = \frac{Pr_v}{\mu_v \alpha_t} C_{th2} y^* + \frac{Pr_v}{\mu_v \alpha_t} \left[ A_{th1} + C_{th2} \left( y^* - \frac{1}{\alpha_t} \right) \right] \ln Y_T + D_{th} \quad (4.37)
\]

where \( D_{th} \) is a constant of integration.

The unknown constants \( A_{th1}, A_{th2} \) and \( D_{th} \) can be obtained from the boundary conditions:

- at \( y^* = y_v^* \): \( \left( \frac{\partial \Theta_1}{\partial y^*} \right)_{y_v^*} = \left( \frac{\partial \Theta_2}{\partial y^*} \right)_{y_v^*} \)
- at \( y^* = y_v^* \): \( (\Theta_1)_{y_v^*} = (\Theta_2)_{y_v^*} \)
- at \( y^* = y_n^* \): \( (\Theta_2)_{y_n^*} = \Theta_n \)

The temperature profile in the fully turbulent region is then finally given by:

\[
\Theta_2 = \frac{Pr_v}{\mu_v \alpha_t} C_{th2} (y^* - y_v^*) + M \ln Y_T + R - b_\mu R_\mu + \Theta_{wall} \quad (4.38)
\]

where

\[
M = \frac{Pr_v}{\mu_v \alpha_t} \left[ A_{th1} + C_{th1} y_v^* - \frac{C_{th2}}{\alpha_t} \right] \quad (4.39)
\]

\[
R = \frac{Pr_v y_v^*}{\mu_v} \left[ \frac{C_{th1}}{2} y_v^* + A_{th1} \right] \quad (4.40)
\]

\[
R_\mu = \frac{Pr_v y_v^{*2}}{2\mu_v} \left[ \frac{C_{th1}}{3} y_v^* + A_{th1} \right] \quad (4.41)
\]

In the case of prescribed wall heat flux boundary condition, the wall temperature can be obtained by substituting \( y_n^* \) in equation 4.38 where \( \Theta_n \) is calculated from the interpolation of nodal temperature values:
\[ \Theta_{wall} = \Theta_n - \frac{Pr_v}{\mu_v} \left[ \frac{C_{th2}}{\alpha_t} (y_n^* - y_v^*) + \frac{\ln Y_T n}{\alpha_t} \left( A_{th1} + C_{th1} y_v^* - \frac{C_{th2}}{\alpha_t} \right) \right] \\
+ \frac{Pr_v}{\mu_v} \left[ y_v^* \left( A_{th1} + \frac{C_{th1} y_v^*}{2} \right) \right] + b_{\mu} \frac{Pr_v y_v^{*2}}{2 \mu_v} \left[ \frac{C_{th1} y_v^*}{3} + A_{th1} \right] \] (4.42)

where

\[ A_{th1} = -\frac{q_{wall}}{c_p} \frac{\mu_v}{\rho_v \sqrt{k_P}} \] (4.43)

In the case of prescribed wall temperature, the wall heat flux can be obtained by finding the derivative of equation 4.38 and substituting \( y^* = y_n^* \):

\[ A_{th1} = \frac{(\Theta_n - \Theta_{wall}) \mu_v}{Pr_v} \frac{1 \alpha_t C_{th2} (y_n^* - y_v^*)}{\ln Y_T n + y_v^* - \frac{1}{2} b_{\mu} y_v^{*2}} - \frac{C_{th1} y_v^{*2}}{2} + b_{\mu} \frac{C_{th1} y_v^{*3}}{6} \]
\[ + \frac{1 \alpha_t}{\ln Y_T n + y_v^* - \frac{1}{2} b_{\mu} y_v^{*2}} \] (4.44)

### 4.2.3 Thermal Wall Function When the Near Wall Cell Is Thinner Than the Viscous Sublayer

When the viscous sublayer is thicker than the near wall control volume, the above integration is carried out only over the viscous layer part \((\mu_t = 0)\) of the near wall control volume based on the following boundary conditions:

- at \( y^* = 0 \): \((\Theta_1)_{y^*=0} = \Theta_{wall} \)
- at \( y^* = y_n^* \): \((\Theta_1)_{y_n^*} = \Theta_n \)

Then the temperature profile across near wall control volume is given by:

\[ \Theta = \frac{Pr_n}{\mu_n} \left[ \frac{C_{th} y_v^{*2}}{2} + A_{th} y_v^* \right] \]
\[ + \frac{Pr_n b_{\mu}}{\mu_n} \left[ \frac{C_{th} y_v^{*3}}{3} - \frac{C_{th1} y_v^* - A_{th1} y_v^{*2}}{2} - A_{th} y_n^* y_v^* \right] + \Theta_{wall} \] (4.45)
In the case of prescribed wall heat flux, the wall temperature can be calculated by:

$$\Theta_{wall} = \Theta_n - \frac{Pr_n y_n^*}{\mu_n} \left( \frac{C_{th} y_n^*}{2} + A_{th} \right) + b_{\mu} \frac{Pr_n y_n^*}{2\mu} \left[ \frac{C_{th} y_n^*}{3} + A_{th} \right]$$  \hspace{1cm} (4.46)

where

$$A_{th} = -\frac{q_{wall}}{c_p \rho_n \sqrt{k_p}}$$  \hspace{1cm} (4.47)

In the case of prescribed wall temperature, the wall heat flux can be calculated as:

$$q_{wall} = -\frac{\rho_n c_p \sqrt{k_p}}{\mu_n} A_{th}$$  \hspace{1cm} (4.48)

where

$$A_{th} = \frac{(\Theta_n - \Theta_{wall}) \frac{Pr_n}{\mu_n} \frac{\mu_n}{c_p} y_n^* + b_{\mu} C_{th} y_n^3}{y_n^* - \frac{1}{2} b_{\mu} y_n^2} \frac{C_{th} y_n^*}{6}$$  \hspace{1cm} (4.49)

### 4.2.4 Hydrodynamic Wall Function When the Near Wall Cell Is Thicker Than the Viscous Sublayer

We now integrate the simplified wall-parallel momentum equation over the near-wall cell in a similar manner to that just outlined for the temperature equation. First we convert the simplified differential transport equation for momentum into $y^*$ coordinates:

$$\frac{\partial}{\partial y^*} \left[ (\mu + \mu_t) \frac{\partial U}{\partial y^*} \right] = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{\partial (\rho U U)}{\partial x} + \frac{\partial (\rho V U)}{\partial y} + \frac{dP}{dx} \right]_p + \frac{\mu_v^2}{\rho_v^2 k_P} [-\rho_{ref} g \beta \left( \Theta - \Theta_{ref} \right)]$$  \hspace{1cm} (4.50)

or

$$\frac{\partial}{\partial y^*} \left[ (\mu + \mu_t) \frac{\partial U}{\partial y^*} \right] = C + b (\Theta - \Theta_{ref})$$  \hspace{1cm} (4.51)

where

$$b = -\frac{\mu_v^2}{\rho_v^2 k_P \rho_{ref} g \beta}$$  \hspace{1cm} (4.52)
\[ C = \frac{\mu^2}{\rho^2 k_p} \left[ \frac{\partial (\rho U U)}{\partial x} + \frac{\partial (\rho V U)}{\partial y} + \frac{dP}{dx} \right] \]  

(4.53)

\( C \) is the convective and pressure gradient terms in the momentum equation. The convective and pressure gradient terms in the viscous sublayer is called \( C_1 \) and the convective and pressure gradient terms in the fully turbulent region is called \( C_2 \).

In the viscous sub-layer region \( y^* < y_t^* \) we thus have:

\[ \frac{\partial}{\partial y^*} \left[ \mu \frac{\partial U_1}{\partial y^*} \right] = C_1 + b (\Theta_1 - \Theta_{ref}) \]  

(4.54)
After replacing $\Theta_1$ from the thermal analytical wall function, expression of equation 4.30, and substituting for $\mu$ from equations 4.24 and 4.25, then integrating the above equation in the viscous sublayer region, the gradient of the wall-parallel component of velocity is:

$$
\frac{\mu_v}{\mu} \frac{\partial U_1}{\partial y^*} = C_1 y^* + A_1 + b (\Theta_{wall} - \Theta_{ref}) y^* + b \frac{Pr_v}{\mu} \left[ \frac{C_{th1} y^{*3}}{6} + \frac{A_{th1} y^{*2}}{2} \right] + b \frac{Pr_v}{\mu} \left[ \frac{C_{th1} y^{*4}}{6} + \frac{A_{th1} y^{*3}}{2} - \frac{C_{th1} y^{*2}}{2} \right] + b \frac{Pr_v}{\mu} \left[ \frac{C_{th1} y^{*5}}{12} - \frac{C_{th1} y^{*4}}{2} \right] - b \frac{Pr_v}{\mu} \left[ \frac{C_{th1} y^{*6}}{6} - \frac{(C_{th1} y^{*5} - A_{th1}) y^{*3}}{2} \right] + A_1 b \mu (y^* - y_v^*)
$$

(4.55)

where $A_1$ is a constant of integration.

After a second integration and applying the boundary condition that at $y^* = 0 : U_1 = 0$, the wall parallel velocity profile within the zero $\mu_t$ sublayer is:
\[\mu_v U_1 = \frac{C_1}{2} y^* + A_1 y^* + \frac{b y^*}{2} \left[ (\Theta_{wall} - \Theta_{ref}) + \frac{Pr_v y^*}{3 \mu_v} \left( \frac{C_{th1} y^*}{4} + A_{th1} \right) \right] + b\mu y^* \left( \frac{y^*}{3} - \frac{y_v^*}{2} \right) \left[ C_1 + b (\Theta_{wall} - \Theta_{ref}) \right] + \frac{bb_2 Pr_v y^*}{\mu_v} \left[ \frac{C_{th1} y^*}{20} - \frac{(C_{th1} y_v^* - 2A_{th1}) y^*}{10} - \frac{A_{th1} y_v^*}{3} \right] + \frac{bb_2 Pr_v y^*}{6 \mu_v} \left[ \frac{C_{th1} y^*}{12} - \frac{(3C_{th1} y_v^* - 2A_{th1}) y^*}{10} \right] + \frac{bb_2 Pr_v y^*}{6 \mu_v} \left[ \frac{(C_{th1} y_v^* - 4A_{th1}) y^*}{4} y^* + A_{th1} y_v^* \right] + b\mu A_1 y^* \left( \frac{y^*}{2} - y_v^* \right) \] (4.56)

In the fully turbulent region \( y^* > y_v^* \) the momentum equation is written as:

\[ \frac{\partial}{\partial y^*} \left[ (\mu + \mu_t) \frac{\partial U_2}{\partial y^*} \right] = C_2 + b (\Theta_2 - \Theta_{ref}) \] (4.57)

First the equation for \( \Theta_2 \) from the thermal analytical wall function (equation 4.38) is substituted into the above equation. Then we assume that the turbulent viscosity varies linearly across the fully turbulent region, according to equation 4.20. After a first integration, the wall-parallel velocity gradient across the fully turbulent region of the near wall control volume is:

\[ \mu_v \frac{\partial U_2}{\partial y^*} = \frac{C_2}{[1 + \alpha (y^* - y_v^*)]} y^* + \frac{A_2}{[1 + \alpha (y^* - y_v^*)]} + b \frac{(R + \Theta_{wall} - \Theta_{ref}) y^*}{[1 + \alpha (y^* - y_v^*)]} + b \frac{Pr_v C_{th2}}{2 \mu_v \alpha_t} \left[ 1 + \alpha (y^* - y_v^*) \right] - b \frac{Pr_v C_{th2} y_v^*}{\mu_v \alpha_t} \left[ 1 + \alpha (y^* - y_v^*) \right] + b \frac{M}{\alpha_t} Y_T \left[ 1 + \alpha (y^* - y_v^*) \right] (\ln Y_T - 1) - \frac{bb_2 R_v y^*}{\alpha_t} \left[ 1 + \alpha (y^* - y_v^*) \right] \] (4.58)

The parameters, \( M, R \) and \( R_v \), are calculated by imposing the appropriate boundary conditions.

Let

\[ Y = [1 + \alpha (y^* - y_v^*)] \] (4.59)
\[ Y_n = [1 + \alpha (y_n^* - y_n)] \]  

(4.60)

After a second integration we need to apply the following boundary conditions to obtain the parameters, \( M, R \) and \( R_\mu \).

- at \( y^* = y_v^* \): \( \mu_v (U_1) y_v^* = \mu_v (U_2)y_v^* \)
- at \( y^* = y_v^* \): \( \mu_v \left( \frac{\partial U_1}{\partial y^*} \right) y_v^* = \mu_v \left( \frac{\partial U_2}{\partial y^*} \right) y_v^* \)
- at \( y^* = y_n^* \): \( (U_2) y_n^* = U_n \)

where
\[
M = \frac{Pr_v}{\mu_v \alpha_t} \left[ A_{th1} + C_{th1} y_v^* - \frac{C_{th2}}{\alpha_t} \right] \tag{4.61}
\]

\[
R = \frac{Pr_v y_v^*}{\mu_v} \left[ \frac{C_{th1}}{2} y_v^* + A_{th1} \right] \tag{4.62}
\]

\[
R_\mu = \frac{Pr_v y_v^*}{2 \mu_v} \left[ \frac{C_{th1}}{3} y_v^* + A_{th1} \right] \tag{4.63}
\]

After replacing the constants resulting from the above boundary conditions, the wall parallel velocity profile in the fully turbulent region of the near wall control volume is given as:

\[
\mu_v U_2 = \frac{C_2}{\alpha} \left[ y^* - \left( \frac{1}{\alpha} - y_v^* \right) \ln Y \right] + \frac{A_2}{\alpha} \ln Y \\
+ \frac{b}{\alpha} \left( R + \Theta_{wall} - \Theta_{ref} \right) \left[ y^* - \left( \frac{1}{\alpha} - y_v^* \right) \ln Y \right] \\
+ \frac{bPr_v C_{th2}}{2\mu_v \alpha_t} \left[ \frac{y_v^{*2}}{2} - y^* \left( \frac{1}{\alpha} - y_v^* \right) + \left( \frac{1}{\alpha} - y_v^* \right)^2 \ln Y \right] \\
- \frac{bPr_v C_{th2}}{\mu_v \alpha_t} \left[ y_v^* - \left( \frac{1}{\alpha} - y_v^* \right) \ln Y \right] \\
+ \frac{bM}{\alpha} \left[ \frac{Y_T}{\alpha_t} (\ln Y_T - 1) - y^* \right] + \frac{bM}{\alpha^2} \left( 1 - \frac{\alpha}{\alpha_t} \right) \ln Y \\
- \frac{bM}{\alpha} \left( 1 - \frac{\alpha}{\alpha_t} \right) \int \frac{\ln Y_T}{Y} dy^* \\
- \frac{bb\mu R_\mu}{\alpha} \left[ y^* - \left( \frac{1}{\alpha} - y_v^* \right) \ln Y \right] + B_2 \tag{4.64}
\]
where

\[
B_2 = y_v^* \left( \frac{C_1}{2} y_v^* - \frac{C_2}{\alpha} \right) + \frac{by_v^2}{2} \left[ (\Theta_{\text{wall}} - \Theta_{\text{ref}}) + \frac{Pr_v y_v^*}{3\mu_v} \left( \frac{C_{th1} y_v^*}{4} + A_{th1} \right) \right] \\
- b \left[ \frac{(R + \Theta_{\text{wall}} - \Theta_{\text{ref}})}{\alpha} y_v^* - \frac{Pr_v C_{th2} y_v^*}{2\mu_v \alpha_t} \left( \frac{y_v^* + 1}{\alpha} \right) - M \left( \frac{1}{\alpha_t} + y_v^* \right) \right] \\
- b \frac{y_v^3}{6} \{ [C_1 + b (\Theta_{\text{wall}} - \Theta_{\text{ref}})] \} \\
+ b \frac{y_v^3}{6} \left\{ b \frac{Pr_v y_v^*}{\mu_v} \left[ \left( \frac{C_{th1}}{5} y_v^* + A_{th1} \right) - \frac{b \mu y_v^*}{5} \left( \frac{C_{th1}}{6} y_v^* + A_{th1} \right) \right] \right\} \\
+ \frac{bb \mu R_\mu y \nu^*}{\alpha_t} + A_1 y_v^* \left( 1 - \frac{b \mu y_v^*}{2} \right) \\
\text{ (4.65)}
\]

\[
A_2 = (C_1 - C_2) y_v^* \\
+ b \left[ \frac{Pr_v y_v^*}{2\mu_v} \left( \frac{C_{th1} y_v^*}{3} + A_{th1} \right) + \frac{Pr_v C_{th2} y_v^*}{2\mu_v \alpha_t} + M - R y_v^* \right] \\
- bb \mu \left[ \frac{Pr_v y_v^3}{3\mu_v} \left( \frac{C_{th1} y_v^*}{4} + A_{th1} \right) - R y_v^* \right] + A_1 \\
\text{ (4.66)}
\]

\[
A_1 = \frac{\mu U_n - N}{\left[ \ln \frac{y_v^*}{\alpha} + y_v^* - b \mu \frac{y_v^2}{2} \right]} \\
\text{ (4.67)}
\]

\[
N = \frac{C_2}{\alpha} \left[ y_n^* - \left( \frac{1}{\alpha} - y_v^* \right) \ln Y_n \right] + \left( \frac{C_1 - C_2}{\alpha} y_v^* \ln Y_n + \left( \frac{C_1}{2} y_v^* - \frac{C_2}{\alpha} \right) y_v^* \right] \\
+ b \left[ \frac{(R + \Theta_{\text{wall}} - \Theta_{\text{ref}})}{\alpha} f_{b1} + \frac{Pr_v C_{th2}}{\mu_v \alpha t} \left( \frac{f_{b2}}{2} - y_v^* f_{b3} \right) + M f_{b4} \right] \\
+ \frac{y_v^2}{2} \left[ \Theta_{\text{wall}} - \Theta_{\text{ref}} + \frac{Pr_v y_v^*}{3\mu_v} \left( \frac{C_{th1} y_v^*}{4} + A_{th1} \right) \right] \\
+ \frac{y_v^2}{2} \left[ \frac{Pr_v}{\mu_v \alpha} \left( \frac{C_{th1} y_v^*}{3} + A_{th1} \right) \ln Y_n - \frac{2 R}{y_v^*} \ln Y_n \right] \\
- \frac{b \mu y_v^3}{6} \{ C_1 + b (\Theta_{\text{wall}} - \Theta_{\text{ref}}) \} \\
- \frac{bb \mu}{6 \mu_v} \left[ \left( \frac{C_{th1}}{5} y_v^* + A_{th1} \right) - \frac{b \mu y_v^*}{5} \left( \frac{C_{th1}}{6} y_v^* + A_{th1} \right) \right] \\
- \frac{bb \mu}{3 \mu_v \alpha} \left( \frac{C_{th1} y_v^*}{4} + A_{th1} \right) \ln Y_n - bb \mu R_\mu f_{b4} \\
\text{ (4.68)}
\]
The integration of $Q_n = \int_{y^*_n}^{y^*_v} \ln \frac{y^*_n}{y^*_v} dy^*$, appearing in $f_{b4}$, can not analytically be determined. Therefore, a series of numerical integrations are calculated in the range of $0.7 \leq Pr \leq 10$ and the following expression is obtained to approximate $Q_n$ as:

$$Q_n = \frac{2 \left( y^*_n - y^*_v \right) Pr^{0.26}}{1 + \sqrt{y^*_n - y^*_v}} \quad (4.69)$$

Then the wall shear stress can be calculated as:

$$\tau_{wall} = -\frac{\rho_v \sqrt{K_P}}{\mu_v} A_1 \quad (4.70)$$

The average production of turbulence kinetic energy within the near wall control volumes is now calculated as:

$$\overline{P_k} = \frac{1}{y_n} \frac{\rho_v \sqrt{K_P}}{\mu_v} \int_{y_n^*}^{y_v^*} \mu_v \alpha \left( y^*_n - y^*_v \right) \left( \frac{\partial U_2}{\partial y^*_v} \right)^2 dy^* \quad (4.71)$$
where

\[
\frac{\partial U_2}{\partial y^*} = \frac{C_2 y^* + A_2}{\mu_v Y} + \frac{b}{\mu_v Y} \left[ (R + \Theta_{wall} - \Theta_{ref}) y^* + \frac{Pr_v}{\mu_v \alpha_t} C_{th1} y^* \left( \frac{y^*}{2} - y_v^* \right) \right] \\
+ \frac{b}{\mu_v Y} \left[ \frac{M}{\alpha_t} Y_T (\ln Y_T - 1) \right] \\
- \frac{b b \mu R \mu_y^*}{\mu_v Y}
\]  

(4.72)

The value of \( P_k \) is calculated by numerical integration within the near-wall cell in this study. The buoyancy generation term in the \( k \) transport equation is:

\[
G_k = -\rho \beta g_i u_i \theta
\]

where, using an eddy diffusivity model for the turbulent heat fluxes:

\[
\overline{u_i \theta} = -\frac{\nu_t}{Pr_t} \frac{\partial \Theta}{\partial x_i}
\]  

(4.73)

Two ways have been tested in this research to calculate the buoyancy generation term. One way is to calculate the temperature gradient in equation 4.73 from the nodal values and the other way is to calculate a cell-averaged generation term from the AWF equations as:

\[
\overline{G_k} = \frac{\rho_v \sqrt{k_p}}{\mu_v} \frac{1}{\Delta y} \frac{g_i \beta \mu_v \alpha}{Pr_t} \int_{y_v^*}^{y_n^*} (y^* - y_v^*) \frac{\partial \Theta_2}{\partial y^*} dy^*
\]

Comparisons show that both methods produce the same temperature distribution and Nusselt number predictions, although the cell-average temperature gradient is used in this study.

### 4.2.5 Hydrodynamic Wall Function When Near Wall Cell Is Thinner Than Viscous Sublayer

In the situation that the viscous sublayer is thicker than the near wall control volume, the following equations are simplified compared to the previous
section by setting a zero value to $\mu_t$.

$$A_1 = \frac{\mu_n U_n - N}{y_n^* - b \mu \frac{y_n^*}{2}}$$

(4.74)

$$N = \frac{C y_n^*}{2} + b y_n^* \left[ (\Theta_{wall} - \Theta_{ref}) + \frac{P r_n y_n^*}{\mu_n} \left( \frac{C_{th} y_n^*}{4} + A_{th} \right) \right]$$

(4.75)

and the wall shear stress equation is:

$$\tau_{wall} = -\frac{\rho_n \sqrt{k_p}}{\mu_n} A_1$$

(4.76)

Then the average production of turbulence kinetic energy is:

$$\overline{P_k} = 0$$

(4.77)

### 4.2.6 Convection and pressure gradient terms

The evaluation of the convective and pressure gradient terms is a very important feature of the AWF, especially in the case of the thermal convective terms. These terms in the momentum and energy equations respectively are:

$$C = \frac{\mu_v^2}{\rho_v^2 k_p} \left[ \frac{\partial (\rho U U)}{\partial x} + \frac{\partial (\rho V U)}{\partial y} + \frac{dP}{dx} \right]$$

$$C_{th} = \frac{\mu_v^2}{\rho_v^2 k_p} \left[ \frac{\partial (\rho U \Theta)}{\partial x} + \frac{\partial (\rho V \Theta)}{\partial y} \right]$$

When the thickness of the zero $\mu_t$ sublayer is less than that of the near wall control volume, there are two regions in the near wall control volume. One is the zero $\mu_t$ sublayer and the other is the region beyond it. Therefore there are two strategies that could be adopted to calculate the convective terms. One option is to calculate a single convection term for the entire control volume
and the other option is to calculate $C_1$ and $C_{th1}$ within the zero $\mu_t$ sublayer and separately approximate $C_2$ and $C_{th2}$ within the turbulent region.

In order to discretize the convective terms, the central differencing was employed, as illustrated below, and its performance was generally good. The simplest way for calculation of convective and pressure gradient terms is to treat the near-wall control volume as a single layer as shown below:

**One layer case**

\[
C = C_1 = C_2 = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{P_e - P_w}{\Delta x_{ew}} + \rho U_p \frac{U_e - U_w}{\Delta x_{ew}} + \rho V_p \frac{U_n - U_s}{\Delta y_{ns}} \right] \tag{4.78}
\]

\[
C_{th} = C_{th1} = C_{th2} = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \rho U_p \frac{\Theta_e - \Theta_w}{\Delta x_{ew}} + \rho V_p \frac{\Theta_n - \Theta_s}{\Delta y_{ns}} \right] \tag{4.79}
\]

The subscript P denotes the value at the center of the control volume, $v$ denotes the value at the edge of viscous sublayer, and e,w,n,s respectively denote values at the east, west, north and south faces.

After studying a buoyancy-driven flow inside a square cavity that heated and cooled from vertical walls, it was found that there are some differences between the calculated wall heat transfer and experimental data. Even though the calculated data was closer to experimental data compared with SWF results, the alternative methods were investigated to obtain more accurate calculated wall heat transfer. After testing various methods for calculation of convective terms, and comparing the numerical data with the experimental measurements, the following way of calculating convective terms has been chosen to give the the best hydrodynamic and thermal results.

**Two layer case**

With this method, the component of convection parallel to the wall is treated as in the one layer approach generally everywhere. But the component of convection normal to the wall is calculated differently in the one layer and two layer schemes. It has been found that to get the best results, the wall-normal
convection should be treated as in the one layer approach when the flow is di-
rected towards the wall and it should be treated as in the two layer approach
below when the flow is moving away from the wall. When the zero $\mu_t$ sub-
layer is thicker than the near wall control volume, the convective terms are
generally calculated from the one layer approach.

In the two layer approach the turbulent hydrodynamic convective terms are:

$$C_1 = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{P_e - P_w}{\Delta x_{ew}} + \rho U_p \frac{U_e - U_w}{\Delta x_{ew}} + \rho \left( \frac{V_v}{2} \right) \frac{U_v - U_s}{\Delta y_v} \right]$$

$$C_2 = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \frac{P_e - P_w}{\Delta x_{ew}} + \rho U_p \frac{U_e - U_w}{\Delta x_{ew}} + \rho \left( \frac{V_v + V_n}{2} \right) \frac{U_n - U_v}{\Delta y_{ns} - \Delta y_v} \right]$$

The thermal convective terms are:

$$C_{th1} = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \rho U_p \frac{\Theta_e - \Theta_w}{\Delta x_{ew}} + \rho \left( \frac{V_v}{2} \right) \frac{\Theta_v - \Theta_s}{\Delta y_v} \right]$$

$$C_{th2} = \frac{\mu_v^2}{\rho_v^2 k_P} \left[ \rho U_p \frac{\Theta_e - \Theta_w}{\Delta x_{ew}} + \rho \left( \frac{V_v + V_n}{2} \right) \frac{\Theta_n - \Theta_v}{\Delta y_{ns} - \Delta y_v} \right]$$

The subscript $v$ denotes the value at the edge of viscous sublayer. $\Delta y_v$ is
the thickness of the viscous sublayer. The vertical component of velocity at the
dge of viscous sublayer is calculated by using linear interpolation between
the vertical velocity at the wall, which is zero in these cases, and the near-wall
nodal value of vertical velocity.

Finally it is worth mentioning that a number of other methods were tested
to calculate the convective terms, including calculation of averaged values of
the parallel component of velocity within the viscous sublayer obtained from
analytical integration of AWF equations [39]. These showed no difference in
results in the square cavity test case heated and cooled from vertical walls,
and so the simpler approach outlined above has been adopted in the cases
presented later.

In this chapter different wall functions approaches have been presented
and the way that these equations are calculated and implemented have been
explained. It might be noted that the forms of both the standard and analytical wall function are implicitly derived for steady-state flows (since no time-dependent terms are explicitly included). Nevertheless, as is routinely done in computations with wall functions, they are both applied without modification to both steady-state and time-dependent simulations in the present work.
Chapter 5

Numerical Implementation

In this chapter all the numerical methods implemented in the research are explained. Firstly, two in-house FORTRAN codes which were used for the simulations are introduced in sections 5.1 and 5.2. The discretization methods implemented to approximate the convection and diffusion terms are explained in section 5.3. The way that the mass conservation is enforced to the calculations is shown in section 5.4. In section 5.5, the calculation loop is discussed alongside with a diagram. The terms added to enable time dependent calculations to be performed are presented in section 5.6. The ways that boundary conditions are treated in the numerical simulations are discussed in section 5.7. Finally, sections 5.8-5.9 describe the way in which the inclined cavity geometries were defined.

5.1 TEAM Code

The TEAM code has been employed in this research to carry out the 2D simulations. TEAM stands for “Turbulent Elliptic Algorithm of Manchester” [58]. This code was developed to simulate two-dimensional, elliptic and incompressible flows by the finite volume method. It can analyze the flow under both steady state and transient conditions. The TEAM code is capable of employing either the SIMPLE [59] or PISO [60] algorithms for velocity-pressure linkage. Regarding the discretization of convective terms, QUICK [61] and PLDS [62] schemes are included in the TEAM code. Furthermore, the TEAM code is designed so that it can employ different near wall treatments. The
near-wall treatments available in the present version of the TEAM code are low-Re-number turbulence models, the standard wall function and the analytical wall function, as described in Chapter 4.

![Figure 5.1](image)

**Figure 5.1** – Grid and control volume arrangements for pressure and mean velocity storage in the TEAM code.

The grid arrangement adopted in the TEAM code is a fully staggered pattern, as shown in Figure 5.1. This arrangement is used in order to avoid chequerboard effects in the velocity and pressure fields.

The method for grid generation in the TEAM code involves first generating the scalar control volume boundaries and then placing the scalar nodes at the centre of each control volume. The velocity nodes are located at the mid-point of the faces of the scalar control volumes. This method of grid generation is convenient, especially when wall boundaries of the geometry are located on faces of the scalar control volumes.

### 5.2 STREAM Code

The STREAM code has been employed in this research to simulate the three dimensional 15° stable and unstable tilted tall cavities and annular horizontal penetration. STREAM stands for “Simulation of Turbulent Reynolds-averaged Equations for All Mach numbers”. This code [63] was developed to simulate two or three dimensional flows. It is a fully elliptic code and uses block structured non-orthogonal body fitted grids and the finite volume method. It can analyze the flow under both steady state and transient conditions. The
STREAM code employs the SIMPLE algorithm for velocity-pressure linkage. The grid arrangement adopted in the STREAM code is a collocated grid [64]. This means that both scalar and velocity variables are stored at the centre of control volumes. Regarding the discretization of convective terms, QUICK, UPWIND [65] and UMIST [66] schemes are included in the STREAM code. The STREAM code is also designed so that it can employ different near wall treatments, and again the available choices in the present version are low-Re-number models, the standard wall function and the analytical wall function. A wide range of RANS turbulence models is available in the STREAM code.

The method for grid generation in the STREAM code involves first reading the scalar control volume boundaries from external files and then placing the nodes at the centres of the control volumes.

5.3 Approximation of Convective and Diffusive terms

To study fluid flow problems, the effects of convection and diffusion must be accounted for. The general form of a steady convection-diffusion transport equation for a quantity $\phi$ is:

$$
\text{div} (\rho u \phi) = \text{div} (\Gamma \text{grad} \phi) + S_{\phi}
$$

(5.1)

In the case of steady one-dimensional convection and diffusion in the absence of sources, the above equation reduces to:

$$
\frac{d}{dx} (\rho u \phi) = \frac{d}{dx} \left( \Gamma d\phi \right)
$$

(5.2)

whilst the continuity equation is:

$$
\frac{d}{dx} (\rho u) = 0
$$

(5.3)

after integration of equation 5.2 over the control volume around node P (Figure 5.2):
\( (\rho u A \phi)_e - (\rho u A \phi)_w = \left( \Gamma A \frac{\partial \phi}{\partial x} \right)_e - \left( \Gamma A \frac{\partial \phi}{\partial x} \right)_w \)  
\( (5.4) \)

where \( A \) is the area of control volume’s faces and the spatial gradient of quantity \( \phi \) at the east face of the control volume can be approximated as:

\[
\left( \frac{\partial \phi}{\partial x} \right)_e = \frac{\phi_E - \phi_P}{x_E - x_P} = \frac{\phi_E - \phi_P}{\Delta x_e}
\]

where Subscripts P and E denote the node and its neighboring nodes at the west and east respectively and subscripts w and e refer to control volume faces. Integration of the continuity equation leads to:

\( (\rho u A)_e - (\rho u A)_w = 0 \)  
\( (5.5) \)

To simplify notation, two variables \( F \) and \( D \) are defined as the convective mass flux per unit area and diffusion conductance respectively:

\[
F = \rho u \\
D = \frac{\Gamma}{\Delta x}
\]

Assuming that \( A_w = A_e = A \), the convection-diffusion equation 5.4 can be re-written as:

\[
F_e \phi_e - F_w \phi_w = D_e (\phi_E - \phi_P) - D_w (\phi_P - \phi_W) \]  
\( (5.6) \)
and continuity equation:

\[ F_e - F_w = 0 \] (5.7)

Calculation of diffusive fluxes is not difficult. The approximation of the convective terms, on the other hand, is not as straightforward as that of the diffusive fluxes. It becomes necessary to calculate the transported property \( \phi \) at the \( e \) and \( w \) faces. In the STREAM code, there are three options for the difference scheme to approximate the convective term. They are UPWIND, QUICK and UMIST. In the TEAM code, there are two options for discretization of convective terms which are PLDS and QUICK.

The UPWIND scheme is formulated as follows:

\[
\phi_w = \begin{cases} 
\phi_W & U_w > 0 \\
\phi_P & U_w < 0 
\end{cases}
\] (5.8)

The advantage of this scheme is that it is a stable scheme. PLDS (Power Law Difference Scheme) is a convective discretization scheme available in the TEAM code. This scheme for a uniform mesh is formulated as follows when \( U_w \) is positive:

\[
\phi_w = \begin{cases} 
\phi_W + \frac{\phi_P - \phi_W}{2} \left( \frac{1-0.1Pe_w}{1-0.05Pe_w} \right)^5 & \text{for } 0 \leq Pe_w \leq 10 \\
\phi_W & \text{for } Pe_w > 10 
\end{cases}
\] (5.9)

\( Pe_w \) is the grid Peclet number and it is defined as:

\[
Pe_w = \left( \frac{U \Delta x}{\Gamma} \right)
\] (5.10)

This scheme and UPWIND scheme have some disadvantages. One is that there is large artificial diffusion when the grid is not fine enough. Also it is not very accurate in capturing twist and turns inside the velocity field because it only takes into account the velocity value of the neighboring node. But the good point about the PLDS is that it is a stable scheme. Another option available in STREAM is QUICK (Quadratic Upstream Interpolation for Convective Kinematics). This scheme for a uniform grid at the west cell face is formulated by fitting a quadratic function through 2 upwind and 1 downwind nodes as:
\[
\phi_w = \phi_W + \frac{\phi_P - \phi_W}{2} - \frac{\phi_P - 2\phi_W + \phi_{WW}}{8} \quad (5.11)
\]

similar expressions can be derived for the other faces of the control volume. This scheme is more accurate than PLDS and is capable of resolving the direction of velocity vectors better than PLDS. But the QUICK scheme is an unstable method because the equations are not bounded and may produce values less than the minimum nodal values or more than the maximum of the nodal values. The other available convection scheme in the STREAM code is Upstream Monotonic Interpolation for Scalar Transport Scheme (UMIST). This scheme calculates the face value by fitting a quadratic function through 2 upwind and 1 downwind nodes similar to QUICK scheme. The difference is that the scheme is designed to be bounded. This means that it makes sure that calculated face values stay between the minimum and maximum of nodal values. It can be formulated as:

for \( U_e > 0 \):

\[
\phi_e = \phi_P + 0.5(\phi_E - \phi_P) \left\{ \max \left[ \min \left( 2r, \frac{1}{4} + \frac{3}{4}r, \frac{3}{4} + \frac{1}{4}r, 2 \right) , 0 \right] \right\} \quad (5.12)
\]

where \( r = \frac{\phi_P - \phi_W}{\phi_E - \phi_P} \). And for \( U_e < 0 \):

\[
\phi_e = \phi_P + 0.5(\phi_P - \phi_E) \left\{ \max \left[ \min \left( 2r, \frac{1}{4} + \frac{3}{4}r, \frac{3}{4} + \frac{1}{4}r, 2 \right) , 0 \right] \right\} \quad (5.13)
\]

where \( r = \frac{\phi_E - \phi_W}{\phi_P - \phi_E} \)

Therefore UMIST scheme is more numerically stable than the other higher convective schemes like QUICK and in the meantime it keeps the accuracy at the same level as those more unstable schemes.

In the 2D simulations by the TEAM code, the QUICK scheme is used for the momentum transport equations and for the rest of equations, PLDS is employed. In the 3D simulations by the STREAM code, The UMIST scheme is used for the momentum transport equations and for the rest of equations, UPWIND is employed.
The convection-diffusion transport equation can easily be extended to two-dimensional problems by application of the derivation in the second direction. The discretized form of the two dimensional convection-diffusion transport equation is thus:

\[
F_e \phi_e - F_w \phi_w + F_s \phi_s - F_n \phi_n = D_e (\phi_E - \phi_P) - D_w (\phi_P - \phi_W) \\
+ D_s (\phi_S - \phi_P) - D_n (\phi_P - \phi_N) \tag{5.14}
\]

and continuity equation:

\[
F_e - F_w + F_s - F_n = 0 \tag{5.15}
\]

where subscripts S and N denote the neighboring nodes in the south and north directions respectively and subscripts s and n refer to control volume faces.

The general form of the transport equation for velocity (momentum equation) is:

\[
\frac{\partial}{\partial x} \left( \rho U_j U_i \right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right] \right) + S
\]

After discretising the convection-diffusion part of the momentum equation and differencing the gradient of the pressure, the general form of the discretised momentum equation is:

\[
a^U_e U_e = \sum a^U_i U_i + (P_P - P_E) (\Delta x) + S \Delta x
\]

where \( S \) refers to physical phenomena influential in the velocity field such as buoyancy force, acceleration due to rotation, etc.

### 5.4 The velocity-pressure linkage scheme

In incompressible flows, there is no equation for pressure explicitly. In order to resolve the pressure field, the continuity equation has to be employed. In the TEAM code there two options for velocity-pressure linkage algorithm.
The schemes are SIMPLE (“Semi-Implicit Method for Pressure-Linkage Equations”) and PISO (“Pressure Implicit Solution by Split Operator Method”). In the STREAM code the SIMPLE scheme is used for the velocity-pressure linkage algorithm. A summary of this velocity-pressure linkage algorithm is as follows.

The discretised equation for $U_e$ can be written as:

$$a^U_e U_e = \sum a^U_i U_i + (P_P - P_E) (\Delta x)$$

$$a^V_n V_n = \sum a^V_i V_i + (P_P - P_N) (\Delta x) \quad (5.16)$$

First, the pressure field $P^*$ is guessed and then from this assumed pressure field the discretized velocity field is calculated as:

$$a^U_e U^*_e = \sum a^U_i U^*_i + (P^*_P - P^*_E) (\Delta x)$$

$$a^V_n V^*_n = \sum a^V_i V^*_i + (P^*_P - P^*_N) (\Delta y) \quad (5.17)$$

with a similar discretized equation for $W$ in 3D calculations.

This resultant velocity field will not necessarily satisfy the continuity equation. In order to obtain the correct velocity and pressure field, we assume the corrections $U', V', (W'), P'$ should be added, so that:

$$U = U^* + U'$$

$$V = V^* + V'$$

$$P = P^* + P' \quad (5.18)$$

By subtracting equation 5.17 from equation 5.16, the velocity and pressure corrections can be related:
$$U' = \sum a_i^U U'_i + \left( P'_p - P'_e \right) \frac{\Delta x}{a_e^U}$$

$$V' = \sum a_i^V V'_i + \left( P'_p - P'_n \right) \frac{\Delta y}{a_n^V} \quad (5.19)$$

Assume that the pressure correction can be split into two parts:

$$P' = P'_1 + P'_2 \quad (5.20)$$

and for convenience we introduce the quantity $D_U$ and $D_V$, defined as:

$$D_U = \frac{\Delta x}{a_e^U}$$

$$D_V = \frac{\Delta y}{a_n^V} \quad (5.21)$$

Equation 5.19 can then be written as:

$$U'_e = U'_{e,1} + U'_{e,2}$$

$$V'_n = V'_{n,1} + V'_{n,2} \quad (5.22)$$

where the two parts of the corrections are chosen such that

$$U'_{e,1} = \left( P'_{p,1} - P'_{e,1} \right) D_U$$

$$V'_{n,1} = \left( P'_{p,1} - P'_{n,1} \right) D_V \quad (5.23)$$

$$U'_{e,2} = \sum a_i^U U'_i + \left( P'_{p,2} - P'_{e,2} \right) D_U$$

$$V'_{n,2} = \sum a_i^V V'_i + \left( P'_{p,2} - P'_{n,2} \right) D_V \quad (5.24)$$

Therefore the corrected velocity and pressure field are:

$$U_e = U'_e + U'_{e,1} + U'_{e,2}$$

$$V_n = V'_n + V'_{n,1} + V'_{n,2} \quad (5.25)$$
\[ P_e = P^* + P'_1 + P'_2 \]  

(5.26)

As a first approximation, the SIMPLE scheme assumes that \( U'_{e,2} = 0 \) and \( V'_{n,2} = 0 \). Then from equations 5.25 and 5.26:

\[
U^{**}_e = U^*_e + \left( P'_{P,1} - P'_{E,1} \right) D_U \\
V^{**}_n = V^*_n + \left( P'_{P,1} - P'_{N,1} \right) D_V
\]  

(5.27)

The velocities \( U^{**}, U^{**}_w, V^{**}, V^{**}_n \) must satisfy the discretized continuity equation, requiring that:

\[
\left[ (\rho U^{**})_w - (\rho U^{**})_e \right] \Delta x + \left[ (\rho V^{**})_n - (\rho V^{**})_s \right] \Delta y = 0
\]  

(5.28)

Substituting for \( U^{**} \) and \( V^{**} \) from equation 5.27 leads to a discretized equation for the pressure corrections \( P'_1 \), of the form:

\[
a_{P} P'_{P,1} = \sum a_{i}^{P'} P'_{i,1} + b_1
\]  

(5.29)

where

\[
b_1 = \left[ (\rho U^{*})_w - (\rho U^{*})_e \right] \Delta x + \left[ (\rho V^{*})_n - (\rho V^{*})_s \right] \Delta y
\]  

(5.30)

Solution of this equation produces a discretized pressure correction field, which is used to update the pressure field and also to correct the velocity field, through eqn 5.27, so that it can satisfy continuity. For a converged velocity field, solution of eqn 5.29 would result in zero \( P' \) values at all nodal locations.

In the SIMPLE algorithm, only \( P'_1 \) is used for the pressure field correction. In the PISO algorithm, a second correction \( P'_2 \) for pressure field is included. Therefore \( U'_{e,2} \) and \( V'_{n,2} \) which are assumed zero in the SIMPLE scheme are non-zero in the PISO scheme and can be calculated by equation 5.24. The sum of neighbouring corrections is based only on the first stage velocity corrections (so that is effectively a known quantity).
\[ U'_{e,2} = \sum a_i^{U} U_{i,1}^{'} + \left( P_{P,2}^{'} - P_{E,2}^{'} \right) D_U \]
\[ V'_{n,2} = \sum a_i^{V} V_{i,1}^{'} + \left( P_{P,2}^{'} - P_{N,2}^{'} \right) D_V \] (5.31)

Then the above equations are substituted into the continuity equation and results in:
\[ a_P^{P'} P_{P,2}^{'} = \sum a_i^{P'} P_{i,2}^{'} + b_2 \] (5.32)

where
\[ b_2 = \rho_w \left( \sum a_i^{U} U_{1,i}^{'} \right) w \Delta x - \rho_e \left( \sum a_i^{U} U_{1,i}^{'} \right) e \Delta x + \rho_s \left( \sum a_i^{V} V_{1,i}^{'} \right) s \Delta y - \rho_n \left( \sum a_i^{V} V_{1,i}^{'} \right) n \Delta y \] (5.33)

5.5 Sequence of solution steps

Solution of the discretized transport equations starts with initializing the variables such as \( U^*, V^*, P^*, k^* \) and \( \varepsilon^* \). With these initializations, coefficients of the discretized transport equations can be calculated. Then it is necessary to impose the boundary conditions by modifying the coefficients and source terms. In the next step, the discretized \( U \) and \( V \) (momentum equations) (and, in 3-D the \( W \) equation) can be solved. In order to impose continuity, the equation for \( P_1' \) must be solved and consequently the velocity field can be modified based on these pressure corrections. There are two options available in TEAM code for velocity-pressure linkage, namely the SIMPLE and PISO schemes. In the STREAM code only the SIMPLE scheme is available. If the SIMPLE scheme is taken up, after updating pressure, the updated velocity field are computed. The code then carries on to solve for other variables such as \( k \) and \( \varepsilon \) with the updated velocity field. This process must be repeated till the convergence criteria are reached. A flowchart of the calculation procedure is shown in Figure 5.3.
Figure 5.3 – Calculation procedure

1. Guess velocities and pressures
2. Calculate coefficients of equation 5.16
3. Solve $U_i^*$ from equation 5.16
4. Calculate coefficients and $b_1$ of equation 5.29
5. Solve $P_1'$ from equation 5.29
6. Update $P'$ from equation 5.26 with $P_2' = 0$
7. Update $U_i^{**}$ from equation 5.27
In the solution sequence, in order to prevent steep changes in the variables because of the strong non-linearities of the transport equations, under-relaxation is used to slow down the rate of the solution process. The under-relaxation can be implemented as:

\[ \phi_P = \alpha \phi_P^{new} + (1 - \alpha) \phi_P^{old} \]  

where “old” and “new” superscripts refer to old and current iterations and the under-relaxation factor \( \alpha \) has a value \( 0 < \alpha < 1 \).

This under-relaxation can be introduced directly to the discretized equations of momentum, temperature, \( k, \varepsilon \), etc. as:

\[ \frac{a_P^\phi}{\alpha} \phi_P = \sum_i a_i^\phi \phi_i + S^\phi + \frac{(1 - \alpha)}{\alpha} a_P^\phi \phi_P^{old} \]  

In the pressure correction equation, it is implemented by adding only a fraction of \( P' \) to \( P^* \) as:

\[ P = P^* + \alpha P' \]  

The under-relaxation \( \alpha \) is set to 0.1 in all of the transport equations and pressure-correction for all of the 2D simulations. For 3D steady-state simulations of inclined cavities, 0.1 is used as under-relaxation factor. The value of under-relaxation is set to 0.2 for the transport equations and 0.1 for pressure-correction in 3D time-dependent simulations of inclined cavities. For the 3D simulation of the horizontal circular cavity, 0.2 is the under-relaxation factor for the transport equations while 0.1 is set for under-relaxation of pressure-correction. The under-relaxation factors needed to be as low as the values mentioned above, at least during some stages of the calculations, since computations carried out with larger under-relaxation factors became unstable and led to divergence.
5.6 Time-dependent computations

In this research, time-dependent computations have also been carried out for all of the test cases in order to determine whether the modelling and computational procedure will result in purely steady state flows or not. Here the Crank-Nicolson and fully implicit schemes have been employed for temporal discretization. To explain how these are implemented within the current strategy, the general form of the transport equations can be written as:

\[
\frac{\partial}{\partial \tau} (\rho \phi) + \text{div} (\rho u \phi) = \text{div} (\Gamma \text{grad} \phi) + S_\phi
\]

where \( \phi \) is an arbitrary variable, \( \Gamma \) is the diffusion coefficient and \( \tau \) represents time. To discretise the above equation, it is integrated over the control volume and time interval \( \tau \) to \( \tau + \Delta \tau \). There are two different treatments for evaluating the time integrals employed in this study, known as Crank-Nicolson and the fully implicit. They can be expressed in the following formulation to approximate the integral of a general variable \( \phi \) over the time step \( \tau \) to \( \tau + \Delta \tau \):

\[
\int_{\tau}^{\tau + \Delta \tau} f(\tau) d\tau = (\theta f(\tau + \Delta \tau) + (1 - \theta)f(\tau)) \Delta \tau
\]

After discretisation of the one dimensional general form of a transport equation based on the above equation, the equation is written as:

\[
a_P \phi_P = a_W [\theta \phi_W + (1 - \theta) \phi_W^0] + a_E [\theta \phi_E + (1 - \theta) \phi_E^0] \\
+ [a_p^0 - (1 - \theta) a_W - (1 - \theta) a_E] \phi_P^0 + b
\]

(5.38)

where

\[
a_P = \theta (a_W + a_E) + a_P^0
\]

\[
a_P^0 = \rho_p^0 \frac{\Delta x}{\Delta \tau}
\]

\[
b = \overline{S} \Delta x
\]

Equation 5.38 is dependent on the parameter \( \theta \). In the case that \( \theta = 1/2 \), the value of \( \phi \) at the new time level is calculated by the \( \phi \) values at the old and new
time levels. This is called the Crank-Nicolson scheme. In the case that $\theta = 1$, the value of $\phi$ at the new time level is calculated totally based on the $\phi$ values at the new time step. This is called the fully implicit scheme. In the present study, all the time-dependent calculation were carried out by the fully implicit scheme. This scheme is a bounded method and there is not any criteria for the stability of the time-dependent computations regarding the selection of time steps. To ensure accuracy of the time-dependent computations, appropriate time-step is selected as explained here. Initially steady-state computation is carried out. The steady-state calculation gives the maximum velocity exists in the computational domain. Then maximum velocity and smallest cell size in the grid give us an indication of the smallest time scale that may occur in the large scale flow field. Finally the time-step is chosen less than the calculated time scale. This assures that CFL values across computational domain are less than unity where CFL is defined as:

$$\text{CFL} = \frac{\text{velocity} \times \text{time step}}{\text{cell size}}$$

This is believed that the condition can satisfy the accuracy of the time-dependent computations.

### 5.7 Boundary conditions

The test cases studied here involved enclosed cavities and an open cavity, so the boundary conditions that need to be considered are the wall boundary condition and non-wall boundary conditions. The wall boundary condition implementation for low-Re-number-model and high-Re-number-model computations are somewhat different. In the low-Re-number simulations, all velocity components are zero in the case that the wall is stationary. Regarding $k$ and $\varepsilon$, in the Launder-Sharma model which is employed in this study, the values of $k$ and $\varepsilon$ on the walls are set to zero. In the high Reynolds-number model calculations wall boundaries are treated via wall functions as described in Chapter 4.
For the thermal boundary conditions, two different conditions have been used in the current research. One is that of prescribed wall temperature, which here can be constant, uniform, and non-uniform. The other thermal boundary condition is that of an insulated wall. In order to simulate constant temperature wall boundary conditions in low-Re-number computations, $T_{wall}$ is assigned to the temperature at the wall node. To simulate the insulated wall, the wall heat flux must be zero and according to Fick’s law:

$$ q_{wall} = -\lambda \left( \frac{\partial \Theta}{\partial y} \right)_{wall} $$  \hspace{1cm} (5.39)

To have zero wall heat flux, the gradient of temperature at the wall must therefore be zero. Therefore, in a low-Re-model, the temperature at the node on the wall and the node adjacent to the wall are set equal.

For the high-Re-number computations, a different approach is adopted. In the momentum equations, the wall shear stress (multiplied by the cell face area), which is calculated from the wall-function equations, is added to the source term of near wall cells, and the matrix coefficient which links the near wall node and the wall is set to zero. For the turbulent kinetic energy transport equation, the production and dissipation source terms for near wall cells are calculated according to the wall-function equations. The $\varepsilon$ transport equation is not solved for near wall cells. Instead, the value of $\varepsilon_P$ is assigned to near wall cells, as:

$$ \varepsilon_P = \frac{k^{3/2}}{c_{t}u_{P}} $$  \hspace{1cm} (5.40)

Therefore, in the $\varepsilon$ equation the near wall cell is effectively treated as the boundary condition for the rest of the domain.

For the temperature transport equation, on boundaries at which the thermal conditions are those of prescribed wall temperature, the wall heat flux is calculated from the wall-function equations and then adding into the discretized source term as

$$ S_U = S_U + \frac{q_{wall}}{c_p} Area $$  \hspace{1cm} (5.41)
The matrix coefficient that links the near wall node and the wall is set to zero.

Regarding the non-wall boundary condition, three types of boundary conditions are employed in the study. Two of them are inlet and outlet boundaries across which the fluid is entrained into the computational domain and exits it respectively. The other boundaries employed in the present work are treated as symmetry boundaries. At this type of boundary, the fluid neither enters to the domain nor exits.

A fixed constant temperature is set for the inlet boundary condition and zero gradient condition is imposed for the rest of the variables. Zero gradient conditions are also set for all variables at the outlet of the computational domain. For the symmetry boundaries, the component of velocity perpendicular to the boundary is set to zero and zero gradient conditions are imposed to the rest of the variables.

5.8 Simulation of inclined cavity

The most expedient way to simulate inclined cavity flows was to rotate the coordinate system and consequently the velocity components relative to the vertical, as shown in Figure 5.4. This means that the gravity vector now has components in both the x and y directions.

![Figure 5.4 – Schematic of the inclined cavity.](image)

The components of gravity acceleration are:
\[ g_x = g \cos(\Psi) \]  \hspace{1cm} (5.42)

\[ g_y = -g \sin(\Psi) \]  \hspace{1cm} (5.43)

where \( g \) is the magnitude of the gravitational acceleration vector.

### 5.9 Implementation of buoyancy related terms in the STREAM code

The version of the STREAM code originally employed here did not include buoyancy terms in the flow and turbulence transport equations. Therefore these equations had to be amended to include buoyancy related terms. The terms from equation 4.52 were included in the mean momentum and turbulence transport equations, whilst buoyancy-related contributions were also added to the AWF coding. Again, as above, these were coded in such a way that the angle of inclination could be altered by suitable definitions of \( g_x \) and \( g_y \).

### 5.10 Closing Remarks

In this Chapter, two in-house solvers used for the simulations are introduced. The main numerical methods which are implemented in this research were then discussed. The way that boundary conditions are defined numerically was discussed as well. The extra terms for time dependent calculations are shown. Finally, the way in which the inclined cavity geometries were defined, was discussed.
Chapter 6

Inclined Cavity-2D simulations

This chapter is divided in six main sections which correspond to the three test cases which have been computed in the study and one section at the end for closing remarks. They all relate to the study of tall cavities with the same aspect ratio, but with different angles of inclination. The TEAM code was used for all of the computations presented in this chapter. In the cases reported here the tall walls make a $5^\circ$ stable, a $60^\circ$ stable or a $60^\circ$ unstable angle with the horizontal, where in the stable cases the lower surface is maintained at a lower temperature. The three test cases have been selected so that they cover various physical phenomena. Results of the RANS turbulence modelling are compared to available LES data[41] for the $5^\circ$ stable case and to available experimental data [67] for the other two. For the $5^\circ$ stable case, the LES data used for validation were computed assuming repeating flow conditions in the span-wise direction and thus ignoring the effects of side walls. In section 6.2, the test case used for the $60^\circ$ stable and unstable is described in detail. In section 6.3, an overview of the effect of the angle of inclination on the flow development is presented, through predictions at regular intervals of angles of inclination, which range from $0^\circ$ stable to $0^\circ$ unstable. In the sections 6.4-6.6, 2D RANS simulations of the tall cavity are presented at angles $60^\circ$ stable, $60^\circ$ unstable and $5^\circ$ stable. For the $5^\circ$ stable test case, the $k-\varepsilon$ model, with the AWF, SWF wall functions and also with the Launder-Sharma low-Re extension, LRN, is employed to simulate the flow within the cavity. In addition, a stress transport model has been tested for the stable $60^\circ$ and the unstable $60^\circ$ cases. For the $60^\circ$
stable and unstable inclined tall cavities a more elaborate equation for the turbu-
bulent heat fluxes has been employed as well, when the stress transport model
was in use. In this case note that attention is also drawn to the similarities and
differences with previously reported results of the vertical, 90° case[39]. In the
5° and 60° stable cases, the hot wall is located above the cold wall. Therefore
this creates a stable structure within the cavity. The reason is that in this con-
figuration, the hot fluid adjacent to the top wall has a lower density than that
of the cold fluid adjacent to the lower wall. Consequently, the buoyancy force
tries to confine the hot fluid to the top of the cavity and the cold to the bottom,
which leads to thermal stratification. For the 60° unstable case the hot wall
is located below the cold wall which produces unstable thermal stratification.
As it turned out, flow in the stable and unstable 60° cavities did not deviate
radically from that in the vertical case. Nevertheless, the recent emergence of
detailed experimental data, ensures that these two remain very attractive test
cases. Finally, in Section 6.7, the closing remarks are presented.

6.1 Grid

Following grid sensitivity tests for the test cases (Appendix A), the follow-
ing grids have been adopted to carry out numerical analysis. For the inclined
tall cavity with the heated walls at angles of inclination stable 60° and unsta-
ble 60°, when the low-Re-number $k-\varepsilon$ is employed a 100x270 grid (Figure 6.1)
is used. As can be seen in this figure, even though the nodes are concentrated
near the walls, the grid resolution away from the walls is relatively fine. Use
of this type of mesh is necessary because of the possible existence of multiple
recirculating cells within the inclined tall cavity under unstable heating con-
ditions. In the cases of the high-Re-number $k-\varepsilon$ and RSM, after extensive grid
sensitivity tests, a 40x220 grid (Figure 6.2) is used.
As intended, introduction of the wall function approach removes the need to use a fine mesh near the walls. In order to assess the degree of near-wall grid refinement, it is often convenient to examine the non-dimensional distance to the wall $y^*$, of the near-wall node. In the low-Reynolds-number near wall strategy the near-wall $y^*$ value should be less than 1, in order to ensure that enough nodes are located in viscous sublayer to capture the sharp flow.
variations. In these calculations, $y^*$ values vary from $10^{-4}$ and 0.062. The low $y^*$ values are due to the fact that the high turbulence is located at the core of the cavity, whilst the rather low near-wall turbulence levels result in low $y^*$ values. In the wall function approach, the first node should be located outside the viscous sublayer and the value of $y^*$ should therefore be in the range of 20 to several hundreds. The $y^*$ values of the near-wall nodes for grids used in the high-Re-model computations were in the range of 4 to 22. These test cases are relatively low turbulence flows. Therefore there was a compromise between increasing $y^*$ values of the near-wall nodes and the minimum number of nodes needed between the hot and cold walls to capture the buoyant flow. Due to low level of turbulence, the low-Re terms in the $k$ and $\varepsilon$ transport equations were also included.

Regarding the inclined $5^\circ$ stable tall cavity, the 80x180 grid was adopted. The reason the grid for this test case is different to those of the other test cases is that the flow inside this cavity has lower turbulence levels which lead to low $y^*$ values. Therefore a coarser grid can be employed for this test case. In common with the grid used for computations in the $60^\circ$ stable inclined tall cavity, the grid for low-Re-number k-$\varepsilon$ is generated (Figure 6.3) so that there is fine mesh near the walls to resolve the variation near the walls. Figure 6.4 shows the grid employed for computations with the wall function approach for the $5^\circ$ stable inclined tall cavity. It is a 40x180 grid.
6.2 Test case description

The test cases all relate to a tall cavity with aspect ratio of 28.7, where the tall heated and cooled walls are at different angles of inclinations to the horizontal. The end walls are insulated. Experimental data are available for a
Rayleigh number \( Ra = \frac{\rho g \Delta \Theta L^3}{\nu \alpha} = 0.8 \times 10^6 \) (Cooper et al. [8]) for the 60\(^\circ\) stable and unstable, and the 15\(^\circ\) stable and unstable configurations. The hot and cold wall temperatures are 34\(^\circ\)C and 16\(^\circ\)C respectively. These uniform wall temperatures are produced by imposing heat flux to the corresponding walls. The fluid inside the tall cavity is air. The height of the cavity, \( H \), is 2.18m and its width, \( L \), is 0.076m. The spanwise dimension of the cavity used in the experiment was 0.52m. Comparisons with the experimental traverses (between the hot and cold sides) of [8] have been made along 4 different cross sections, which are shown in Figure 6.5. The Nusselt numbers are also compared with available experimental data along the hot and cold walls. The local Nusselt numbers are calculated as \( Nu = -L \frac{(\partial \Theta / \partial x_i)_w}{(\Theta_h - \Theta_c)} \), where \( L \) is the distance between the hot and the cold walls. The velocity scale is defined as \( V_0 = \sqrt{g\beta L \Delta \Theta} \), when \( \beta \) is the volumetric expansion factor and \( \Delta \Theta = \Theta_h - \Theta_c \).

**Figure 6.5** – Schematic of inclined tall cavity and its thermal boundary condition.

Figure 6.6 shows the experimental data of wall-parallel velocity and rms velocity fluctuations in the 60\(^\circ\) stable inclined cavity. It shows the velocities at the mid-height (\( y/H=0.5 \)) and near the two ends of the cavity (\( y/H=0.1 \) and 0.95). At each longitudinal section, the profiles at the centreline in the spanwise direction (\( z/D=0 \)) are compared with the measured velocities off the centreline (\( z/D=0.25 \)). The experimental data reported that in this cavity, as
suggested by Figure 6.6, the flow was two-dimensional in the most of the cavity, apart from the regions near the two cavity ends. To illustrate this last point, Figure 6.7 shows the measured wall-parallel velocity near the end of the cavity (y/H=0.95) at the centre-plane (z/D=0). As can be inferred from Figure 6.7, the amount of the fluid moving up in this place is not equal to the fluid which is moving down. This means that some fluid must be moving in the third direction. Therefore, some of the discrepancies to be seen between data and 2D numerical simulations at Y=0.95 may be related to the three dimensionality of the flow in this region.

Figure 6.6 – V and \( V_{rms} \) profiles at and off, the centreline of the 60° stable cavity, to illustrate the two-dimensionality of the flow, Cooper et al. [8]
Figure 6.7 – Velocity profile in the 60° stable cavity at the centreline where $Y=0.95$, Cooper et al. [8]

Figure 6.8 – Temperature profiles at, and off, the centreline of the 60° unstable cavity, to illustrate the two-dimensionality of the flow, Cooper et al. [8]
For the 60° unstable cavity, the experimental data is available only for the thermal field which is shown to be reasonably two dimensional (Figure 6.8). From the above comments on the experimental data, it is believed to be appropriate to computationally model the 60° stable and unstable cavity configurations as two-dimensional problems.

Conversely, the experimental data implied the existence of strong three-dimensional flows in the 15° stable and unstable cavity configurations, as shown in Figures 6.9-6.10. Figure 6.9 shows measured wall-parallel velocity profiles at the mid-height (y/H=0.5) and near the two ends of the cavity (y/H=0.1 and 0.9) in the 15° stable inclined cavity. As can be seen these are significant variations in the velocity across the cavity span, indicating strong three-dimensional effects. In the 15° unstable inclination case the velocity component normal to the long wall is, on average, more dominant than that parallel to them, and so to illustrate the three-dimensional effects in this case, Figure 6.10 shows profiles of this velocity component across the spanwise direction at the mid-height (y/H=0.5) and near the two cavity ends (y/H=0.1 and 0.9). Again, the significant spanwise variations in the velocity indicate strong three-dimensional flow features.

Figure 6.9 – Wall-parallel velocity profiles in the 15° stable cavity along the spanwise direction where X=0.5, to illustrate the three-dimensionality of the flow, Cooper et al. [8]
6.3 Overview of 2-dimensional flow in the tall, differentially heated cavities at different angles of inclination

Figures 6.11-6.19 show the streamlines and temperature contours within inclined tall cavities, with different angles of inclination in stable or unstable configurations. The figures show that when the cavity is horizontal and stable, at $0^\circ$, the flow is stagnant throughout the cavity. When the cavity is inclined at $5^\circ$ stable configuration, due to density variation along the horizontal direction, the imbalance between the hydrostatic pressure force and the gravitational force generates fluid motion, parallel to the thermally active walls of the cavity. This in turn creates circulation inside the cavity. As this angle of inclination increases towards the vertical, the component of gravitational acceleration parallel to the tall wall increases. This gives rise to stronger circulations.

When the cavity is inclined at the $10^\circ$ unstable angle, the hot wall is located below the cold wall. This means that hot fluid with lower density is located below the cold fluid with higher density. The buoyancy force moves the hot fluid upwards and cold fluid downwards. As it is shown in the figures, there are multiple circulation cells inside the $10^\circ$, $5^\circ$ and $0^\circ$ unstable cavities due to unstable thermal stratification.

Figure 6.10 – Wall-normal velocity profiles in the $15^\circ$ unstable cavity along the spanwise direction where $X=0.5$, to illustrate the three-dimensionality of the flow, Cooper et al. [8]
Temperature contours

Stream lines

Figure 6.11 – Temperature and stream lines within a 0° stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figure 6.12 – Temperature and stream lines within a $5^\circ$ stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Temperature contours

Stream lines

**Figure 6.13** – Temperature and stream lines within a $10^\circ$ stable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figure 6.14 – Temperature and stream lines within a 15° stable inclined cavity resulting from RSM-Basic (Ra = 0.86 \times 10^6), (scale of the small side is enlarged by a factor of 7).
Figure 6.15 – Temperature and stream lines within a 60° stable inclined cavity resulting from RSM-Basic (Ra = $0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figure 6.16 – Temperature and stream lines within a $90^\circ$ vertical cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figure 6.17 – Temperature and stream lines within a $10^\circ$ unstable inclined cavity resulting from RSM-Basic ($Ra = 0.86 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figures 6.18-6.23 show the predicted velocity and turbulent kinetic energy
profiles between the two thermally active walls, at the middle of the cavity, at different angles of inclination, with the $k-\varepsilon$ model, using LRN, AWF and SWF and also with the Basic RSM using AWF. As is shown in the diagrams, all the turbulence models predict that the highest turbulent kinetic energy is found in the vertical cavity case. This is contrary to what it might be expected, that the turbulence level would increase as the cavity is tilted toward the unstably stratified case. The explanation, as mentioned in the previous sections, is that based on the experimental study of the $15^\circ$ unstable inclined tall cavity, there are significant 3-dimensional flow structures within the cavity, whereas the numerical simulations are purely 2-dimensional. In the case where the cavity is tilted to give stably stratified flow, all the turbulence models behave as expected. They show that the turbulence level is reduced as the angle of inclination of the tall cavity is reduced, until the flow becomes fully stagnant at $0^\circ$ stable.

Regarding the mean velocity distribution between the two thermally active walls at the middle of the cavity, again all turbulence models predict velocity profiles as expected for the vertical and stably stratified cases. It is worth to note that there might be some discrepancies between 2D numerical simulations of unstable flows and the real thermally unstable flows. This is based on the fact that in reality in unstably stratified cases, there are significant 3-dimensional flow structures, whereas the numerical results are based on purely 2-dimensional simulations. The other source of discrepancy in the velocity profiles for the unstably stratified flows is related to the fact that at some of the angles of inclinations some turbulence models return multiple cell circulations. The velocity profiles between the two thermally active walls shown in Figures 6.20-6.23, which are shown along the section at the middle of the cavity, which in the cases with multiple circulation cells, may lie on the weaker or stronger part of cells, or even the direction of the circulation may be different from one turbulence model to another along the section at the middle of the cavity.

In Figure 6.24, the turbulent kinetic energy distributions along the line normal to the insulated end-walls, at the middle of the cavity are shown. The
diagrams show that the LRN $k-\varepsilon$ starts predicting multiple cell circulations from an angle of inclination of $10^\circ$ unstable, when the k results can be seen to oscillate along that section of the cavity. The angle at which multiple cells appear for the $k-\varepsilon$ model using AWF is $5^\circ$ unstable and for the $k-\varepsilon$ model using SWF and Basic RSM using AWF, it is $15^\circ$ unstable.

**Figure 6.20** – Mean velocity and $k$ profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k-\varepsilon$-LRN.
Figure 6.21 – Mean velocity and k profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k-\varepsilon$-AWF.
Figure 6.22 – Mean velocity and k profiles at mid-height of the tall cavity at different angles of inclinations resulting from $k$-$\varepsilon$-SWF.
Figure 6.23 – Mean velocity and k profiles at mid-height of the tall cavity at different angles of inclinations resulting from RSM-Basic-AWF.
6.4 Inclined tall cavity - $60^\circ$ stable

6.4.1 Flow Pattern (including comparison with $90^\circ$ case)

The predicted flow pattern inside the $60^\circ$ stable inclined tall cavity is shown in Figures 6.25-6.26. In this test case, because of the temperature difference between the tall walls, air near the hot wall moves up and then impinges on to the top insulated wall and moves over the top insulated wall, until it impinges on to the cold wall and becomes cold and moves down. The air then moves down the cold wall and impinges on to the bottom insulated wall, changes direction and flows over the bottom wall until it impinges on to the hot wall. This creates a single cell circulation inside the inclined tall cavity. Figure 6.25 shows that there is turbulent flow in the cavity core. The temperature contours reveal that most of the temperature change from the hot to the cold side is confined to the thin layers along the two thermally active walls, with only a gentle temperature change across the cavity core.
Inclined Cavity-2D simulations

Figure 6.25 – Temperature and turbulent kinetic energy contours in tall cavity resulting from $k$-$\varepsilon$ using LRN ($Ra = 0.8 \times 10^6$), (scale of the small side is enlarged by a factor of 7).
Figure 6.26 – Mean velocity vector plot and stream lines in tall cavity resulting from $k$-$\varepsilon$ using LRN ($Ra = 0.8 \times 10^6$), (scale of the small side is enlarged by a factor of 7).

In Figure 6.27, the predicted temperature contours and stream lines for the vertical and inclined $60^\circ$ stable tall cavities are compared to each other. The temperature contours show that the temperature in the core of the tall cavity remains almost constant and equal to the average temperature of the hot and the cold walls. The temperature distribution near the top and the bottom walls is slightly modified, due to the inclination in the $60^\circ$ stable case. The figures show that the inclination assists the hot fluid to penetrate further toward the cold wall near the top end of the cavity and the cold fluid to penetrate further toward the hot wall near the lower end. Consistent with the temperature contours, the stream traces remained almost unchanged apart from slight changes at the two ends.
Figure 6.27 – Temperature contours and stream traces within vertical and inclined 60° stable tall cavity resulting from $k$-$\varepsilon$ using LRN, (scale of the small side is enlarged by a factor of 7).
6.4.2 $k$-$\varepsilon$ predictions

In Figure 6.28, the temperature distributions within the vertical and inclined tall cavities, resulting from $k$-$\varepsilon$ with LRN, AWF and SWF approaches, are compared to each other and also to experimental data. The experimental measurements confirm that there is a slight change in temperature distributions from the vertical cavity to the inclined $60^\circ$ stable cavity. The experimental measurements show that the temperature profiles in the core of the inclined cavity are slightly lower than those for the vertical cavity. This can be explained by the fact that in the inclined tall cavity the component of the buoyancy force parallel to the tall walls is weaker than that in the vertical tall cavity. This in turn causes a reduction of the component of velocity parallel to the tall walls. Then convective heat transfer is expected to decrease because of the reduction of the velocity. Despite these differences the shape of the temperature distributions for the vertical and inclined tall cavities are quite similar to each other.

The comparisons show that there are only small differences among the three sets of predictions. Over most of the cavity, all models return a reasonable temperature variation. Closer to the top and bottom end walls, however, the predictions of the two high-Re models start to deviate from the data for the vertical tall cavity, while those of the low-Re $k$-$\varepsilon$ continue to remain close. This suggests that the low-Re model is better able to cope with the flow impingement. This discrepancy between model predictions becomes smaller for the temperature distributions into the $60^\circ$ stable inclined cavity. It may be concluded that because of the reduction of velocity parallel to the tall walls in the $60^\circ$ stable case, compared with the vertical cavity, there is weaker impingement at the top and bottom of the tall cavity, and therefore the High-Re models predict temperature distributions in those regions that are in closer agreement with those of the LRN schemes and, in this case, close to the experimental measurements.
Figure 6.28 – Temperature distributions inside vertical and 60° stable inclined tall cavities resulting from \( k-\varepsilon \) using LRN, AWF and SWF.
In Figure 6.29, the Nusselt number predictions from the $k$-$\varepsilon$ model, using LRN, AWF and SWF approaches in the vertical and stable $60^\circ$ inclined cavities are compared to each other. These predictions are also compared to the Nusselt number measurements for the inclined tall cavity. Figure 6.30 shows the experimental Nusselt number distribution along the cold wall of the stable $60^\circ$ inclined cavity compared with the Nusselt prediction resulting from the $k$-$\varepsilon$-AWF. Based on the Nusselt number measurements for the inclined $60^\circ$ stable cavity, the Nusselt number distribution for this case is slightly lower than in the vertical case based on the explanation given in the previous section. The measurements show that the peak Nusselt numbers are located at the bottom of the hot wall and the top, along the cold wall. The high velocity fluid at the bottom, along the hot wall impinges to the bottom of the hot wall and causes this increase of the Nusselt number. Then the Nusselt number remains nearly constant along the middle of the hot and cold walls, before decreasing because of the separation of the fluid at the top of the hot wall and the bottom of the cold wall.

The predicted Nusselt numbers for the stable $60^\circ$ inclined tall cavity are generally slightly lower than those in the vertical cavity as expected. The exception to this is at the bottom of the hot wall where the SWF treatment results in slightly higher Nusselt numbers for the inclined case than for the vertical cavity. This may be another indication that the SWF does not accurately capture the effects of the impingement here.

**Figure 6.29** – Nu distribution along cold wall of vertical and $60^\circ$ stable inclined tall cavity resulting from $k$-$\varepsilon$ using Low-Re-Number and AWF and SWF.
Figure 6.30 – Nusselt distribution along the cold wall of 60° stable inclined tall cavity resulting from $k$-$\varepsilon$-AWF.

Figure 6.31 – Mean velocity distributions within a 60° stable inclined tall cavity resulting from $k$-$\varepsilon$ using LRN, AWF and SWF.

Figure 6.31 shows mean velocities parallel to the hot and cold walls resulting from the $k$-$\varepsilon$ model, using LRN, AWF and SWF compared with the corresponding experimental measurements. The data shows that the fluid moves up parallel to the hot wall from the bottom of the hot wall with high velocity and decelerates as it approaches the top of the hot wall. Then the fluid
moves down from the top of the cold wall with high velocity and decelerates as it approaches the bottom of the cavity. All of the near-wall treatments produce results which are broadly in agreement with the experimental data. One inconsistency exists at the top of the cold wall where the turbulence models predict higher velocities compared with the experiment. Both numerical and experimental data show higher velocity at the bottom of the hot wall. The hydrodynamic flow pattern at the bottom of the hot wall is similar to that at the top of the cold wall. Therefore some portion of this inconsistency might be from the fact that at those regions the flow involves impingement and change of direction, the flow is more complicated at the two ends than at the other regions. The models employed therefore, possibly due to the use of the effective viscosity approximation, are unable to fully reproduce the measured mean flow development at the two ends. Moreover, AWF predicts higher velocity compared with the other models. As it shown, at the bottom of hot wall, AWF predicted higher velocity which continued to the top of the hot wall. This may be caused due to higher turbulence resulted from $k-\varepsilon$-AWF adjacent to the walls (Figure 6.32).

In Figure 6.32, the rms velocity fluctuations resulting from the $k-\varepsilon$ using LRN, AWF and SWF are compared for the vertical and inclined cavities. Measurements for the inclined case are also included. The experimental measurements reveal that the turbulent stresses are higher at the positions near the mid-height of the tall cavity compared to those near to the top and bottom of the cavity. Turbulence is generated due to the mean velocity gradients (mean flow shear) and buoyancy (mean temperature gradients). As can be seen in the mean velocity profiles of Figure 6.31, the mean flow shear is stronger at the middle of the cavity. These comparisons show that all turbulence models significantly under-predict the near-wall turbulence levels by as much as 20 percent in the core region. On a more positive note, all predictions reproduce the severe reduction in turbulence levels near the end walls.

Generally the predictions from the LRN, AWF and SWF approaches are quite similar. This is because of the use of the eddy-viscosity approximation in all three sets of computations, which is a rather simple relation and is not able
to reproduce the effect of complex flows and body forces on turbulence. Over most of the cavity the $V_{rms}$ levels in the vertical cavity are under-estimated by the EVM model. The numerical predictions of velocity fluctuations for the 60° stable inclined tall cavity are generally lower than those in the vertical cavity. The reason is again that in the inclined cavity there is a weaker convective flow parallel to the tall walls, which causes a reduction of generation of turbulence within the inclined tall cavity.
Figure 6.32 – rms velocity fluctuations distributions within vertical and tilted $60^\circ$ stable tall cavity resulting from $k-\varepsilon$ using LRN, AWF and SWF.

6.4.3 RSM predictions

In Figures 6.33-6.34, the temperature distribution along cross sections at different levels of the cavity are shown, resulting from the Basic Reynolds stress model and the Two-Component-Limit (TCL) version. Figure 6.33 shows
that the Basic Reynolds stress model predicts the temperature distribution quite well at the middle of the cavity. Its predictions deviate slightly from the experimental measurements near the two ends of the cavity. The shape of the temperature profile at the top of the cavity near the cold wall and at the bottom of the cavity near the hot wall suggests that in these regions the model over-predicts turbulence level compared to experimental data. Likewise in Figure 6.34, this turbulence over-prediction is increased by employing the TCL Reynolds stress model. The turbulence over-prediction resulting from the Reynolds stress models has already been observed in the vertical case [39], but it seems that tilting the tall cavity augments the problem. Figures 6.35-6.36 show shear stress distributions resulting from RSM-Basic and TCL compared with $k-\varepsilon$-AWF. The comparisons show that RSM-Basic and TCL both over-predict turbulent shear stress near the two ends of the cavity compared with the $k-\varepsilon$-AWF prediction. This over-prediction becomes significant where the RSM-TCL is employed. The overestimation of turbulence levels by the RSM near the two end-walls, is consistent with the observed deviations between the measured thermal behaviour and that returned by the RSM.

Figure 6.37 shows the prediction of Nu number distribution along the cold wall of the $60^\circ$ stable cavity resulting from RSM compared with the experimental data. The comparison shows there is a good match between RSM predictions and experimental data. It also shows some improvements in Nu predictions compared with $k-\varepsilon$-AWF (Figure 6.30). It may be due to more accurate calculation of stresses and turbulent heat fluxes by RSM at the near-wall regions.
Figure 6.33 – Temperature distributions resulting from RSM-basic(AWF).
Figure 6.34 – Temperature distributions resulting from RSM-TCL(AWF).
Figure 6.35 – Shear stress distributions resulting from RSM-basic(AWF).
Figure 6.36 – Shear stress distributions resulting from RSM-TCL(AWF).
Figure 6.37 – Nusselt distribution along the cold wall of 60° stable inclined tall cavity resulting from RSM.

6.4.4 Employment of $\theta^2$ and $\varepsilon_\theta$ transport equations

In Figure 6.38, the predicted temperature distribution produced by the $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$ model using AWF is compared with the experimental measurements. The figures show that the turbulence model captures the temperature distributions reasonably well. However, because the simple $k$-$\varepsilon$ model (on which the 4 equations scheme is based) simulates the flow within the inclined 60° stable cavity equally well, we can not conclude that introduction of the 4-equation version results in any significant additional predictive benefits for this test case.

In Figure 6.39, the temperature distributions at different heights of the tall cavity resulting from the RSM and RSM-$\theta^2$-$\varepsilon_\theta$ - Basic and TCL schemes using the AWF are compared with experimental measurements. The diagrams show that in the case of the Basic RSM there is a slight improvement in the prediction of temperature near the top and the bottom of the cavity by employing the additional $\theta^2$ and $\varepsilon_\theta$ equations in the thermal field. The improvement is more significant regarding the TCL version of the Reynolds Stress Model. The diagrams show that after implementing the RSM-$\theta^2$-$\varepsilon_\theta$ - TCL, there is significant improvement in the prediction of temperature near the top and bottom of the cavity. Nevertheless, there is still room for some further improvement.

Figures 6.40-6.41 show the mean velocity predictions resulting from RSM-Basic, RSM-Basic-$\theta^2$-$\varepsilon_\theta$, RSM-TCL and RSM-$\theta^2$-$\varepsilon_\theta$ - TCL compared with the
corresponding experimental measurements at the different sections from the bottom to top of the cavity. The comparisons show that, perhaps contrary to expectations, the mean velocity profiles produced by the RSM, are not as close to the experimental ones, as those produced by the $k-\varepsilon$ model. As can be seen in Figures 6.42 and 6.43, introduction of the RSM models result in an increase in the predicted turbulence levels. This change is certainly consistent with the higher mean shear predicted by the RSM models at the cavity core. Also the comparisons show that employment of $\theta^2$ and $\varepsilon_\theta$ transport equations, with the more elaborate algebraic model for the turbulent heat fluxes, does not have a significant effect on the velocity and stress field results except at the top of the cavity when RSM-$\theta^2-\varepsilon_\theta$ - TCL is used.
Figure 6.38 – Temperature distributions resulting from $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$ - AWF.
Figure 6.39 – Temperature distributions resulting from RSM-$\theta^2\varepsilon_\theta$ - Basic and TCL.
Figure 6.40 – Mean velocity distributions within a 60° stable inclined tall cavity resulting from RSM-Basic and RSM-θ^2-ε_θ - Basic.

Figure 6.41 – Mean velocity distributions within a 60° stable inclined tall cavity resulting from RSM-TCL and RSM-θ^2-ε_θ - TCL.
Figure 6.42 – rms velocity fluctuation distributions within a $60^\circ$ stable inclined tall cavity resulting from RSM-Basic and RSM-$\theta^2-\varepsilon_\theta$ - Basic.

Figure 6.43 – rms velocity fluctuation distributions within a $60^\circ$ stable inclined tall cavity resulting from RSM-TCL and RSM-$\theta^2-\varepsilon_\theta$ - TCL.
In Figures 6.44-6.46, the rms temperature fluctuations in the vertical and inclined stable $60^\circ$ tall cavities predicted by the $k\varepsilon-\theta^2\varepsilon_\theta$ and Reynolds stress model when solving $\theta^2$ and the more elaborate algebraic equations for turbulent heat fluxes are compared with experimental measurements. In the vertical cavity, Figure 6.44, employing the Reynolds stress model improves significantly the temperature fluctuations prediction, compared to those of the $k\varepsilon-\theta^2\varepsilon_\theta$ scheme. In the Reynolds stress model, separate transport equations are solved for each component of the Reynolds stress tensor, and these then feed into the algebraic equations for the turbulent heat fluxes, which in turn feed into the production term of the temperature variance transport equation. In the $k\varepsilon$ model, on the other hand, the stress field is approximated by the simple eddy viscosity model. It is not unreasonable, therefore, that the Reynolds stress model should produce the more accurate predictions of the temperature fluctuations. In the stable $60^\circ$ inclined tall cavity test case, switching from an EVM to the Basic Reynolds stress model, again resulted in improvements to the temperature fluctuation predictions. This is not, however, the case for the TCL Reynolds stress model, which predicted almost the same levels as the EVM. This can be explained by the fact that the RSM-TCL appears to over-predict turbulence levels in the stable $60^\circ$ inclined cavity compared to the EVM. It is worth to note that even the RSM-basic predictions, which are the best for the vertical cavity, are not as close to the experimental data for the $60^\circ$ stable case. The experimental data also show that temperature variance levels are reduced significantly as the cavity is tilted from the vertical to the $60^\circ$ stable, whereas the computations do not show any such reduction. It thus appears that it is necessary to first improve the RSM-Basic and TCL predictions of the turbulence kinetic energy in order to improve the temperature fluctuations as well.

In the case of the $k\varepsilon$, the introduction of the more elaborate turbulent heat flux model, raises the predicted local Nusselt number to levels higher than those measured (Figure 6.47). This suggests that the more elaborate expressions for the turbulent heat fluxes, do not reproduce realistic heat flux levels when they have to rely on the effective viscosity approximation for the Reynolds stresses. By contrast when used with the second-moment closure,
this more elaborate thermal model, has little effect on the predicted local Nusselt number, which is already in close agreement with the measurements.
Figure 6.44 – Temperature fluctuations within vertical cavity using RSM-TCL(AWF) and 4-equations EVM.
Figure 6.45 – Temperature fluctuations within 60° stable inclined cavity using RSM-basic(AWF) and 4-equations EVM.
Figure 6.46 – Temperature fluctuations within 60° stable inclined cavity using RSM-TCL(AWF) and 4-equations EVM.
6.5 Inclined tall cavity - 60° unstable

6.5.1 Numerical predictions

Figure 6.48 shows mean temperature distributions along four sections from the bottom to the top of the cavity. These figures present the numerical predictions by $k-\varepsilon$ using AWF, SWF and LRN. They are compared with the experimental measurements. The experimental data shows that there are sharp temperature gradients near the hot and cold walls and only a modest temperature change across the cavity core. This suggests increased turbulence levels within the cavity core. The high turbulence mixing causes uniform temperature at the cavity’s core. The comparisons of numerical predictions and experimental measurements show that the $k-\varepsilon$ using AWF, SWF and LRN overall manages to reproduce the measured variation of the mean temperature, although $k-\varepsilon$/AWF shows small discrepancies near the top and bottom.
of the cavity. Figure 6.49 presents the comparisons of mean temperature predictions by RSM-Basic using AWF with the experimental measurements. The comparisons show that employment of RSM-Basic using AWF improves mean temperature predictions near the top and bottom of the cavity compared with $k$-$\varepsilon$-AWF results. In the middle of the cavity on the other hand, the mean temperature profiles predicted by the RSM/Basic, are not as close to the measured profiles as those of the EVM models, generally returning a steeper temperature variation across the core. This suggests that the RSM-Basic using AWF produces extra turbulence near the hot and cold walls near the mid-height of the cavity. As can be seen in the comparisons of Figure 6.50, introduction of the RSM/TCL model leads to stronger discrepancies between the predicted and measured profiles of the mean temperature.

The modelling of the turbulent heat fluxes for the $60^\circ$ unstable inclined cavity, is further explored through the introduction first of the $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$-AWF model with the resulting comparisons presented in Figures 6.51 to 6.52. The comparisons show that solution of extra transport equations of $\theta^2$ and $\varepsilon_\theta$ and use of more elaborate algebraic equation for turbulent heat fluxes can improve the mean temperature prediction near the top and bottom of the cavity compared with $k$-$\varepsilon$-AWF results. As shown in Figure 6.52 on the other hand, the $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$-AWF model over-estimates the levels of temperature variance over the entire cavity. The modelling of the turbulent heat fluxes for the $60^\circ$ unstable inclined cavity, is finally further explored through the introduction of the RSM-$\theta^2$-$\varepsilon_\theta$-AWF model with the corresponding comparisons presented in Figures 6.53 to 6.54. The comparisons show that solution of extra transport equations of $\theta^2$ and $\varepsilon_\theta$ and the use of more elaborate algebraic equations for the turbulent heat fluxes does not improve mean temperature predictions compared with the RSM-Basic using the AWF. However, the prediction of the temperature variance is improved compared with that of $k$-$\varepsilon$-$\theta^2$-$\varepsilon_\theta$-AWF, near the mid-height of the cavity. Care has to be taken for comparisons of normal and shear stresses between EVM and RSM due to the fact that EVM predicts a single cell circulation within the cavity whilst RSM predicts multiple cells as it is shown in the Figure 6.55. Therefore the sections at which comparisons are made, might be located at the high or low turbulence regions of the cells in
the RSM predictions. The reason for the difference between EVM and RSM results may be that EVM tends to produce too high values for turbulent viscosity which prevent the flow breaks down to smaller cells while it is not the case for RSM.

Figure 6.56 shows comparisons between the local Nusselt number distributions predicted by the $k$-$\varepsilon$-AWF and the RSM- $\theta^2$-$\varepsilon$-$\theta$-Basic and the measured distribution of Cooper et al. [8]. As seen earlier, the $k$-$\varepsilon$-$\theta^2$-$\varepsilon$-$\theta$ and the RSM erroneously return a multi-cellular structure, and consequently a local Nusselt number variation which is far removed from the measured behaviour. It is therefore pointless to include them in any detailed comparisons with experimental data. Both the $k$-$\varepsilon$-AWF and the RSM-$\theta^2$-$\varepsilon$-$\theta$-Basic models return similar distributions along the cold wall, with low Nu levels at the bottom corner which increase monotonically towards the top corner. This is consistent with the presence of single cell structure and also in close agreement with the data.
Figure 6.48 – Mean temperature distributions within a $60^\circ$ unstable inclined tall cavity resulting by $k-\varepsilon$ using LRN, AWF and SWF.

Figure 6.49 – Mean temperature distributions within a $60^\circ$ unstable inclined tall cavity resulting by RSM-Basic.
**Figure 6.50** – Mean temperature distributions within a $60^\circ$ unstable inclined tall cavity resulting by RSM-TCL.

**Figure 6.51** – Mean temperature distributions within a $60^\circ$ unstable inclined tall cavity resulting by $k-\varepsilon\theta^2-\varepsilon_\theta$-AWF.
**Figure 6.52** – Temperature variance distributions within a 60° unstable inclined tall cavity resulting by $k$-$\varepsilon-\theta^2$-$\varepsilon_\theta$-AWF.

**Figure 6.53** – Mean temperature distributions within a 60° unstable inclined tall cavity resulting by RSM-$\theta^2$-$\varepsilon_\theta$-Basic.
Figure 6.54 – Temperature variance distributions within a $60^\circ$ unstable inclined tall cavity resulting by RSM-$\theta^2-\theta$-Basic.

Figure 6.55 – Stream lines in tall cavity resulting from RSM-Basic, (scale of the small side is enlarged by a factor of 7).
6.6 Inclined tall cavity - $5^\circ$ stable

6.6.1 Test case description

The second test case is that where the tall walls of the cavity are inclined at an angle of $5^\circ$ stable to the horizontal. The tall walls are again the hot and cold walls and the short walls are insulated. In this case LES data are available for a Rayleigh number value $Ra = 4.16 \times 10^8$ [41]. In the LES simulation, repeating flow conditions are imposed in the third directions, which means that in the LES case, and in contrast to the conditions of the experimental study, the cavity has an infinite aspect ratio in the spanwise direction. This is likely to prevent the development of large-scale three-dimensional structures. The hot and cold wall temperatures are $19^\circ C$ and $17^\circ C$ respectively. The fluid inside the tall cavity is air. The height of the cavity is 28.7m and its width is 1.0m. In the LES computation, the gravity acceleration is numerically set to 19.38 to create the above mentioned Ra number. The Ra number could be increased by increasing the temperature difference. Addad et al. [41] argue that increment of temperature difference leads to a large Prandtl number variation, whilst Prandtl number was meant to be constant in the LES calculations. This could cause error in the simulation results. That is why gravity was modified to provide the Ra number. To be fully consistent with the LES data, the same approach is adopted in the RANS computations. Comparisons have been made along 5 different cross sections, which are shown in Figure 6.57 as dashed lines. Nusselt numbers along the hot wall are also compared with available LES data.

Figure 6.56 – Nusselt distribution along the cold wall of $60^\circ$ unstable inclined tall cavity resulting from $k-\varepsilon$-AWF and RSM-Basic.
The local Nusselt numbers are calculated as \[ Nu = -L \left( \frac{\partial \Theta}{\partial x_1} \right)_{w} / (\Theta_h - \Theta_c) \], where \( L \) is distance between the hot and the cold walls.

![Figure 6.57 – Schematic of 5° stable inclined tall cavity and its thermal boundary condition.](image)

### 6.6.2 Flow pattern

The flow pattern of air inside the 5° stable inclined tall cavity predicted by the \( k-\varepsilon \) using LRN, AWF and SWF is shown in Figures 6.58-6.63. In this test case, because of the temperature difference between the tall walls, 3 circulation cells are created. One circulation cell covers most of the cavity apart from the top right corner and the bottom left corner regions, in which the fluid is almost stagnant. Within this primary circulation cell, for some cases, there are two smaller counter rotating circulation cells near the centre of the cavity. The predicted distance between these two smaller circulation cells, from different turbulence models varies. The temperature contours reveal that within the top left corner and the bottom right corner heat is transferred almost in conduction mode only, because of almost stagnant fluid and weak convection in those regions. It is worth to note that the uniform temperature near the two ends of the cavity is due to convection which moves the hot flow toward the top and the cold flow toward the bottom. In the case of high turbulence flow, due to high mixing, the temperature field tends to be uniform and equal to the average temperature. In this test case, low-turbulent flow exists near the hot and the cold walls where circulation is present.
Figure 6.58 – Stream traces and vector plot resulting from Low-Re-Number $k-\varepsilon$, (scale of the small side is enlarged by a factor of 7).

Figure 6.59 – Temperature contours resulting from Low-Re-Number $k-\varepsilon$, (scale of the small side is enlarged by a factor of 7).

Figure 6.60 – Stream traces and vector plot resulting from $k-\varepsilon$ - AWF, (scale of the small side is enlarged by a factor of 7).
Figure 6.61 – temperature contours resulting from $k$-$\varepsilon$ - AWF, (scale of the small side is enlarged by a factor of 7).

Figure 6.62 – Stream traces and vector plot resulting from $k$-$\varepsilon$ - SWF, (scale of the small side is enlarged by a factor of 7).

Figure 6.63 – temperature contours resulting from $k$-$\varepsilon$ - SWF, (scale of the small side is enlarged by a factor of 7).

6.6.3 Comparisons of $k$-$\varepsilon$ using LRN, AWF and SWF with LES data within the inclined $5^\circ$ stable tall cavity

In Figures 6.64-6.65, the velocity parallel to the tall walls and temperature within the inclined $5^\circ$ stable tall cavity predicted by the $k$-$\varepsilon$ using LRN, AWF
and SWF treatments is compared to LES data at 5 different sections along cavity. Because the flow in the test case is low turbulence, to ensure that the effect of very low turbulence is considered it was decided to include the additional low-Reynolds-number damping functions $f_\mu, f_1$ and $f_2$ (section 3.3.3) everywhere, except in the near-wall row of cells where wall functions are applied, for the $k$-$\varepsilon$ using AWF and SWF treatments. Although, the results showed that there was slight difference in the results with or without the modification. These comparisons show that the $k$-$\varepsilon$ model, the LRN version and also the high-Re form with either AWF or SWF, predict the correct variation of the mean velocity and temperature in this test case apart from a slight velocity profile difference at $Y=0.5$ between the RANS and LES. This discrepancy arises from the prediction of the positions of the two small counter-rotating circulation cells at the mid-height of the cavity. The LES solution shows them further from each other and nearer to the tall walls than do the RANS schemes.

In Figure 6.66, the variation of the turbulent kinetic energy predicted by the $k$-$\varepsilon$ using LRN, AWF and SWF is compared with that of the LES data at 5 different sections along the cavity. The comparisons show that apart from at the mid-height ($Y=0.5$), the RANS predictions do not match well with the LES data. Considering the fact that the RANS and LES data match quite well for the mean velocity and temperature, it would be reasonable to conclude that because turbulence levels are generally low, the turbulence field exerts only a weak influence of the overall flow and thermal developments.

In Figure 6.67, the $k$-$\varepsilon$ using LRN, AWF and SWF can be seen to predicts Nusselt numbers matching very well with the LES data. For the RANS turbulence models, Nusselt prediction is a challenging point for turbulent flows. As with the mean flow and thermal comparisons, existence of weak turbulence or almost laminar flow can be an explanation for the almost spot-on Nusselt number prediction for this test case.
Inclined Cavity-2D simulations

Figure 6.64 – Mean velocity profiles within 5° stable tilted tall cavity.
Figure 6.65 – Temperature distributions within 5° stable tilted tall cavity.
Figure 6.66 – Turbulent kinetic energy distributions within $5^\circ$ stable tilted tall cavity.
6.7 Closing remarks

In this Chapter, three different test cases have been numerically investigated regarding turbulent natural convection. The test cases were selected so that they cover various physical phenomena such as single and multiple cell circulation and stable and unstable stratified flow. The comparisons in this Chapter showed that the $k-\varepsilon$ model using the AWF near-wall strategy produces reliably accurate predictions for the $5^\circ$ stable, $60^\circ$ stable and $60^\circ$ unstable. Introduction of the RSM models over-estimates turbulent kinetic energy near the top and the bottom of tall cavity in the $60^\circ$ stable inclined case. The over-prediction is significant in the case of the RSM-TCL using AWF. The introduction of the $\overline{\theta^2}$ and $\varepsilon_\theta$ transport equations, with more complex algebraic thermal flux models improves the RSM-basic and TCL predictions near the top and the bottom of the tall cavity. The improvement is more significant in RSM-$\overline{\theta^2}$-$\varepsilon_\theta$-TCL case. In the $60^\circ$ stable case these extended stress transport models then produce better predictions than the corresponding eddy viscosity schemes.
Chapter 7

Inclined Cavity-3D time-dependent simulations

From the experimental study of the flow in the $15^\circ$ stable and $15^\circ$ unstable inclined tall cavities, it was concluded it is necessary to investigate these test cases as three-dimensional and unsteady. The experimental study of these two test cases reported existence of three dimensional flow structures. Therefore it was necessary to carry out 3D numerical simulation of these two test cases. Moreover, the experimental study reported that the flow in the $15^\circ$ unstable case is time-dependent. In addition, 3D steady-state computations of the $15^\circ$ unstable case did not fully converge. Therefore, 3D time-dependent computations were adopted to study these cases. In Section 7.1, validation of the second in-house code employed in the research is presented. This code is called STREAM and is capable of solving both 2D and 3D simulations in contrast to the in-house code TEAM, used to produce the computations presented in the previous chapters which is only developed for 2D simulations. Up to this point only the TEAM code was used for 2D simulations of various test cases. Then the buoyancy-force-related terms in the transport equations and analytical wall functions were introduced to the STREAM code. The vertical and $15^\circ$ unstable inclined tall cavities have been two-dimensionally simulated using the STREAM code and the results have been compared with the corresponding simulations of the TEAM code to ensure that the buoyancy terms and the AWF extensions have been correctly introduced.
After validation of the STREAM solver, the 3D time-dependent simulation of 15° unstable inclined tall cavity has been carried out. The result of this numerical simulation is presented in section 7.2. In section 7.3, the 3D numerical simulation of 15° stable degree test case is compared with the experimental data. Finally, in the section 7.4, the closing remarks are presented.

7.1 Validation of STREAM

The first step in utilizing the STREAM code was to introduce buoyancy related terms in the transport equations and analytical wall functions to the code. The modifications made to the STREAM code have been discussed and explained in Chapter 4. After those terms were added it was necessary to validate the modified code to make sure those modifications are correct. The STREAM code is capable of 2D and 3D simulations. Therefore two test cases vertical and 15° unstable inclined cavities are chosen to be two-dimensionally simulated by STREAM and then been compared with the already available simulations of the same test cases with the TEAM code. It is worth to note that all parameters in both calculations are set the same. They are steady state calculations and exactly the same grids have been employed for both calculations. The grids for analytical wall functions near wall treatment are 20x80 for the vertical tall cavity and 40x220 for the 15° unstable inclined tall cavity. The grids used for Low-Reynolds-Number near wall treatment are 40x120 for the vertical cavity and 100x270 for the 15° unstable inclined tall cavity.

The Figures 7.1-7.9 show the comparisons between the results of the STREAM and the TEAM codes using both AWF and LRN near wall treatments. Based on these comparisons it can be concluded that the STREAM code predicts essentially identical solutions to the TEAM code. The comparisons show that the two codes (TEAM and STREAM) produce identical predictions when the same grids and turbulence models are used. The implementation of the AWF in STREAM is done in a general wall-oriented reference frame, and is the above comparisons give confidence that the AWF formulation is consistent with that coded in the TEAM code.
Figure 7.1 – Temperature distributions comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes (k-ε-AWF).
Figure 7.2 – Mean velocity distributions comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes (\(k-\varepsilon\)-AWF).
Figure 7.3 – rms velocity fluctuations comparisons of results from 2D simulation of vertical cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).
Figure 7.4 – Temperature distributions comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).
Figure 7.5 – Mean velocity distributions comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).
Figure 7.6 – rms velocity fluctuations comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-AWF).
Figure 7.7 – Temperature distributions comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-LRN).
Figure 7.8 – Mean velocity distributions comparisons of results from 2D simulation of 15° unstable tilted cavity with TEAM and STREAM codes ($k$-$\varepsilon$-LRN).
7.2 3D time dependent simulation of inclined tall cavity - 15° unstable

7.2.1 Test case description

This test case, shown in Figure 7.10 is that of the cavity where the tall walls are now inclined at 15° to the horizontal so that the hot wall is located below the cold wall. Experimental data are available for a Rayleigh number Ra = 0.8 × 10^6 (Cooper et al. [8]). The hot and cold wall temperatures are 34°C and 16°C respectively. Apart from the hot and cold walls, adiabatic condition is imposed to the rest of the walls. All the fluid inside the tall cavity is air. As also noted in previous chapters, the height of the cavity, H, is 2.18m and
the distance between the hot and cold walls, L, is 0.076m and the width of the cavity, W, is 0.52m, which results in a spanwise direction, W/L, is 6.84. Comparisons with experimental measurements have been made along several cross sections in the x-y plane at Z=0.25, Z=0.5 and Z=0.75 and also in the y-z plane at X=0.5, as defined in Figure 7.10.

![Figure 7.10 – Inclined cavity with angle 15° unstable.](image)

### 7.2.2 Flow Pattern

In Figure 7.11, the iso-temperature surfaces predicted by the 3D time dependent $k$-$\varepsilon$ model, using AWF at $\tau^* = 440$ are shown (where $\tau^* = \tau \frac{V_0}{L}$ and $V_0 = \sqrt{g\beta \Delta \Theta L}$ and $\tau$ is time in second). The selected time ($\tau^* = 440$) is only a snapshot of the results. This picture shows that the time dependent $k$-$\varepsilon$ using AWF predicts a highly 3D structure within the inclined 15° unstable cavity. The iso-temperature surfaces show that plumes of air separating from the hot and cold walls at irregular locations which change with time as well.
Figure 7.11 – Temperature iso-surfaces in tall 15° unstable tilted cavity resulting from $k$-$\varepsilon$ using AWF at $\tau^* = 440$ ($Ra = 0.8 \times 10^6$).

7.2.3 Grid for analytical wall functions

Figures 7.12-7.13 show the grids which have been used for the numerical simulations. The first one is the preliminary grid 40x220x50 (440000 nodes). The numerical domain is split to 4 blocks and parallel processing (mpi) is employed to speed up the calculation. The second grid consists of a finer mesh of 30x300x80 (720000 nodes). This grid is split to 8 blocks using parallel processing. The resulting computations show that both grids produce similar results.
Figure 7.12 – Grid within (a) x-y (b) x-z (c) y-z planes of 15° unstable inclined tall cavity resulting (coarse mesh).
Figure 7.13 – Grid within (a) x-y (b) x-z (c) y-z planes of 15° unstable inclined tall cavity resulting (fine mesh).
7.2.4 3D time dependent simulation by $k-\varepsilon$-AWF

Figures 7.14-7.19 show the solution of 3D time dependent simulation of the stable $15^\circ$ tilted cavity by $k-\varepsilon$-AWF. Interestingly, attempts to compute this case as 3D steady, using the same code, failed to produce a fully converged numerical solution, probably because of the inherent unsteadiness of this flow.

In the time-dependent calculations, the normalized time step is set equal to 0.055, normalized by $\frac{L}{V_0}$ (where $L$ is the distance between the hot and cold walls, $V_0 = \sqrt{g\beta\Delta\Theta L}$) and the first order implicit scheme is employed for time discretisation. The convergence criterion at each time step was set to $10^{-3}$ for all normalised residuals. The calculations have been carried out for 30000 time steps which means the normalized total time is 1650. The computed flow shows no signs of settling to a steady state. The solution is compared with the experimental measurements of hydraulic and thermal fields. The temperature contours and vector plots are shown at different times and different planes. The experimental study has produced mainly time-averaged flow and thermal data. The numerical results shown in this section present the temporal evolution of the instantaneous flow and thermal fields. Comparisons between the time-averaged computed and measured flow and thermal parameters are presented in the section that follows.

In Figures 7.14-7.15, the temperature contours and vector plots in the x-y plane, which is the longitudinal plane between the two thermally active sides, are shown. The contour plots show that there are multiple circulation cells in this plane which change with time. The vector plots show that these circulations are weak at this plane and this is consistent with finding from the experimental study. These contour plots (Figures 7.14-7.15) also show that several plumes separate from the the hot and the cold walls where $x=0$ is the hot wall and $x=0.076$ is the cold wall. The number of these plumes changes with time and also the location of these plumes changes as well. These changes take place irregularly with time during the period of time that the numerical simulation has been carried out.

Figures 7.16-7.17, show the temperature contours and vector plots in the x-z
plane when Y=0.5, which is the spanwise plane at mid-height. The turbulence model has captured the longitudinal vortices while their sizes and numbers change with time. The experimental study has concluded there are 4 longitudinal vortices, which when time-averaged are continuous all the way along the y direction whereas in the numerical simulation, these vortices are not continuous. As it is mentioned before, the experimental study has measured the time average of the hydraulic and thermal field variables. Therefore it is necessary to produce the numerical time average of the variables and see if the numerical simulation can predict the four longitudinal vortices.

Figures 7.18-7.19 show temperature contours plot in the y-z plane, which is the plane parallel to the thermally active sides. It can be seen that irregular separation of plumes from the hot and cold walls gives rise to an irregular temperature distribution in this plane. It is worth noting that the locations of these plumes change with time irregularly.
Figure 7.14 – Temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF ($Z=0.5$).
Figure 7.15 – Vector plots within x-y plane of $15^\circ$ unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF ($Z=0.5$).
Figure 7.16 – Temperature contours within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF ($Y=0.5$).
Figure 7.17 – Vector plots within x-z plane of $15^\circ$ unstable inclined tall cavity resulting from 3-D time dependent $k-\varepsilon$-AWF ($Y=0.5$).
Figure 7.18 – Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k-\varepsilon$-AWF (X=0.5).
7.2.5 Time-averaged data by $k-\varepsilon$-AWF

In this section the time-averaged numerical data produced using the $k-\varepsilon$-AWF are compared with the corresponding experimental data. As it mentioned in the previous section, the experimental data are time-averaged data. Therefore it is necessary to produce numerical time-averaged data to have consistent comparisons. In Figure 7.20, the time-averaged mean temperatures within the mid-longitudinal plane, $Z=0.5$ are compared with the corresponding experimental data at several traverse locations from the bottom to top of the cavity. The comparisons show very close agreement between time-averaged numerical and experimental data. They generally show steep temperature gradients along the thermally active walls and practically isothermal conditions in the core, where the fluid temperature is close to the average of the hot and cold side temperatures. This suggests the presence of strong mixing throughout the cavity. In Figure 7.21, time-averaged velocities parallel to the thermally active walls, along the longitudinal, $y$, direction, in the mid-longitudinal plane ($x$-$y$ plane at $Z=0.5$) are compared with the experimental
data. The comparisons show that the 3D time-dependent computation returns the correct low magnitude of the wall-parallel component of the velocity. Agreement with the experimental data is not complete, but given the very low magnitude of the measured mean velocities in this case, the fact that the correct magnitude is being predicted, is far more important than any discrepancies. In Figure 7.22, the time-averaged component of the velocity normal to the thermally active sides, x direction, within the mid-longitudinal plane, Z=0.5 are compared with the corresponding experimental data. Again the comparisons show that the 3D time-dependent computations, based on the $k$-$\varepsilon$-AWF, with some minor discrepancies, successfully predict the very low levels of wall normal velocity within the mid-longitudinal plane.

Figures 7.23-7.24 present the time-averaged velocity fluctuations along the wall-parallel and wall normal directions, within the mid-longitudinal plane, Z=0.5. Basically, the time-averaged total velocity fluctuations consists of resolved turbulence $\overline{u^2}$ and modeled turbulence $2/3\overline{k}$. The comparisons show that the time-averaged velocity fluctuations resulting from the $k$-$\varepsilon$-AWF are higher than those of the experimental study while the instantaneous numerical data is in agreement with the experimental data. Time-dependent numerical simulation of this test case showed that the flow within the cavity is strongly time-dependent. That is why the time-averaged total velocity fluctuations calculated from the time-dependent simulation is higher than that of the instantaneous data which only includes only modelled portion of velocity fluctuations (Figures 7.25-7.26). The over-prediction of the velocity fluctuations compared with the experimental data may be due to the modeled part of the time-averaged velocity fluctuations. It is expected to improve the numerical results by employment of Differential Stress Model (RSM) which solves the transport equations of stresses instead of implementing Eddy Viscosity Model (EVM).

Figures 7.27-7.28 show the time-averaged mean temperatures in x-y plane where Z=0.25 and 0.75 compared with the experimental measurements at the same sections. In common with the corresponding comparisons at the mid-plane (Z=0.5), $k$-$\varepsilon$-AWF results agree well with the experimental data. As
shown in the time-averaged temperature profiles along the centreline, conditions are practically isothermal at the core with steep gradients across the near-wall regions. This is again a consequence of the mixing caused by the three-dimensional, unsteady flow structures.

In Figure 7.29, the profiles of the longitudinal, y direction, component of the time-averaged mean velocity along traverses in the spanwise, z, direction and half-way between the two thermally active sides, X=0.5, are compared with corresponding experimental data. The experimental data, which extend to just over half the spanwise distance, show a nearly sinusoidal variation, with two negative peaks, one near the wall and one in the middle of the cavity and a positive peak in between. Cooper et al. [8] argues that, assuming symmetric behaviour in the spanwise direction, this suggests the presence of four longitudinal recirculation cells within the plane parallel to the thermally active sides, which extend over the entire length of the cavity. The predicted profiles, with the exception of the region near the lower end wall, Y=0.1, are reasonably close to the data. The $k$-$\varepsilon$-AWF URANS, thus is able to return organised longitudinal re-circulation cells within the plane parallel to the thermally active sides, but not necessarily four, which is what is suggested by the data. Figure 7.30, compares the profiles of the component of the time-averaged velocity normal to the thermally active planes, x direction, along traverses in the spanwise, z, direction and half-way between the two thermally active sides, X=0.5. The experimental data show that the wall-normal, $U$, component is stronger than the wall-parallel, $V$, component, plotted in Figure 7.29. In common with the wall-parallel component, the wall normal also exhibits a nearly sinusoidal variation in the spanwise direction, with negative peaks near the side wall and the centre of the cavity and a positive peak in between. Once again the experimental spanwise traverses do not extent over the entire width of the cavity, but assuming that the two halve are symmetric, Esteifi suggests that these data indicate the presence of four vortices which extend over the entire length of the inclined cavity. At the middle of the cavity, Y=0.5, the spanwise profile returned by the $k$-$\varepsilon$-AWF, is close to the available data and also returns a symmetric distribution in the spanwise direction. The three dimensional URANS with the $k$-$\varepsilon$-AWF is thus able to reproduce the most dominant feature of the
flow at this angle of inclination. This is what led to the improvements in the predictions of the mean flow and thermal fields, identified earlier. The predicted profiles near the two end walls show that, in contrast to the measured behaviour, the vertical motion is reversed.

Figures 7.31 and 7.32 present comparisons of the fluctuating velocity components in the wall-parallel and wall-normal directions, along the same spanwise traverses as those of Figures 7.29 and 7.30. As mentioned earlier, the numerical time-averaged velocity fluctuations consist of resolved and modeled parts. The $k-\varepsilon$-AWF URANS predictions return rms values of the same order as those of the measurements though predicted levels are generally higher by about 20 percent. With the exception of the near-wall regions, the predicted distribution is reasonably uniform, a feature which is consistent with the mixing of the flow by the three-dimensional structures and which is also in accord with the variation of the measured values. The rapid increase in the predicted near-wall levels, is most likely related to the use of a high-Re model. In Figure 7.33, the time-averaged longitudinal distributions of the Nusselt number at four spanwise locations, $Z=0.5, 0.625, 0.75, 0.865$, predicted by the $k-\varepsilon$-AWF, are compared to the measured data. There is close agreement between the predictions and the measurements, both in terms of the levels and the distributions of the Nusselt number in the longitudinal and spanwise directions. This close agreement suggests that at this angle of inclination, it is essential to predict the three-dimensional flow structures present, in order to return the correct thermal behaviour. The main predictive deficiency appears to be an over-estimation of the Nusselt number levels along the centreline, $Z=0.5$.

Figures 7.34 to 7.36, show temperature contour and vector plots, within spanwise planes normal to the thermally active planes, $x-z$, and also within planes parallel to the thermally active planes, $y-z$. These plots provide a clearer picture of the flow structures presented in this section. Figure 7.34-7.35 present the velocity vector plot and temperature contours in spanwise planes normal to the thermally active walls, $x-z$, at three longitudinal locations $Y=0.1, 0.5$ and 0.9. They show there are four longitudinal vortices all the way through from the bottom to top of the cavity. The size and strength of the vortices change
at different sections. Figure 7.36 shows the velocity vector plot and temperature contours in the plane parallel to the thermally active planes, x-y, halfway between them, at Z=0.5. While the flow in this plane is weak, nevertheless, longitudinal flow structures can still be detected. Conditions are isothermal over most of the plane with the exception of the end wall regions, where there is a rise in temperature near the top end wall and a corresponding reduction near the bottom end wall.
Figure 7.20 – Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.21 – 3D-Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.22 – Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.23 – Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.24 – Time average rms velocity fluctuations resulting from $k-\varepsilon$-AWF, $Z=0.5$. 
Inclined Cavity-3D time-dependent simulations

Figure 7.25 – rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.5$, $\tau^* = 550$. 
Figure 7.26 – rms velocity fluctuations resulting from $k-\varepsilon$-AWF, $Z=0.5$, $\tau^* = 550$. 
Figure 7.27 – Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. 
Figure 7.28 – Time average temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. 
Inclined Cavity-3D time-dependent simulations

Figure 7.29 – Time average mean velocity distributions resulting from \( k-\varepsilon \)-AWF, \( X=0.5 \).
Figure 7.30 – Time average mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $X=0.5$. 
Figure 7.31 – Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. 
Figure 7.32 – Time average rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. 
Figure 7.33 – Nusselt distributions resulting from $k$-$\varepsilon$-AWF.
Figure 7.34 – Time average vector plots within x-z plane of $15^\circ$ unstable inclined tall cavity at different Y sections resulting from 3-D time dependent $k-\varepsilon$-AWF ($Y=0.5$).
Figure 7.35 – Time average temperature contours within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent $k-\varepsilon$-AWF (Y=0.5).
Figure 7.36 – Time average vector plots and temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent $k$-$\varepsilon$-AWF (X=0.5).
7.2.6 FFT analysis

In the Figures 7.37-7.42, the dominant frequencies of resolved mean temperature resulting from $k$-$\varepsilon$-AWF are shown at several locations. They are also compared with the frequencies resulting from the experimental measurements. The comparisons show that the URANS $k$-$\varepsilon$-AWF predictions are able to reproduce the range of dominant frequencies and their power densities. Moreover, the variations in the measured spectral distributions with physical location are also well reproduced by the computations. This suggests that the URANS computations are able to resolve the large-scale fluctuations related to the large-scale three-dimensional flow structures.
Figure 7.37 – Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.38 – Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.39 – Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. 
Figure 7.40 – Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.25$. 
Figure 7.41 – Power spectral diagram for temperature resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. 
In this section the velocity vector plots and temperature contour plots resulting from 3D time-dependent RSM computations are presented within different planes and at different times. In the diagrams, $\tau^*$ denotes the normalized time which is defined as $\tau^* = \tau V_0 / L$ where $V_0 = \sqrt{g\beta \Delta \Theta L}$ is buoyant velocity. Figures 7.43-7.44 show the temperature contours and vector plots in
the central, \( z=0.5 \), longitudinal, \( x-y \), plane at various instances. The diagrams clearly show the time dependency of the buoyant flow inside the \( 15^\circ \) unstable inclined cavity. The flow pattern in the \( x-y \) plane shows a flow structure that plumes of hot air separate from the hot wall at the bottom and move upward and plumes of cold air separate from the cold wall and move downward. The location of the plumes occurs randomly and is highly time dependent. In the spanwise \( x-z \) planes, Figures 7.45-7.46, the cross sections of longitudinal vortices can be seen. As mentioned earlier, the experimental study suggests that there are four longitudinal vortices over the entire length of this inclined cavity. The instantaneous flow predictions of Figures 7.45 and 7.46, show that the RSM URANS predictions, reproduce these longitudinal vortices. Based on the numerical results, the sizes and strength of these vortices vary with time. The number of the vortices also changes with time even though they are mostly three apart from an instance when they become four vortices. These vortices convect thermal energy between the hot and cold walls.

The contours of the instantaneous temperature field within the mid-plane parallel to the thermally active sides, \( y-z \), of Figure 7.47, provide a picture of the flow patterns within this cross-section. They suggest the presence of meandering and unstable longitudinal flow structures. In the next section, these instantaneous flow predictions are supplemented with time-averaged computations, which allow for a direct comparison with experimental data.
Figure 7.43 – Temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5).
Figure 7.44 – Vector plots within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5).
Figure 7.45 – Temperature contours within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Y=0.5).
Figure 7.46 – Vector plots within x-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Y=0.5).
Figure 7.47 – Temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent numerical simulations (X=0.5).
7.2.8 Time-averaged data by RSM

In the previous section, 3D time-dependent simulation of 15° unstable inclined cavity by $k$-$\varepsilon$-AWF was presented. Although there is reasonable agreement with the data, some differences still exist between experimental and numerical solutions. Therefore it is informative to employ more elaborate turbulence modelling approach, to establish if it is possible to obtain further improvements in the numerical predictions. In this section, Basic version of the RSM with the Analytical wall function is implemented to simulate the 15° unstable inclined cavity and its results are compared with the experimental data.

Figures 7.48-7.50 show profiles of the mean temperature along traverse lines normal to the thermally active planes, within three longitudinal, x-y planes, the mid-plane, Z=0.5 and also two planes closer to the side walls, 0.25 and 0.75. The comparisons show close agreement between the experimental and numerical data. As noted earlier, the experimental data indicate that the cavity core is practically isothermal, due to the enhanced mixing. Figures 7.51 and 7.52 present profiles of the longitudinal and wall-normal velocities, V and
U, along a number of traverse lines normal to the thermally active planes, all within the central longitudinal plane, x-y plane at Z=0.5, and Figures 7.53 and 7.54 present profiles of the same velocity components along spanwise traverse lines, within the mid-plane parallel to the thermally active plane, y-z plane at X=0.5. The velocity profiles within the x-y plane resulting from RSM-Basic are similar to those which result from the use of the $k$-$\varepsilon$-AWF. This is expected because the flow inside the x-y plane is weak where Z=0.5 and both turbulence models produce almost the same velocity profiles. The velocity comparisons within the y-z plane now show some improvements compared with the numerical data from $k$-$\varepsilon$-AWF. As it was shown in the previous section, $k$-$\varepsilon$-AWF predicts 4 longitudinal vortices the sizes and strength of which vary from the bottom to top of the cavity. The RSM on the other hand produces 3 longitudinal vortices the sizes and strength of which is almost constant from the bottom to top of the cavity, a feature that is closer to experimental findings. The velocity comparisons along the spanwise, z, direction now show better agreement with the experimental data. Figures 7.55 and 7.56 present profiles of the normal stresses in the longitudinal and wall-normal direction, along a number of traverse lines normal to the thermally active planes, all within the central longitudinal plane, x-y plane at Z=0.5, and Figures 7.57 and 7.58 present profiles of the same normal stresses along spanwise traverse lines, within the mid-plane parallel to the thermally active plane, y-z plane at X=0.5. The Basic RSM predicts normal turbulent stresses which are in closer agreement with the measurements, than the corresponding $k$-$\varepsilon$-AWF profiles, especially for the spanwise traverses. This is not unexpected, since the RSM represents exactly physical processes like the production rate and the convection of each stress component, and it approximate the effects of others, like the dissipation rate and the redistribution of turbulence energy among the three directions. In Figure 7.59, the time-averaged longitudinal distributions of the Nusselt number at four spanwise locations, Z=0.5, 0.625, 0.75, 0.865, predicted by the RSM-AWF, are compared to the measured data. The comparisons show that the RSM predictions, like the $k$-$\varepsilon$ ones, are in close agreement with most of the available data, even at the central, Y=0.5, location, but unlike the $k$-$\varepsilon$ profiles, they show a strong fluctuating variation in the longitudinal direction, which is consistent with the presence of a strong multi-cell flow structure. The available data are
not sufficiently detailed to provide confirmation for this feature, but they cer-
tainly confirm that the RSM model returns the correct Nusselt number levels at the locations at which measurements are available.

Figures 7.60-7.61 show vector plots and mean temperature contours within spanwise, x-z, planes at longitudinal locations Y=0.1, 0.5 and 0.9. Both sets of plots indicate the presence of three longitudinal vortices, which extend over the entire length of the cavity. The time-averaged velocities resulting from the $k-\varepsilon$-AWF produced four longitudinal vortices. The experimental measurements do not cover the entire length in the spanwise direction. With assumption of symmetric flow and extrapolation of the experimental data, it can be concluded that there are four longitudinal vortices, but on the other hand the RSM spanwise profiles are closer to the experimental data. Figure 7.62 shows the velocity vector plot contours and the mean temperature, within the central, Z=0.5, longitudinal, x-y, plane. They confirm that in the cavity core conditions are practically isothermal, with the temperature close to the average of the hot and cold side temperatures and that the flow velocity is stronger near the end walls. Figure 7.63 presents the vector plot and mean temperature contours within the plane parallel to the thermally active planes, y-z, at the halfway location, X=0.5. Again both plots show the presence of three longitudinal cells which extend over most of the length of the cavity. This is consistent with the predicted presence of the three longitudinal vortices shown Figures 7.60 and 7.61.
Figure 7.48 – Time average temperature distributions resulting from RSM, Z=0.5.
Figure 7.49 – Time average temperature distributions resulting from RSM, Z=0.25.
Figure 7.50 – Time average temperature distributions resulting from RSM, Z=0.75.
Figure 7.51 – 3D-Time average mean velocity distributions resulting from RSM, Z=0.5.
Figure 7.52 – Time average mean velocity distributions resulting from RSM, Z=0.5.
Figure 7.53 – Time average mean velocity distributions resulting from RSM, X=0.5.
Figure 7.54 – Time average mean velocity distributions resulting from RSM, X=0.5.
Figure 7.55 – Time average rms velocity fluctuations resulting from RSM, $Z=0.5$. 
Figure 7.56 – Time average rms velocity fluctuations resulting from RSM, Z=0.5.
Figure 7.57 – Time average rms velocity fluctuations resulting from RSM, X=0.5.
Figure 7.58 – Time average rms velocity fluctuations resulting from RSM, \( X = 0.5 \).

Figure 7.59 – Nusselt distributions resulting from RSM.
Figure 7.60 – Time average vector plots within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent RSM (Y=0.5).
Figure 7.61 – Time average temperature contours within x-z plane of 15° unstable inclined tall cavity at different Y sections resulting from 3-D time dependent RSM (Y=0.5).
Figure 7.62 – Time average vector plots and temperature contours within x-y plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (Z=0.5).
Figure 7.63 – Time average vector plots and temperature contours within y-z plane of 15° unstable inclined tall cavity resulting from 3-D time dependent RSM (X=0.5).

7.2.9 FFT analysis-RSM

In the Figures 7.64-7.69, the dominant frequencies of resolved mean temperature resulting from RSM are shown at several locations. They are also compared with the frequencies resulting from the experimental measurements. The comparisons show that the time-dependent RSM well-predicted the range of dominant frequencies and their power densities compared with the experimental data. It is worth to note that URANS $k$-$\varepsilon$-AWF reproduced the range of dominant frequencies and their power densities close to experimental data as well. Therefore both time-dependent $k$-$\varepsilon$-AWF and RSM are able to resolve the large-scale fluctuations related to the large-scale three-dimensional flow structures.
Figure 7.64 – Power spectral diagram for temperature resulting from RSM, Z=0.5.
Figure 7.65 – Power spectral diagram for temperature resulting from RSM, Z=0.5.
Figure 7.66 – Power spectral diagram for temperature resulting from RSM, Z=0.25.
Figure 7.67 – Power spectral diagram for temperature resulting from RSM, Z=0.25.
Figure 7.68 – Power spectral diagram for temperature resulting from RSM, $Z=0.75$. 

Numerical

Experimental


Figure 7.69 – Power spectral diagram for temperature resulting from RSM, Z=0.75.

7.3 3D time-dependent simulation of inclined tall cavity - $15^\circ$ stable

This test case is that where the tall walls of the cavity are inclined at an angle of $15^\circ$ to the horizontal. In this case experimental data are available for a Rayleigh number value $Ra = 1.4 \times 10^6$ (Cooper et al. [8]). The hot and
cold wall temperatures are 53°C and 19°C respectively. Apart from the hot and cold walls, adiabatic condition is imposed to the rest of the walls. The fluid inside the tall cavity is air. The height of the cavity is 2.18m (H) and the distance between the hot and cold faces is 0.076m (L) and the span wise length is 0.52m (W). Comparisons between predictions and measurements are presented for seven different planes, of which six are longitudinal, x-y, and one is a spanwise, y-z, plane. The x-y planes are located at Z=0.25, 0.5, 0.625, 0.655, 0.75 and 0.875. The y-z plane is located at X=0.5. Within each plane the comparisons are made at the several traverse lines. Experimental data are available for the mean temperature, longitudinal and wall-normal mean velocities and the corresponding velocity fluctuations.

Figure 7.70 – 15° stable inclined cavity.

7.3.1 Flow Pattern

The temperature Iso-surfaces of Figure 7.71, provide some idea of the kind of flow predicted by the $k$-$\varepsilon$-AWF. The $k$-$\varepsilon$-AWF predicts that the flow deviates from thorough two-dimensionality, though it does remain symmetric, near the end walls.
Figure 7.71 – Temperature iso-surfaces in tall 15° stable tilted cavity resulting from $k$-$\varepsilon$ using AWF at $Ra = 1.4 \times 10^6$.

### 7.3.2 Simulation by $k$-$\varepsilon$-AWF

In the Figures 7.72-7.88, the results of the numerical simulation by $k$-$\varepsilon$-AWF turbulence modelling are compared with the experimental measurements along several sections at various locations in the cavity. In Figure 7.72-7.76, the temperature, velocity and velocity fluctuations resulting from the numerical simulation are compared with experimental data within the central longitudinal plane, $x$-$y$ plane at $Z=0.5$, along traverse lines at a number of longitudinal locations, from the bottom to the top of the cavity. The temperature comparisons, Figure 7.72, show close agreement between the numerical and experimental data. Near the bottom, the core temperature is closer to the cold temperature, which suggests that fluid from the cold side is convected to the core.
The reverse feature is observed at the top end, where the core fluid is at a temperature closer to that of the hot side. Figure 7.73 shows large differences between the numerical and experimental data. The numerical simulation is mostly similar to the two-dimensional simulation in which within the central longitudinal plane there is a circulation cell with fluid moving upwards along the hot, upper side and then returning downwards along the cold, lower side. In contrast, the experimental data show that within this plane the predominant motion is in the downward direction, which suggests that within other longitudinal planes, closer to the side walls, there must be a compensating downward motion and thus strongly three-dimensional flow conditions. The experimental data suggests that probably because at this angle of inclination the buoyancy force is weak, even with a spanwise aspect ratio as high as 6.8, the presence of the end walls causes the development of a three-dimensional flow, with downward flow in the middle and upward flow along the sides. This is a feature that the \( k-\varepsilon \)-AWF model, is unable to reproduce. The reason may be that EVM tends to produce too high values for turbulent viscosity which prevent the flow breaks down to smaller cells. Corresponding comparisons for the velocity fluctuations in the longitudinal and normal directions are presented in Figures 7.75 and 7.76. Comparisons show that while near the end walls the velocity fluctuations are severely under-estimated, over the rest of the cavity the \( k-\varepsilon \)-AWF returns the same fluctuation levels as those found in the measurements, though the distribution is different. Nevertheless, the large deviation between the predicted and measured flow fields, makes these comparisons less meaningful.

Figures 7.77-7.80 present comparisons between measured and predicted temperature profiles in the off-centre longitudinal planes, \( Z=0.25, 0.655, 0.75 \) and 0.875. The comparisons show that, as is also the case for the central longitudinal plane, \( Z=0.5 \), the core temperature is close to that of the cold side near the bottom end wall and close to that of the hot side near the top end wall and also that the predictions are in reasonable agreement with the measurements. The fact that measured temperature field, in agreement with the predicted one is mainly two-dimensional, suggests that the three-dimensional flow features exert a weak influence on the temperature field. Figures 7.81 and 7.82 present
comparisons of the longitudinal mean velocity within two off-centre longitudinal planes, at Z=0.625 and Z=0.75, respectively. These comparisons show that at these off-centre planes, the deviations between predictions and measurements, while still substantial, are not as large as those within the centre plane, observed in Figure 7.73. This is especially the case near the two end walls, where the predicted velocity profiles are no longer anti-symmetric and have the same shape as the measured ones, though not the same levels. These deviations from two-dimensionality in the predicted flow at the end-wall regions are consistent with the corresponding deviations in the temperature field, observed in the iso-surface plots of Figure 7.71. The corresponding comparisons for the velocity fluctuations in the longitudinal direction, are presented in Figures 7.83 and 7.84. In common with what has been observed in the comparisons for the central longitudinal plane of Figure 7.75, near the end walls fluctuation levels were under-estimated and the over the rest of the cavity the core levels predicted are similar to those measured, though as also noted in the central plane comparisons, the actual distribution is not well reproduced. Again due to the deviations between the predicted and measured mean flow fields, further discussion of these comparisons is not likely to be fruitful.

Finally Figures 7.85, 7.86, 7.87 and 7.88, present comparisons of the longitudinal and normal components of the mean velocity and of velocity fluctuations respectively, along spanwise (z-direction) traverse lines, within the mid-plane parallel to the thermally active planes, x-z plane. These Figures clearly shows the strong three dimensionality of experimental data. Figure 7.85 clearly show that based on the experimental measurements, there are two strong circulation cells which move upward near Z=0 and Z=1 and move downward near Z=0.5. The numerical data produce the similar but weaker movement near the top of the cavity but it is not strong enough to penetrate all the way down to the bottom of the cavity. Also the numerical simulation produces similar circulations at the bottom with the opposite direction.

As also seen in the previous comparisons, the $k-\varepsilon$-AWF, even when used in a three-dimensional mode, in this case fails to reproduce the three-dimensionality of the flow field. The corresponding comparisons of the rms longitudinal and
normal components in Figures 7.87 and 7.88 also for the most part confirm the conclusions of the earlier comparisons. Near the upper and lower end walls, the fluctuation levels are under-estimated and elsewhere predicted core levels are close to those measured and now some of the features of the spanwise variation are also reproduced.

Clearly in contrast to the $15^\circ$ unstable angle, the $k-\varepsilon$-AWF, even though it reproduces the main features of the temperature field, it has failed to predict the three-dimensionality of this flow. This predictive failure may be caused by the fact that the three-dimensional flow features are not as strong as for the $15^\circ$ unstable angle of inclination. In order to explore ways of improving the URANS predictions, the subsequent sections present comparisons obtained first with a version of the $k-\varepsilon$-AWF which includes a limiter for the turbulent viscosity based on realisability criteria and then with the Basic Reynolds Stress Closure.
Figure 7.72 – Temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.73 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.74 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.75 – rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.5$. 
Figure 7.76 – rms velocity fluctuations resulting from $k-\varepsilon$-AWF, $Z=0.5$.

Figure 7.77 – Temperature distributions resulting from $k-\varepsilon$-AWF, $Z=0.25$. 
Figure 7.78 – Temperature distributions resulting from \(k-\varepsilon\)-AWF, \(Z=0.655\).

Figure 7.79 – Temperature distributions resulting from \(k-\varepsilon\)-AWF, \(Z=0.75\).
Figure 7.80 – Temperature distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.875$.

Figure 7.81 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.625$. 
Figure 7.82 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, $Z=0.75$.

Figure 7.83 – rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.625$. 
Figure 7.84 – rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $Z=0.75$. 
Figure 7.85 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, X=0.5.
Figure 7.86 – Mean velocity distributions resulting from $k$-$\varepsilon$-AWF, X=0.5.
Figure 7.87 – rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF, $X=0.5$. 

![Graphs showing rms velocity fluctuations](image)
7.3.3 Simulation by $k$-$\varepsilon$-AWF with turbulent viscosity limitation

Figures 7.89-7.105 show results of the 3D numerical simulation of 15° stable inclined cavity compared with the experimental data. The numerical simulation is carried out by $k$-$\varepsilon$-AWF with implementation of turbulent viscosity constraint presented in equation 3.14. In the previous Section, the results from $k$-$\varepsilon$-AWF without the turbulent viscosity limitation were shown. The calculation without the turbulent viscosity limit produced steady-state results even with the time dependent simulation. The calculation with the turbulent viscosity limit, on the other hand, results in a time-dependent solution. Therefore time-averaged values of mean temperature, mean velocity and rms velocity are compared with the experimental data within several planes and sections. In general the flow pattern (e.g. Figure 7.90) is similar to the one explained in the previous section (Figure 7.73). Fluid movement mainly takes place within the longitudinal, x-y plane where the fluid moves up along the hot wall and moves down along the cold wall. Within the plane parallel to the thermally
active sides, y-z, there are recirculation cells similar to those discussed in the previous section.

Figures 7.89, 7.94, 7.95, 7.96 and 7.97 show the temperature distributions within a number of longitudinal planes. These comparisons reveal that the implementation of turbulent viscosity limit does not make significant changes to the temperature profiles. There are only slight changes near the top and bottom of the cavity. Figures 7.90, 7.98, 7.99 present the corresponding longitudinal velocity distributions within longitudinal, x-y, planes and Figures 7.102 and 7.103 the distribution of the longitudinal and normal velocities within the mid-plane parallel to the thermally active sides, x-z plane at Y=0.5. The comparisons show that introduction of the viscosity limiter, in addition to enabling the $k$-$\varepsilon$-AWF to return unstable flow conditions, leads to some improvements in the predicted time-averaged velocity field, but there still large deviations between the predicted and measured flow fields. Figures 7.92, 7.93, 7.100, 7.101, 7.104 and 7.105 show the corresponding comparisons for the corresponding rms velocity profiles. Improvements in the predicted rms are evident, most notably near the end walls, where the predicted levels are now significantly higher than those of $k$-$\varepsilon$-AWF predictions without the viscosity limiter, and close to the measured values. These predictive improvements are most likely to result from the fact that the predicted rms velocities now include both the modelled component from the solution of the transport equation for $k$, which represents the contribution of the small-scale motion, and the resolved component, obtained by post processing the instantaneous velocity and which represents the contribution of the large-scale unsteady motion. This is consistent with the fact that the largest improvements in the predicted rms velocity field is observed near the two end walls, where the predicted flow is most unstable. Finally in Figure 7.106, the time-averaged longitudinal distributions of the Nusselt number at four spanwise locations, $Z=0.5, 0.625, 0.75, 0.865$, predicted by the $k$-$\varepsilon$-AWF, are compared to the measured data. At all spanwise locations, the low Nu levels near the bottom end wall are under estimated, the higher levels near the top end wall are well predicted, but the predicted longitudinal variation in local Nusselt number from the bottom to the top end wall is non-monotonic. The local Nusselt number rises from zero at the bottom end
wall to either a maximum, or a plateau, then shows a dip, before rising again near the top end wall. The experimental data are not detailed enough but a local Nu variation like the one indicated by the \( k-\varepsilon \)-AWF predications is unlikely. The three-dimensional \( k-\varepsilon \)-AWF computations, thus return the correct levels for the Nusselt number, but not the correct variation.

Figure 7.89 – Time-averaged temperature distributions resulting from \( k-\varepsilon \)-AWF with limiter, \( Z=0.5 \).
Figure 7.90 – Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. 
Inclined Cavity-3D time-dependent simulations

Figure 7.91 – Time-averaged mean velocity distributions resulting from $k-\varepsilon$-AWF with limiter, $Z=0.5$. 
Figure 7.92 – Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. 
Figure 7.93 – Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.5$. 

Inclined Cavity-3D time-dependent simulations
Figure 7.94 – Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.25$.

Figure 7.95 – Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.655$. 
Figure 7.96 – Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. 
Figure 7.97 – Time-averaged temperature distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.875$. 
Figure 7.98 – Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.625$. 
Figure 7.99 – Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. 
Figure 7.100 – Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.625$. 
Figure 7.101 – Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, $Z=0.75$. 
Figure 7.102 – Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $X=0.5$. 
Figure 7.103 – Time-averaged mean velocity distributions resulting from $k$-$\varepsilon$-AWF with limiter, $X=0.5$. 

---

$U/V_0$ vs $Z$ for $Y=0.9$, $0.5$, and $0.1$. The plots show the comparison between experimental data (EXP) and the $k$-$\varepsilon$-AWF model.
Figure 7.104 – Time-averaged rms velocity fluctuations resulting from $k$-$\varepsilon$-AWF with limiter, X=0.5.
Figure 7.105 – Time-averaged rms velocity fluctuations resulting from $k-\varepsilon$-AWF with limiter, $X=0.5$. 
7.3.4 Simulation by RSM-Basic

Since neither of the two versions of the $k-\varepsilon$ tested has been as successful in reproducing the three-dimensional flow features of the $15^\circ$ stable cavity, as was the case for the $15^\circ$ unstable angle, the next logical step was to introduce the Basic Reynolds stress model, RSM, again with the analytical wall function. In this section, the flow simulation by RSM-Basic using AWF is compared with the experimental data. Even though the computation was time-dependent as well as three-dimensional, the solution reached the steady-state conditions in the middle of the cavity, with some unsteadiness near the end walls. For this reason all the comparisons presented in this section, involve time-averaged predictions.

Figures 7.107, 7.112, 7.113 and 7.115 present the mean temperature distribution within longitudinal planes. The comparisons show that, in common with the $k-\varepsilon$, the Basic RSM, apart from some minor discrepancies, is able to
reproduce the measured temperature field. As pointed out earlier, the temperature field remains mostly two-dimensional, and it is consequently easier to reproduce than the three-dimensional flow field. Figures 7.108, 7.109, 7.116, 7.117 present comparisons between RSM-predicted and measured profiles of the longitudinal and normal mean velocity within longitudinal planes and Figures 7.120 and 7.121 show similar comparisons within the mid-plane parallel to the two thermally active sides. The comparisons show that introduction of the Basic RSM results in encouraging improvements in the flow field predictions inside the stable $15^\circ$ inclined cavity. In reasonable, though not complete, agreement with the experimental data, within the central longitudinal plane, Figure 7.108, the Basic RSM predicts a downward overall motion, while as can be seen in Figures 7.116 and 7.117, towards the side-walls, again in accord with the measurements, the balance of the longitudinal flow direction shifts towards the upward direction. This suggests that in agreement with the measurements and in contrast to the $k-\varepsilon$ predictions, the Basic RSM returns a two-longitudinal-cell structure, with the fluid moving downwards within the centre of the cavity and upwards along the two side walls. The predicted profiles along the spanwise traverses, Figures 7.120 and 7.121, provide further and indeed clearer confirmation. Figures 7.110, 7.111, 7.118, 7.119, 7.122 and 7.123 present the rms velocity profiles produced by the Basic RSM. The detailed comparisons show that within the longitudinal planes, fluctuation levels and the distribution are well predicted near the end walls, but over-estimated towards the middle of the cavity. Within the mid-plane parallel to the thermally active sides, Figure 7.122, differences between predicted and measured velocity fluctuations are still substantial. Use of the RSM leads to closer agreement between predicted and measured rms velocities than the deployment of the $k-\varepsilon$, but overall agreement is not as satisfactory as for the mean flow field. Apart from few of the sections which are over-predicted, RSM-Basic improves not only the magnitude of rms velocities but also the shape of the profiles. Figure 7.124 shows comparisons of the time-averaged longitudinal distributions of the Nusselt number at four spanwise locations, $Z=0.5, 0.625, 0.75, 0.865$, predicted by the RSM-AWF, with the measured data. Unlike the corresponding $k-\varepsilon$ computations, the RSM predictions, show a monotonic rise in Nusselt number in the longitudinal direction, which is in accord with the
data. In common with the $k\varepsilon$ predictions, the Nusselt number level near the bottom end wall is under-estimated. Nusselt number levels are also under-estimated along the longitudinal traverse closest to the side-wall, $Z=0.865$. In contrast to the earlier comparisons, the Nu variation of the RSM model is in reasonable agreement with the measurements. This is consistent with earlier findings, that for the $15^\circ$ stable inclination, agreement between experimental and measured flow fields is only acceptable for unsteady three-dimensional flow computations, with a Reynolds stress model.

Figures 7.125 and 7.126 show vector plot and mean temperature contours within spanwise, x-z, planes at longitudinal location $Y=0.5$. The vector plot shows that two counter rotating circulation cells with identical size are located in the plane. The mean temperature contours indicate almost linear distribution of mean temperature from the hot wall to the cold wall which is expected to be observed in the thermally stable configuration. Figure 7.127 shows mean temperature contours within the central, $Z=0.5$, longitudinal, x-y, plane. It indicates that the top left corner is nearly covered with the hot temperature and the bottom right corner has the cold temperature. In the rest of the plane, mean temperature is almost distributed linearly. Figure 7.128 presents vector plot and mean temperature contours within the plane parallel to the thermally active planes, y-z, at the halfway location, $X=0.5$. They show that RSM well-predicted two longitudinal vortices spread over the entire length of the cavity consistent with the experimental data.
Figure 7.107 – Time-averaged temperature distributions resulting from RSM, Z=0.5.
Figure 7.108 – Time-averaged mean velocity distributions resulting from RSM, \( Z=0.5 \).
Figure 7.109 – Time-averaged mean velocity distributions resulting from RSM, \( Z=0.5 \).
Figure 7.110 – Time-averaged rms velocity fluctuations resulting from RSM, \( Z=0.5 \).
Figure 7.111 – Time-averaged rms velocity fluctuations resulting from RSM, \( Z=0.5 \).
Inclined Cavity-3D time-dependent simulations

Figure 7.112 – Time-averaged temperature distributions resulting from RSM, Z=0.25.

Figure 7.113 – Time-averaged temperature distributions resulting from RSM, Z=0.655.
Figure 7.114 – Time-averaged temperature distributions resulting from RSM, Z=0.75.
Figure 7.115 – Time-averaged temperature distributions resulting from RSM, $Z=0.875$. 
Figure 7.116 – Time-averaged mean velocity distributions resulting from RSM, Z=0.625.
Figure 7.117 – Time-averaged mean velocity distributions resulting from RSM, $Z=0.75$. 
Figure 7.118 – Time-averaged rms velocity fluctuations resulting from RSM, $Z=0.625$. 
Figure 7.119 – Time-averaged rms velocity fluctuations resulting from RSM, Z=0.75.
Figure 7.120 – Time-averaged mean velocity distributions resulting from RSM, $X=0.5$. 
Figure 7.121 – Time-averaged mean velocity distributions resulting from RSM, $X=0.5$. 

Inclined Cavity-3D time-dependent simulations
Figure 7.122 – Time-averaged rms velocity fluctuations resulting from RSM, $X=0.5$. 
Figure 7.123 – Time-averaged rms velocity fluctuations resulting from RSM, X=0.5.
Figure 7.124 – Nusselt distributions resulting from RSM.

Figure 7.125 – Vector plots within x-z plane of 15° stable inclined tall cavity where Y=0.5 resulting from RSM.
Figure 7.126 – Temperature contours within x-z plane of 15° stable inclined tall cavity where Y=0.5 resulting from RSM.

Figure 7.127 – Temperature contours within x-y plane of 15° stable inclined tall cavity resulting from RSM.
Figure 7.128 – Vector plots and temperature contours within y-z plane of 15° stable inclined tall cavity resulting from RSM (X=0.5).

7.4 Closing remarks

In this chapter results of 3D time-dependent numerical simulations of 15° stable and unstable inclined cavities are presented. In-house STREAM code was employed to carry out the simulations. Primarily, the buoyancy related terms existing in the momentum and turbulence transport equations were not included in STREAM code. The buoyancy related terms present in AWF were also not included in STREAM code. Thus, these terms were added into the STREAM code. Then 2D simulations of vertical and 60° inclined tall cavities which were already simulated by TEAM code, were carried out again by STREAM code to validate the changes introduced to STREAM code.

The 3D time-dependent simulation of 15° unstable test case by the $k$-$\varepsilon$-AWF
produces very encouraging results. It can capture important features of this complicated flow. The experimental data reported existence of multiple longitudinal vortices extended all the way along the cavity. The $k-\varepsilon$-AWF predicted these longitudinal vortices although the strength and number of vortices do not match everywhere with the experimental data. Thus RSM-Basic is also implemented to examine whether or not it is possible to have further improvements. The comparisons show that RSM-Basic prediction is closer to experimental measurement regarding the position and strength of longitudinal vortices as well as stress field.

The 3D time-dependent simulation of $15^\circ$ stable case by $k-\varepsilon$-AWF resulted in almost two-dimensional flow field. It produces a slightly 3D flow structures near the two ends of the cavity similar with the pattern observed in the experimental study. In contrast, the experimental study reported the 3D flow structure extends all the way along the cavity. Finally, implementation of RSM-Basic to simulate $15^\circ$ stable inclined tall cavity produced significant improvement in capturing the 3D flow structure compared with the experimental data.
Chapter 8

Annular horizontal penetration

In this chapter the result of numerical simulation of natural convection flow within an annular horizontal penetration is presented. Horizontal annular penetrations, in which buoyant flow is generated by the difference in temperature between the cold inner tube and the warmer core fluid outside it, can be found in Advanced Gas Cooled nuclear reactors. The resulting flow is three-dimensional and it also involves a horizontal thermally active surface. There are consequently a number of similarities with the buoyancy-driven flows through highly inclined cavities, especially the unstable 15° case, which involved unstable stratification. In the penetration, underneath the cold inner tube, there is thermally unstable stratification where the cold inner tube is located above hotter fluid. Having demonstrated that the $k$-$\varepsilon$-AWF model, when used in unsteady mode was able to reproduce the resulting flow complexities, this modelling approach was chosen as a suitable compromise between predictive accuracy and computational efficiency. In order to gain a better understanding of the flow development, computations at two Rayleigh number values have been produced. Moreover at each Rayleigh number, steady-state and time-dependent computations have been carried out and the steady-state flow has been compared with the time-averaged flow of the time-dependent computations. In Section 8.1, the test case geometry and thermal boundary conditions are discussed. In Section 8.2, the grids generated are explained. In Section 8.3, the results of the steady-state simulation of the lower Ra case ($Ra = 4.5 \times 10^8$) are shown and they are discussed in detail. In Section 8.4, the results of the time-dependent simulation of the lower Ra case ($Ra = 4.5 \times 10^8$)
are presented. In Section 8.5, the results of the steady-state simulation of the higher Ra case (Ra = 3.1 \times 10^{13}) are discussed. In Section 8.6, the results of the time-dependent simulation of the higher Ra case (Ra = 3.1 \times 10^{13}) are shown. Finally in the section 8.7, the closing remarks are presented.

8.1 Test case description

This test case is about simulation of flow inside a horizontal annulus with one open side. The inner (D1) and outer (D2) diameters of the annulus shown in Figure 8.1 are 0.026m and 0.343m, respectively. The length of the annulus (H) is 5.68m. The size of a domain outside the annulus which is simulated by the numerical computation is chosen as h=0.82m, L1=0.8285m, L2=3.3285m, L=4.5m and W=3.5m. The fluid enters the flow domain from the top boundary at a temperature of 573.3K, while the inner surface of the annulus is maintained at a temperature of 338.3K. The inner surface of the annulus is extended outside the annulus but the outside part is not thermally active and it is set to be adiabatic surface. All other walls which are signed by hatching are set to be adiabatic walls as well. Regarding the domain outside the annulus, symmetry boundary conditions are applied to the faces apart from the inlet, outlet and the walls. At the outlet, zero gradient condition was imposed to all variables except the components of velocity parallel to the outlet plane. These components of velocity are set to zero. Likewise, these components of velocity are set to zero at the inlet as well. Moreover, at the inlet, values of \(k\) and \(\varepsilon\) are set to a constant magnitude which were calculated based on 0.1 percent of \(V_0\) and \(\mu_t/\mu = 2\). The fluid is air, Pr=0.7, at a density of 1.2 kg/m^3 and a dynamic viscosity of \(1.7 \times 10^{-5} \text{ m}^2/\text{s}\). The Rayleigh number based on the diameter of the annulus is \(Ra = 4.5 \times 10^8\). Rayleigh number was increased to \(Ra = 3.1 \times 10^{13}\) by increasing the dimensions of the geometry, where \(Ra = \frac{3g\Delta\Theta D_1^4}{\nu\alpha}\). Therefore, increase of diameter of the tube produced the mentioned higher Ra. The ratio of outer tube diameter to inner tube diameter was kept constant for both lower and higher Ra test cases.
Figure 8.1 – Test case geometry and thermal boundary conditions.

8.2 Grid

The grid generated for this test case consists of 20 blocks for the lower Ra and 39 blocks for the higher Ra and Message Passing Interface (MPI) is used for
parallel processing. In the figures 8.2-8.4, a number of sections of the grid generated for the lower Ra test case are shown. The number of nodes employed for the lower Ra and higher Ra were 2.5 and 6.5 million nodes, respectively. The grids are generated so that the values of $y^*$ at the near-wall cells do not anywhere exceed 200. A FORTRAN code was developed to generate these grids. The code writes the position of nodes, block by block in the data files. The STREAM code, in turn, reads the position of all nodes from the data files. For a grid sensitivity test, grids with 4.5 and 10 million nodes were generated and employed for the lower Ra and higher Ra case, respectively. The computations with these finer grids produced identical results with the coarser grids.

Figure 8.2 – Grid in $y$-$z$ plane.
Figure 8.3 – Grid in x-y plane.

Figure 8.4 – Grid inside the tube.
8.3 Steady-state simulation \((Ra = 4.5 \times 10^8)\)

This section presents the results of steady state simulation of horizontal annular open penetration. Figures 8.5-8.8 show the contour plots of \(U, V, W, P', T, k\) and \(\mu_t\) within the central, \(X=0\), longitudinal, \(y-z\), plane, inside and outside of the penetration. The positive axial velocity \((W)\) denotes velocity from right to left, thus coming out of the penetration, and positive vertical velocity \((V)\) denotes upward motion. It can be seen, from the \(V\) contours of Figure 8.5 that the flow enters to the computational domain at the inlet and moves down. The downward flow outside the penetration becomes faster along the vertical wall, below the open end of the penetration. The axial velocity outside the penetration is considerably lower than the vertical. The contours of both the axial and vertical velocity suggest that within the annular penetration there are irregular flow structures, which is consistent with flow unsteadiness. Axial flow is confined to the penetration and the region just outside it. Over the top half of the penetration, above the cold central core, the axial motion is predominantly inward, towards the closed end. Over the lower half of the penetration, the axial flow direction mostly reversed, with predominantly outward flow, towards the open end of the penetration. The contour plot of the vertical velocity component shows that in the penetration, as already pointed out, within the central vertical plane an irregular flow pattern is observed, with frequent changes in the direction of the vertical velocity along the axial direction.

This observed flow behaviour is consistent with fluid within the annular penetration being cooled by the central core, then due to its higher density falling first out of the cavity and then dropping out of the flow domain at the lower boundary and then hot fluid being drawn into the flow domain from the top and then being drawn into the horizontal penetration to replace the cooled fluid. The spanwise, \(U\), velocity component, as can be seen in Figure 8.5, remains low throughout. The mean temperature contours of Figure 8.7, which show that the fluid at the lower half of the penetration is at a lower temperature than the inlet temperature and that some of it spills out, are consistent with those for the axial and vertical velocity components. It is also worth noting that the fluid temperature drops by only a fraction of the difference between the hot inlet and the cold pipe temperatures. The reason is that
the diameter of the cold pipe is small in comparison to the diameter of the penetration. Contours of the deviation of the pressure field from the hydrostatic condition, presented in Figure 8.6, show that outside the penetration the pressure field deviates very little from the hydrostatic variation, while within the penetration, there is a stronger positive deviation from hydrostatic conditions over the lower half and a more modest negative deviation over the top half. This is consistent with the observed temperature variation.

The $k$ contours and contours of turbulent viscosity, shown in Figures 8.7-8.8, indicate that turbulence levels at this Rayleigh number are low and also that at some regions become negligible. Within the penetration, most high turbulent viscosity regions are found over the lower half, below the cold central core of the annular penetration, where temperature stratification is unstable. The highest turbulent viscosity levels, however, occur outside the penetration, along the vertical wall and close to the lower exit plane, where as noted earlier, the vertical velocity is at its highest.

Figures 8.9-8.10 show the contour plots of $U$, $V$, $W$, $P^r$, $T$ and $\mu_t$ inside the penetration, within cross-sections normal to the axis of the penetration, at locations $Z=0.23$, $0.46$, $0.69$ and $0.87$, which correspond to a quarter of the penetration length away from the closed end, the half-way location, three quarters away from the penetration closed end and at the open end respectively. The mean velocity contours, but especially those of the vertical velocity, demonstrate that the flow field is non-symmetric at the closed end of the penetration and becomes more symmetric towards the open end. At the exit plane, $Z=0.87$, the vertical motion is mostly in the downward direction. At other locations further inside the penetration, while there is always downward motion at the lower surface of the inner pipe this seems to be deflected sideways, like at $Z=0.23$ and $Z=0.46$, or as in the case of $Z=0.69$ to be opposed by an overall upward motion. As far as the axial velocity is concerned, at the penetration inlet, the fluid enters the penetration mainly through the central region and comes out mainly from the lower region of the cross-section. Further inside the cavity, at $Z=0.69$, the inward motion is displaced to the upper part, with outward moving flow over the rest of the cross-section. Closer to the closed end, $Z=0.43$
and $Z=0.23$, the axial motion diminishes.

The corresponding mean temperature contour plots, in Figure 8.10, show that there is a thin fluid layer around the inner pipe which is at a temperature close to that of the cold inner pipe, but as it drops off the pipe, due to its higher density, and mixes with the surrounding warmer fluid, its temperature rises. As a result, even though most of the fluid within the entire penetration is at a temperature which is very close to the hot inlet temperature, there is nevertheless a weak, stable temperature stratification with the fluid at the top half of the cross-section at a slightly higher temperature than the fluid at the lower half. This small temperature differential is of course what generates the bulk fluid motion.

The contours of the deviation of static pressure from the hydrostatic variation, included in Figure 8.10, confirm the earlier observation that the deviation is zero in the middle, modestly negative in the top half and more strongly positive over the lower half of the cross-section. Moreover, the variation in the pressure deviation becomes weaker at the open end of the penetration. These changes in pressure mirror those of the temperature.

The contour plots of the turbulent viscosity, also included in Figure 8.10, suggest that at the closed end, $Z=0.23$, certainly at this Rayleigh number, the flow is mostly laminar with one or two pockets of low turbulence. The turbulent regions expand away from the closed end and at the exit of the penetration, the flow is turbulent over most of the cross-section, though the turbulence levels are rather low. The low turbulence levels observed are a consequence of the low velocities developed, which in turn can be attributed to the fact that the surface area of the cold pipe is small and hence its cooling effect is limited.

Figures 8.11-8.13 present the contour plots of $U$, $V$, $W$, $P''$, $T$ and $\mu_t$ also within planes normal to the penetration, but at locations outside the penetration, $Z=0.88$, near the open end of the penetration, $Z=0.92$, in the middle and $Z=1$, at the symmetry boundary of the computation domain. There is strong
The downward motion close to the penetration exit, $Z=0.88$, which becomes progressively weaker towards the symmetry boundary, $Z=1$. Moreover, within each plane, the vertical, downward motion is strongest at the centre and reduces towards the sides, $X=4.5$ and $-4.5$. Neither of these trends is unexpected, since, the downward motion is generated by the cooling of the fluid within the penetration, which is located at the middle of the domain, $X=0$. The two horizontal velocity components, $U$ and $W$, as seen in Figures 8.11 and 8.12, are practically zero outside the penetration. The corresponding turbulent viscosity contours, Figure 8.13, also show that the turbulent flow region is mainly confined to the region close to the exit of the penetration and below the penetration exit. This is consistent with the shear that develops along the vertical wall, below the penetration exit, as the heavier fluid that comes out of the penetration is pulled downwards along the vertical wall. As was also suggested by the contour plots of Figure 8.7, conditions outside the penetration are practically isothermal, which in turn explains why the pressure field, as shown in Figure 8.12, is practically hydrostatic.

Figures 8.14-8.15 present the contour plots of $V$, $T$, $P^\prime$, $k$ and $\mu_t$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and 0.5, where 0.25 is half way between the centre and the boundary and 0.5 is at the symmetry boundary. All of them show that away from the centre ($X=0$), the flow becomes weak and consequently, the temperature field is left unchanged. As a result of the uniform fluid temperature, the pressure variation is very close to hydrostatic. The turbulence along the vertical wall diminishes with distance from the centre, which is consistent with the variation in vertical velocity.

Figure 8.16 shows contour plots of $V$ and $\mu_t$ within horizontal planes outside the penetration, at locations $Y=0.22$(inlet), $Y=-0.11$(above the open end of the penetration) and $Y=-0.78$(outlet). They show that the maximum downward velocity occurs at the centre ($X=0$). The downward, $V$, velocity is concentrated at the centre of the region outside the penetration and its maximum value increases with downward distance. There is no turbulence generation above the open end of the penetration, but by the exit plane, at $Z=-0.78$, the
boundary layer across the entire width of the vertical wall is turbulent.

Finally, because the steady-state computation was not fully converged and the results were slightly changing from iteration to iteration, it was decided to carry out time-dependent simulation for this case which is presented in the next section.
Figure 8.5 – Contour plots of $U$ and $V$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 
Figure 8.6 – Contour plots of $W$ and $P^r$ within longitudinal, y-z, plane, at X=0, at $Ra = 4.5 \times 10^8$. 
Figure 8.7 – Contour plots of $T$ and $k$ within longitudinal, y-z, plane, at X=0, at $Ra = 4.5 \times 10^8$. 
Figure 8.8 – Contour plots of $\mu_t$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 
Figure 8.9 – Contour plots of U, V and W within cross-sections normal to the axis of the penetration, at Z=0.23, 0.46, 0.69 and 0.87, at Ra = 4.5 × 10^8.
Figure 8.10 – Contour plots of $P^*$, $T$ and $\mu_t$ within cross-sections normal to the axis of the penetration, at $Z=0.23$, 0.46, 0.69 and 0.87, at $Ra = 4.5 \times 10^8$. 
Figure 8.11 – Contour plots of U and V within cross-sections normal to the axis of the penetration, at Z=0.88, 0.92 and 1, at Ra = 4.5 \times 10^8.
Annular horizontal penetration

Figure 8.12 – Contour plots of $W$ and $P''$ within cross-sections normal to the axis of the penetration, at $Z=0.88, 0.92$ and $1$, at $Ra = 4.5 \times 10^8$. 
Figure 8.13 – Contour plots of $T$ and $\mu_t$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 4.5 \times 10^8$. 
Figure 8.14 – Contour plots of $V$, $T$ and $P^*$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and 0.5, at $Ra = 4.5 \times 10^8$. 
Figure 8.15 – Contour plots of $k$ and $\mu_t$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and 0.5, at $Ra = 4.5 \times 10^8$. 
Figure 8.16 – Contour plots of V and $\mu_t$ within horizontal planes outside the penetration, at locations Y=0.22 (inlet), Y=-0.11 and Y=-0.78 (outlet), at $Ra = 4.5 \times 10^8$. 
8.4 Time-dependent simulation \((Ra = 4.5 \times 10^8)\)

Figures 8.17-8.31 show the solution of 3D time dependent simulation of the horizontal penetration test case by \(k-\varepsilon\)-AWF. In the steady-state simulation, it has been observed that results are not symmetrical, specially inside the tube away from the open end of the tube. That is why time dependent calculations are employed for this simulation to obtain a better understanding of how flow develops in this horizontal penetration test case.

In the time dependent calculations, the normalized time step is set equal to 0.083, normalized by \(\frac{D_2}{V_0}\) (where \(D_2\) is the diameter of the tube and \(V_0 = \sqrt{g \beta \Delta \Theta D_2}\)) and the implicit scheme is employed for time discretisation. In each time step, it is set to iterate 20 times unless it gets to convergence criteria (all residuals below \(10^{-3}\)).

Figures 8.17-8.20 show velocity contours within the central \((X=0)\) longitudinal plane, inside and outside the penetration. They show that downward flow which separates from the cold tube is not uniform all along the tube. The shape of the downward flow changes with time. The component of velocity along the tube \((W/V_0)\) is time dependent as well. The contours show that the flow generally moves into the tube from the upper part of the tube and moves out from the lower portion of the tube but the shape of the flow changes with time.

Figures 8.21-8.26 show vector plots and contour plots of \(V, W\) and \(k\) within the penetration at cross-sections normal to its axis, at locations \(Z=0.23, 0.46, 0.69\) and 0.87, at different time steps. Both sets of mean velocity contours show that over most of the penetration \((Z = 0.23\) to 0.79\)) there are strong oscillations. By the exit, however, these oscillations appear to have died down. This is also confirmed by flow animation videos. It thus appears that as the buoyancy effect weakens and the velocities become lower, inside the penetration, flow conditions become more unstable. The corresponding contour plots of the instantaneous turbulent kinetic energy and also the plots of the instantaneous velocity vectors also confirm the oscillatory nature of the flow within the penetration.
One question that consequently arises, is how influential these instabilities are on the flow development, which in turn leads to the question of whether it is necessary to simulate these flows as three-dimensional and time-dependent. To address these questions, time-averaged predictions (resulting from the time-dependent computations) are compared with those obtained in the steady state computations. Time-averaging computations were carried out over a normalised period of time equal to 1000. This time period is larger than the time-scale calculated based on the largest length in the computational domain and $V_0$.

First, Figures 8.27 and 8.28 compare time-averaged and steady-state contours of the vertical and axial velocities respectively. The vertical velocity comparisons show differences in the flow structures within the penetration, with the steady-state computations returning a more fragmented flow field. The steady state computations also return a stronger downward motion along the vertical wall, below the open end of the penetration. The corresponding comparisons for the axial velocity in Figure 8.28, indicate that the time-averaged axial flow inside the penetration is stronger than that of the steady-state computations.

Figures 8.29 and 8.30 show similar comparisons, involving the same velocity components, but for cross-sectional planes normal to the axis of the penetration, at locations $Z=0.23$, 0.46, 0.69 and 0.87 within the penetration. The vertical velocity comparisons, Figure 8.29, show that there are noticeable differences in the two predicted flow fields. In contrast to those of the steady-state predictions, the time-averaged contours of the vertical velocity, are symmetric over the entire length of the penetration. Even at the exit, $Z=0.87$, where both approaches result in symmetric conditions, the two sets of contours have some differences. The corresponding comparisons of the axial velocity contours, Figure 8.30, are perhaps of greater significance. They confirm that the time-dependent simulation returns a stronger inward flow within the penetration than the steady state simulation. This is of course an important characteristic of this flow and these comparisons suggest that, at least for this Rayleigh...
number, it is necessary to employ a time-dependent RANS approach to reproduce it reliably. The corresponding comparisons for the contours of turbulent kinetic energy, which are presented in Figure 8.31 are equally interesting. For the time-averaged predictions, two sets of contours have been included, one for the modelled component, which essentially is the time-averaged solution of the time-dependent transport equation for the turbulent kinetic energy and the total turbulent kinetic energy, which consists of the modelled component plus the component resolved by the time-dependent computations. Within the penetration, $Z$ up to 0.69, there is a substantial difference between the total turbulent kinetic energy and its modelled component, which means that the greatest contribution to turbulence arises from the resolved, large-scale instabilities. This is of course a phenomenon which cannot be captured by the steady RANS, which is why the turbulent kinetic energy levels produced by the steady state computations within the penetration are substantially lower. At the exit of the penetration, $X=0.87$, where conditions were found to be steady, the differences between the total $k$ and its modelled component diminish and agreement between the time-averaged and steady state predictions is improved.

A number of conclusions can be drawn from the computations of flow in this horizontal penetration at a Rayleigh number of the order of $4.5 \times 10^8$. For the most part the change in fluid temperature is only a small fraction between the inlet hot and the pipe cold temperatures. This results in low normalised velocities and in regions of either no, or low turbulence. There are large-scale flow instabilities within the penetration, which need to be resolved by flow computations in order to reliably reproduce the inflow to the penetration.

Finally, it is worth to mention that currently a project is being carried out that is providing LES data for this case which can give an indication about the reliability of the steady-state and time-dependent computations.
Figure 8.17 – Contour plots of $V$ at different time steps within longitudinal, y-z, plane, at X=0, at $Ra = 4.5 \times 10^8$. 
Figure 8.18 – Contour plots of $V$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 

$\tau^* = 240$ 

$\tau^* = 360$
Annular horizontal penetration

$\tau^* = 0$

$\tau^* = 120$

Figure 8.19 – Contour plots of $W$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 
Figure 8.20 – Contour plots of $W$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 
Figure 8.21 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$, at $Ra = 4.5 \times 10^8$. 
Figure 8.22 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.46$, at $Ra = 4.5 \times 10^8$. 
Figure 8.23 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$, at $Ra = 4.5 \times 10^8$. 
Figure 8.24 – Contour plots of V, W and k at different time steps within cross-sections normal to the axis of the penetration, at Z=0.87, at Ra = 4.5 \times 10^8.
Figure 8.25 – Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.23$ and $0.46$, at $Ra = 4.5 \times 10^8$. 
Figure 8.26 – Vector plots at different time steps within cross-sections normal to the axis of the penetration, at Z=0.69 and 0.87, at $Ra = 4.5 \times 10^8$. 
Figure 8.27 – Contour plots of time-averaged and steady-state $V$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 4.5 \times 10^8$. 
Figure 8.28 – Contour plots of time-averaged and steady-state $W$ within longitudinal, y-z, plane, at X=0, at $Ra = 4.5 \times 10^8$. 
Figure 8.29 – Contour plots of time-averaged and steady-state $V$ within cross-sections normal to the axis of the penetration, at $Z = 0.23, 0.46, 0.69$ and $0.87$, at $Ra = 4.5 \times 10^8$. 
Figure 8.30 – Contour plots of time-averaged and steady-state $W$ within cross-sections normal to the axis of the penetration, at $Z=0.23$, $0.46$, $0.69$ and $0.87$, at $Ra = 4.5 \times 10^8$. 
Figure 8.31 – Contour plots of steady-state, time-averaged and total $k$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and $0.87$, at $Ra = 4.5 \times 10^8$. 
8.5 Steady-state simulation (Ra = 3.1 \times 10^{13})

In the previous sections, a horizontal penetration test case was investigated when Ra = 4.5 \times 10^8. In this Rayleigh number as it was shown before, the turbulence level was low. For instance, the ratio of turbulent and molecular viscosity was around 1.0 inside the tube. In this section a horizontal penetration test case is simulated by the $k$-$\varepsilon$-AWF when the Rayleigh number has the value of $3.1 \times 10^{13}$ to study the effect of higher turbulence on the flow structure.

Figures 8.32-8.34 show the flow in the central, X=0, longitudinal, y-z, plane. A comparison between the axial velocity contours at the two different Rayleigh numbers, Figures 8.6 and 8.33, shows that the increase in Rayleigh number causes the outward motion to be confined closer to the lower side of the annulus. Outside the penetration, the axial motion is similar to that at the lower Ra. Corresponding comparisons for the vertical velocity component show for this velocity component the Rayleigh number has little effect. Similar comparisons between Figures 8.7 and 8.34, show that increasing the Rayleigh number has an even weaker effect on the normalised temperature. Comparisons of the turbulent kinetic energy contours, Figures 8.7 and 8.34, show that again within the penetration, there is very little turbulence generation along the surface of the cold tube, and as also noted in the lower Ra case, most of the turbulence is generated by the downward motion along the vertical wall, below the penetration opening. The normalised $k$ levels, however, increase by an order of magnitude as the Rayleigh number increases from $4.5 \times 10^8$ to $3.1 \times 10^{13}$. The spanwise, U, velocity comparisons, Figures 8.5 and 8.32, show very little effect of Rayleigh number, while contours of the pressure deviation from hydrostatic conditions, Figures 8.6 and 8.33, show that at the higher Rayleigh number within the penetration the positive deviation occupies a greater proportion of the lower half, but the overall variation is the same at both Ra values.

Figures 8.35-8.36 show the contour plots of U, V, W, $P''$, $k$ and T inside the penetration, within cross-sectional planes normal to its axis, at locations Z=0.23, 0.46, 0.69 and 0.87. Comparisons of mean velocity contours, Figures 8.9 and 8.35, reveal a stronger Rayleigh number effect. Near the closed end of the penetration, Z of 0.23 and 0.46, the contours of the vertical velocity are
more symmetric at the higher Rayleigh number, which suggests that there is less large-scale unsteadiness. The axial velocity contours indicate that at the higher Rayleigh number, the outward motion along the bottom side of the penetration is stronger, a trend which is consistent with the stronger buoyancy force at the higher Ra level. The normalised spanwise velocity component is generally lower at the higher Ra value, which is consistent with the presence of more symmetric conditions. The corresponding mean temperature comparisons, Figures 8.10 and 8.36, show that the overall features are the same at both Rayleigh numbers, but at the higher Ra value, the size of the lower temperature region at the bottom half of the cross-section is reduced. Since an increase in Ra has been shown to cause an increase in the flow entering the penetration, it is inevitable that the cold inner pipe will have a weaker effect on the temperature of the fluid within the penetration. The contour plots of the turbulent kinetic energy, Figure 8.36, confirm earlier finding, that within the penetration turbulence is generated around surface of the inner pipe and then it is convected downwards. Turbulence levels are also shown to increase closer to the exit of the penetration. The normalised deviation from the hydrostatic pressure is also not significantly affected by the change in Rayleigh number.

Figure 8.36 shows turbulence contour plots inside the cavity at Z=0.23, 0.46, 0.69 and 0.87. As it was expected turbulence level is larger compared to lower Ra case. They also show that turbulence generated underneath the cold tube penetrated further down compared with the lower Ra case. Figure 8.36 shows contour plots of $P'$ and temperature. $P'$ implicates to deviation from the hydrostatic pressure. They show that the maximum deviation from the hydrostatic pressure occurs at the lower half of the tube which temperature is slightly reduced due to existence of the cold tube at the middle. The reason for slight temperature change is that the cooling power is not enough to make significant thermal changes inside the domain.

Figures 8.37-8.39 show the contour plots of U, V, W, $P'$, k and T outside the penetration for the higher Ra case. Outside the penetration, as seen in Figure 8.37, there is strong downward motion in the middle region, which
reduces towards the sides and also with distance from the exit of the penetration. Comparison with the corresponding contour plots at the lower Rayleigh number of Figure 8.11, shows that the main difference is that in the higher Ra case, the downward motion diminishes more rapidly with distance from the penetration exit. Contour comparisons for the normalised spanwise velocity component, $U$, Figures 8.37 and 8.11, show that the change in Rayleigh number has little effect on the weak spanwise velocity field outside the penetration. The most detectable difference being the higher levels near the penetration exit, at the higher Ra case. A similar conclusion can be reached about the Rayleigh number effects on the axial velocity outside the penetration, from Figures 8.38 and 8.12. One difference is that the axial velocity near the open end of the penetration increases more substantially with Rayleigh number. As already seen at the lower Ra case, conditions outside the penetration are practically isothermal, Figure 8.39, which is why there is very little deviation from the hydrostatic conditions, Figure 8.38. The corresponding plots of $k$ contours, Figure 8.39, show that in common with the lower Ra case, Figure 8.7, the generation of turbulence occurs mainly within the boundary layer that develops along the vertical wall, but in contrast to the lower Ra case, it is confined to a narrow central region below the penetration exit.

Figure 8.40 shows the contour plots of $V$, $T$ and $P^\circ$ within off-centre longitudinal planes, $y-z$, at locations $X$ of 0.25 and 0.5. Comparison with the corresponding plots of Figure 8.14, for the lower Ra case, show that the thermal and pressure fields remain isothermal and hydrostatic respectively, but while at the lower Rayleigh number the vertical velocity is higher near the symmetry plane and drops along the vertical wall, at the higher Ra the vertical velocity is uniform within both longitudinal planes. At both Ra values the downward flow becomes weaker with distance from the centre.

The plots of Figure 8.41, of the vertical velocity and $k$ contours within horizontal planes outside the penetration and comparisons with the corresponding low Ra plots of Figure 8.16, confirm the earlier observations on the downward motion, while for the turbulent kinetic energy, they indicate that this is mainly generated at the wall region below the open end of the penetration.
Figure 8.32 – Contour plots of U and V within longitudinal, y-z, plane, at X=0, at Ra = 3.1 × 10^{13}. 
Figure 8.33 – Contour plots of $W$ and $P'$ within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.34 – Contour plots of $T$ and $k$ within longitudinal, $y-z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.35 – Contour plots of U, V and W within cross-sections normal to the axis of the penetration, at Z=0.23, 0.46, 0.69 and 0.87, at Ra = 3.1 \times 10^{13}.
Figure 8.36 – Contour plots of $k$, $P''$ and $T$ within cross-sections normal to the axis of the penetration, at $Z=0.23, 0.46, 0.69$ and $0.87$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.37 – Contour plots of $U$ and $V$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.38 – Contour plots of $W$ and $P'$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.39 – Contour plots of $T$ and $k$ within cross-sections normal to the axis of the penetration, at $Z=0.88$, 0.92 and 1, at $Ra = 3.1 \times 10^{13}$.
Figure 8.40 – Contour plots of $V$, $T$ and $P^*$ within longitudinal planes outside the penetration at the off-centre locations $X=0.25$ and 0.5, at $Ra = 3.1 \times 10^{13}$.
Figure 8.41 – Contour plots of $V$ and $k$ within horizontal planes outside the penetration, at locations $Y=0.22$ (inlet), $Y=-0.11$ and $Y=-0.78$ (outlet), at $Ra = 3.1 \times 10^{13}$. 
8.6 Time-dependent simulation \((\text{Ra} = 3.1 \times 10^{13})\)

Figures 8.42-8.56 show the solution of 3D time dependent simulation of the horizontal penetration test case at the higher Ra using the \(k-\varepsilon\)-AWF. In the steady-state simulation, it has been observed that results are not completely symmetrical, specially inside the tube away from the open end of the tube. That is why time dependent calculations have been carried out for this case, to reach a better understanding how flow develops in the horizontal penetration.

In the time dependent calculations, the normalized time step is set equal to 0.19, normalized by \(\frac{D_2}{V_0}\) (where \(D_2\) is the diameter of the tube and \(V_0 = \sqrt{g\beta \Delta \Theta D_2}\)) and the implicit scheme is employed for time discretisation. In each time step, it is set to iterate 20 times unless it gets to convergence criteria (all residuals below \(10^{-3}\)).

Figures 8.42-8.45 show the velocity contours within the central, \(X=0\), longitudinal (y-z) plane. For the vertical, \(V\), velocity component, Figure 8.43, as was also the case at the lower Ra number, Figure 8.18, the non-uniformity of the contours suggests the presence of large-scale flow structures, which change with time, both within and outside the penetration. It is difficult to detect a Rayleigh number effect from these comparisons. By contrast, the corresponding comparisons for the axial velocity, Figures 8.44 and 8.19, show that at the higher Ra the axial velocity field shows no signs of unsteadiness whereas at the lower Ra, the effects of large-scale unstable structures are clearly evident. This leads to the conclusion that at the higher Ra, the unstable flow structures are less influential.

Figures 8.46-8.51 show contours of the vertical and axial velocities and of the turbulent kinetic energy within a number of cross-sections inside the penetration, at \(Z=0.23, 0.46, 0.69\) and 0.87 and at different times. The contours of the instantaneous vertical velocity, in common with those at the lower Rayleigh number of Figures 8.21 to 8.24, show flow oscillations within the penetration, but perhaps with smaller amplitude. While, also in common with the lower Ra behaviour, these oscillations considerably weaken at the exit, \(Z=0.87\), they can still be detected, but confined to the region below the inner pipe.
The contours of the instantaneous axial velocity within the penetration, Figures 8.46 to 8.49, show that at the exit, \( Z=0.87 \), as also found at the lower Rayleigh number shown in Figures 8.21 to 8.24, the axial flow field is reasonably symmetric, but the inward and inevitably outward flow are stronger than those at the lower Ra. The temporal variation of inflow and outflow at the exit plane is also greater at the higher Ra case. Inside the penetration, in common with what can be observed at the lower Ra case, both the strength of the axial motion and its temporal variation, start to diminish. While for the lower Ra case the bulk of the inward (negative) flow is deflected towards the upper wall of the penetration at the \( Z=0.69 \) and 0.46 locations, at the higher Ra the bulk of the inward motion remains at the middle of the penetration, possibly because of the greater momentum of the inward motion.

The plots of the contours of the instantaneous turbulent kinetic energy within the penetration have also been included in Figure 8.46 to 8.49. It should be clarified that these are plots of only the modelled component, which is related to the smaller-scale eddies. Within the penetration, \( Z=0.23 \) and 0.69, small-scale turbulence is only generated around the surface of the cold inner tube and it is subsequently convected downwards by the oscillating natural convection motion. At the penetration inlet there is a more extensive generation of turbulence of turbulence in the upper half of the penetration, most likely caused by the inflow. Like the axial flow, the distribution of the instantaneous \( k \) also shows strong temporal variation at the exit plane. In comparison to the corresponding plots at the lower Ra, of Figures 8.21 to 8.24, the main difference is that at the higher Ra, normalised turbulence levels at the open end of the penetration are substantially higher, reflecting the fact that the flow in and out of the penetration increases with Rayleigh number.

The oscillatory nature of the flow over the entire length of the penetration is also demonstrated through the vector plots of Figures 8.50 and 8.51. In common with the corresponding plots of Figures 8.25 and 8.26, they show that the cold inner cylinder, by cooling the fluid around it, generates jet of cold fluid
leaving the lower surface of the inner cylinder which both spreads and oscillates in the spanwise, \( x \), direction as it moves downwards. The main effect of the Rayleigh number is to enhance the motion within the cross-sectional planes.

Finally the question of how influential the large-scale unsteadiness is to the flow and thermal development, is addressed through Figures 8.52 to 8.56 and comparisons with those for the low Ra case in Figures 8.27 to 8.31. Mean velocity comparisons for the central longitudinal plane, Figures 8.52 and 8.53 for the high Ra and 8.27 and 8.28 for the low Ra show that at the higher Ra, while differences between the time-averaged and steady state computations are present, these are smaller than those at the lower Ra value. Finally Figures 8.54 to 8.56 present similar comparisons at number of locations within the penetration, for the vertical and axial velocity and also for the turbulent kinetic energy. The corresponding comparisons for the lower Ra case were presented in Figures 8.29 and 8.30. As far as the vertical velocity field within the penetration is concerned, the time-averaged predictions are again close to those of the steady RANS computations and at locations \( Z=0.23 \) to 0.69, the differences between the time-averaged and steady computations are notably smaller than for the corresponding lower Rayleigh number comparisons. This suggests that at the higher Rayleigh number case, though present, the large scale flow oscillations within the penetration are not as influential. This conclusion, however, is only partially confirmed by the comparisons between the time-averaged and steady flow predictions of the axial flow field, shown in Figure 8.55. In terms of the distribution of the axial velocity, the time-averaged and steady flow predictions are in close agreement over the entire length of the penetration. When it comes to the level of the axial velocity on the other hand, the steady RANS predictions return a stronger axial flow within the penetration than the time-averaged unsteady RANS. So for at least one important parameter the use of unsteady RANS does make a difference.

In the final figure, Figure 8.56, the focus is on how the use of unsteady RANS affects the prediction of the turbulence field. As in the lower Rayleigh number case, Figure 8.31, for the unsteady RANS, both the modelled and the
total turbulent kinetic energy are included. In contrast to what was observed at the lower Ra level, there is now little difference between the modelled and total levels of the time-averaged predictions, or between the time-averaged and the steady flow predictions of the turbulent kinetic energy. This means that the resolved turbulence, due to large-scale oscillations, is less significant at this higher Rayleigh number, as a result of which, use of unsteady RANS has only a small effect on the predicted turbulence field.

In conclusion therefore, increasing the Rayleigh number, as expected, enhances the axial motion within the penetration and also increases turbulence levels. While large-scale flow unsteadiness becomes weaker at higher Ra levels, its effects on the mean flow development do not entirely disappear. Use of unsteady RANS therefore is still advisable. The large-scale flow unsteadiness is weaker because turbulence is higher which means mixing is stronger. This, in turn, makes the flow more uniform. This decreases large-scale unsteadiness compared with the lower Ra case.

Finally, it is worth to mention that currently a project is being carried out that is providing LES data for this case which can give an indication about the reliability of the steady-state and time-dependent computations.
Figure 8.42 – Contour plots of $V$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.43 – Contour plots of $V$ at different time steps within longitudinal, $y$-$z$, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.44 – Contour plots of $W$ at different time steps within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.45 – Contour plots of W at different time steps within longitudinal, y-z, plane at X=0, at Ra = 3.1 × 10^13.
Figure 8.46 – Contour plots of V, W and k at different time steps within cross-sections normal to the axis of the penetration, at Z=0.23, at Ra = 3.1 \times 10^{13}.
Figure 8.47 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.46$, at $Ra = 3.1 \times 10^{13}$.
Figure 8.48 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.49 – Contour plots of $V$, $W$ and $k$ at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.87$, at $Ra = 3.1 \times 10^{13}$.
Figure 8.50 – Vector plots at different time steps within cross-sections normal to the axis of the penetration, at Z=0.23 and 0.46, at $Ra = 3.1 \times 10^{13}$.
Figure 8.51 – Vector plots at different time steps within cross-sections normal to the axis of the penetration, at $Z=0.69$ and 0.87, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.52 – Contour plots of time-averaged and steady-state $V$ within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.53 – Contour plots of time-averaged and steady-state $W$ within longitudinal, y-z, plane, at $X=0$, at $Ra = 3.1 \times 10^{13}$. 
Figure 8.54 – Contour plots of time-averaged and steady-state \( V \) within cross-sections normal to the axis of the penetration, at \( Z=0.23, 0.46, 0.69 \) and 0.87, at \( Ra = 3.1 \times 10^{13} \).
Figure 8.55 – Contour plots of time-averaged and steady-state $W$ within cross-sections normal to the axis of the penetration, at $Z = 0.23$, 0.46, 0.69 and 0.87, at $Ra = 3.1 \times 10^{13}$. 

**Time-averaged**  

**Steady state**

---

$Z = 0.23$

$Z = 0.46$

$Z = 0.69$

$Z = 0.87$
Figure 8.56 – Contour plots of steady-state, time-averaged and total \( k \) within cross-sections normal to the axis of the penetration, at \( Z = 0.23, 0.46, 0.69 \) and 0.87, at \( Ra = 3.1 \times 10^{13} \).
8.7 Closing remarks

In this chapter the numerical simulation of annular horizontal penetration by $k$-$\varepsilon$-AWF is presented at two different Rayleigh numbers ($Ra = 4.5 \times 10^8$ and $Ra = 3.1 \times 10^{13}$). The simulations are carried out with both steady-state and time-dependent methods to obtain greater understanding of the flow phenomena. The steady-state simulation of the lower Ra case showed that within the annular penetration there are irregular flow structures, which suggested existence of flow unsteadiness. The flow field is non-symmetric at the closed end of the penetration and becomes more symmetric towards the open end. Turbulence levels at this Rayleigh number are low and also that at some regions become negligible. Time-dependent simulation of the lower Ra case showed that the flow inside the penetration changes with time. The time-dependency of the flow near the closed end of the penetration is strong and it becomes weaker in the cross-sections close to the open end of the penetration. An oscillatory behaviour was observed in the downward flow inside the penetration which separates from the cold tube. The time-averaged axial velocity is stronger than the axial velocity resulting from the steady-state simulation. This is of course an important characteristic of this flow and these comparisons suggest that, for this Rayleigh number, it is necessary to employ a time-dependent RANS approach to reproduce it reliably.

The steady-state simulation of the higher Ra case showed that the normalised $k$ levels increased by an order of magnitude. Downward velocity near the closed end of the penetration are more symmetric at the higher Rayleigh number, which suggests that there is less large-scale unsteadiness. The axial velocity at the higher Rayleigh number is stronger due to the stronger buoyancy force at the higher Ra level. The time-dependent simulation of the higher Ra case showed that the instantaneous vertical velocity, in common with those at the lower Rayleigh number, has oscillatory pattern within the penetration, but perhaps with smaller amplitude. The time-averaged predictions of vertical velocity within the penetration are close to those of the steady RANS computations. Distribution of time-averaged and steady-state axial velocity are in close agreement over the entire length of the penetration but the steady RANS predictions return a stronger axial flow within the penetration than the
time-averaged unsteady RANS. Due to importance of this parameter the use of unsteady RANS does make a difference.
Chapter 9

Conclusions and Future Works

9.1 Conclusions

The research presented in this thesis has followed two main objectives. The first objective is to find how effective is the AWF, which has been recently developed in UMIST and then the University of Manchester, in the computation of buoyancy-induced flows. The second objective is to test different models for the approximation of turbulent stresses and turbulent heat fluxes for simulation of buoyancy-driven flows. Once these two objective were achieved, an appropriate modelling strategy was chosen to simulate buoyant flow in an annular horizontal penetration, with a cold inner core. Therefore, five test cases have been selected to cover a wide variety of buoyancy-induced flows. The first set of test cases was of tall cavities with different angles of inclination whose two opposite long sides are maintained at different temperatures. The angles of inclination ranged from moderate angles of $60^\circ$, with both stable and unstable heating configurations, to high inclination angles of $5^\circ$ stable and $15^\circ$ stable and unstable. In summary, the cases examined are:

- $5^\circ$ stable inclined tall cavity ($Ra = 4.16 \times 10^8$).
- $15^\circ$ stable inclined tall cavity ($Ra = 1.6 \times 10^6$).
- $15^\circ$ unstable inclined tall cavity ($Ra = 1.6 \times 10^6$).
- $60^\circ$ stable inclined tall cavity ($Ra = 0.8 \times 10^6$).
• $60^\circ$ unstable inclined tall cavity ($Ra = 0.8 \times 10^6$).

The two $60^\circ$ cases were only simulated as two-dimensional steady state flows, because the experimental data available indicate that for cavities of high spanwise aspect ratio, conditions are two-dimensional. The $5^\circ$ stable case was also simulated as two-dimensional, because the LES study which provided the validation data imposed repeating conditions in the spanwise direction, instead of simulating a three-dimensional cavity with a finite spanwise aspect ratio. The two $15^\circ$ cases were simulated as three-dimensional unsteady flows, since the experimental study reported strong evidence of three-dimensionality. Based on the experience gained from the computations of the $15^\circ$ cases, the horizontal penetration flow was simulated as both steady and time-dependent. In the annular horizontal penetration case, fluid is drawn into the annular penetration by the fact that its inner core is at temperature lower than that of the ambient fluid.

The turbulence models for the dynamic field (Reynolds stresses) used in this work were:

• $k$-$\varepsilon$ with the Launder-Sharma Low-Reynolds number extension, (denoted LRN) and high-Reynolds-number $k$-$\varepsilon$ models with the conventional, Standard Wall functions, (SWF) and the more rigorous and recently developed Analytical Wall Function, (AWF), for the near wall treatment.

• High-Reynolds-number RSM models (Basic and TCL versions) using AWF near wall treatments.

Turbulence models used in this work for the thermal field (turbulent heat fluxes) were:

• Effective diffusivity approximation.

• Generalised gradient diffusion hypothesis, GGDH.

• More elaborate algebraic expression [68], which also included the temperature variance and its dissipation rate, which in turn were obtained from separate transport equations.
60° Stable Case

Buoyancy-driven flows in the vertical and 60° stable tall cavities have very similar features, which include a turbulent core, a single cell structure, linear velocity and temperature variations across the cavity in the central region and isothermal conditions, and also a weaker velocity variation, near the end walls. The vertical and 60° stable cavities appear to show relatively few differences in the mean flow, mostly confined to the end-wall regions. According to the experimental data the end-wall effects on temperature are stronger in the 60° stable case compared with the vertical cavity. There is also a reduction in the turbulence and Nu levels in the 60° stable case compared with the vertical cavity due to the existence of more stable stratification in the 60° stable case.

All mean flow and thermal parameters are well predicted by the versions of the $k$-$\varepsilon$ model tested. The three versions of the $k$-$\varepsilon$ model return similar predictions for the mean velocity and temperature profiles, which are in close accord with the data. The predictions of the low-Re version are closest to the data, but the differences in the mean flow predictions of the models used are small. The turbulence intensity comparisons show that at the mid region all models return identical predictions which are the same for both angles of inclination and which under-predict the experimental data, though this should be expected, due to the use of the effective viscosity approximation which in a shear flow under-predicts the streamwise stress and over-predicts the cross-stream stress. The $k$-$\varepsilon$ model using LRN, AWF and SWF produced reasonable predictions of Nusselt number which were slightly lower than the experimental data.

From the data, it can be concluded that employing the more sophisticated model for the turbulent heat fluxes alongside a Reynolds stress model produces more accurate predictions of temperature variance compared to the same model for the turbulent heat fluxes used alongside an EVM. Introduction of the RSM models over-predicts turbulent kinetic energy near the top and the bottom of the tall cavity in the 60° stable inclined tall cavity. The over-prediction is significant in the case of the RSM-TCL using AWF. Near the two ends of the
cavity fluid flow impinges on the top of the cold wall and the bottom of the hot wall. It shows that the RSM is not able to accurately predict the turbulence level at the impingement regions. Then employment of the RSM-θ-ε-Basic and TCL using AWF improves the RSM-basic and TCL predictions near the top and the bottom of the tall cavity. The improvement is more significant in the RSM-θ-ε-TCL using AWF, although there is room for further improvements.

The Nusselt number comparisons show there is a good match between RSM predictions and experimental data. It also shows some improvements in Nu predictions compared with the k-ε model. The introduction of the more elaborate turbulent heat flux model, in the case of the k-ε, over-predicted local Nusselt number compared with the measured data. By contrast when used with the second-moment closure, this more elaborate thermal model has little effect on the predicted local Nusselt number, which is already in close agreement with the measurements.

60° Unstable Case

Similar to the 60° stable tall cavity, flow in the 60° unstable tall cavity exhibits very similar features with the flow in the vertical cavity, which include a turbulent core, a single cell structure, linear velocity and temperature variations across the cavity in the central region, and isothermal conditions and also a weaker velocity variation near the end walls. The thermally unstable structure of this case causes an increase in turbulent mixing and more uniform temperature distribution compared with the vertical cavity. It also increases the Nu level compared with the vertical cavity.

The comparisons of numerical predictions and experimental measurements show that the k-ε using AWF, SWF and LRN overall manages to reproduce the measured variation of the mean temperature, although the k-ε-AWF shows small discrepancies near the top and bottom of the cavity. The mean temperature profiles predicted by the RSM-Basic are not as close to the measured profiles as those of the EVM models. The Nusselt numbers that were predicted
using the $k-\varepsilon$-AWF are in close agreement with the data. Introduction of the RSM/TCL model leads to stronger discrepancies between the predicted and measured profiles of the mean temperature. Near the two ends of the cavity, due to the existence of a single circulation cell, flow impinges onto the top of the cold wall and the bottom of the hot wall. To obtain an accurate simulation of turbulent flow in this test case, a turbulence model needs to correctly capture this important feature of the flow. In the case of simulations using the RSM, it seems that the terms in the Reynolds stress transport equations which are responsible for damping of wall-normal stress near the walls and re-distributing it to the wall-parallel components do not produce accurate predictions. The above mentioned terms are tuned by solving flow over a flat plate. Near the two ends of the cavity, the flow impinges with two walls. Moreover, in the region, there is flow impingement. Due these differences, these terms may not work properly in this test case [69].

Inclusion of extra transport equations of $\theta^2$ and $\varepsilon_\theta$ and use of a more elaborate algebraic equation for the turbulent heat fluxes can improve the mean temperature prediction near the top and bottom of the cavity compared with the $k-\varepsilon$-AWF results. By employing the RSM-$\theta^2$-$\varepsilon_\theta$-AWF, the prediction of temperature variances are improved compared with $k-\varepsilon$-$\theta^2$-$\varepsilon_\theta$-AWF results, near the mid-height of the cavity. Introduction of more advanced models than the $k-\varepsilon$ did not produce improvements in the Nu predictions.

5° Stable Case

The 60° stable and unstable cases are characterised by a single-cell structure and a turbulent core, while the 5° stable cavity, on the other hand, displays a different behaviour, with the recirculation cell largely confined to the middle third of the cavity and having an inclination angle different to that of the cavity. The 5° stable case is dominated by strong laminarisation as a result of stable stratification.

In the 5° stable case, the $k-\varepsilon$ using LRN, AWF and SWF return predictions close to the LES data. The mean temperature, mean wall-parallel velocity and
Nusselt number predictions matched very well with the LES data even though the turbulent kinetic energy comparisons on the other hand, appear to show a very different picture. The reason is that in this case the turbulence level is very low and consequently the contribution of turbulence to the mean temperature, mean velocity and Nusselt number is low as well. That is why all turbulence models returned well-predicted mean temperature, mean velocity and Nusselt numbers.

15° Unstable Case

The 15° unstable case has provided a challenging test for the turbulence models. The experimental data show that, as a result of the unstable stratification, the flow breaks down into longitudinal vortices which extend over the entire cavity and also into recirculation cells. These features lead to strong mixing which causes the core of the cavity to become practically isothermal and the wall parallel velocity becomes very low, in comparison to the 60° cases. The strong mixing also causes stronger thermal energy transfer and, in turn, higher Nu levels at the hot and cold walls compared with the 60° cases.

Then 3D time-dependent simulation using the $k-\varepsilon$-AWF was employed to simulate this test case. The comparisons showed very close agreement between time-averaged numerical and experimental data. They generally showed steep wall-normal temperature gradients along the thermally active walls and practically isothermal conditions in the core, where the fluid temperature is close to the average of the hot and cold side temperatures. This suggests the presence of strong mixing throughout the cavity. The comparisons showed that the 3D time-dependent computation returned the correct low magnitude of the wall-parallel component of the mean velocity in agreement with the experimental data. This resulted in a significant improvement compared with the 2D simulations. The comparisons show that the time-averaged normal stresses resulting from the $k-\varepsilon$-AWF are higher than those of the experimental study. The over-prediction of the velocity fluctuations compared with the experimental data may be due to the modeled part of the time-averaged velocity fluctuations. Cooper et al. [8] argues that, assuming symmetric behaviour in
the spanwise direction, suggests the presence of four longitudinal recirculation cells within the plane parallel to the thermally active sides, which extend over the entire length of the cavity. The predicted profiles, with the exception of the region near the lower end-wall, Y=0.1, are reasonably close to the data. There is close agreement between the predictions and the measurements, both in terms of the levels and the distributions of the Nusselt number in the longitudinal and spanwise directions, except for an over-estimation of the Nusselt number levels along the cold wall at Z=0.5 (at the middle in the spanwise direction). The comparisons also showed that the 3D URANS $k$-$\varepsilon$-AWF predictions are able to reproduce the range of dominant frequencies and their power densities. Therefore the three dimensional URANS with the $k$-$\varepsilon$-AWF is able to reproduce the most dominant feature of the flow at this angle of inclination.

The Basic version of the RSM with the Analytical Wall Function was then used to simulate the $15^\circ$ unstable inclined cavity. The comparisons showed close agreement between the experimental and numerical mean temperature. The mean velocity profiles within the x-y plane resulting from the RSM-Basic were similar to those which resulted from the use of the $k$-$\varepsilon$-AWF. This is expected because the flow in the x-y plane is weak and both turbulence models produce almost the same velocity profiles. The $k$-$\varepsilon$-AWF predicted 4 longitudinal vortices, the sizes and strength of which varied from the bottom to top of the cavity. The RSM on the other hand produced 3 longitudinal vortices, the sizes and strength of which were almost constant from the bottom to top of the cavity, a feature that is closer to experimental findings. The Basic RSM predicted normal turbulent stresses which were in closer agreement with the measurements than the corresponding $k$-$\varepsilon$-AWF profiles. This is not unexpected, since the RSM represents exactly physical processes like the production rate and the convection of each stress component, and it approximate the effects of others, like the dissipation rate and the redistribution of turbulence energy among the three directions. The comparisons showed that the RSM Nu predictions, like the $k$-$\varepsilon$ ones, were in close agreement with most of the available data, even at the central, Y=0.5, location. The comparisons showed that the time-dependent RSM well-predicted the range of dominant frequencies and their power densities compared with the experimental data, similarly with the
Conclusions and Future Works

$k$-$\varepsilon$-AWF data.

**15° Stable Case**

The experimental study [8] reported there a two-longitudinal-cell structure, with the fluid moving downwards within the centre of the cavity and upwards along the two side walls. There are also two counter rotating re-circulation cells with identical size located in the x-z plane. This three dimensional structure may be triggered because at this angle of inclination the buoyancy force is weak, and even with a spanwise aspect ratio as high as 6.8 the presence of the end walls causes the development of a three-dimensional flow. These features cause more mixing inside the cavity, which in turn leads to variations from the pure linear temperature distribution which would exist in a thermally stable flow. It also leads to augmentation of heat transfer and increase of Nu levels at the hot and cold walls.

The comparisons revealed that the 2D $k$-$\varepsilon$ computations well-predicted the thermal field, whereas they were unable to reproduce the hydrodynamic field. The $k$-$\varepsilon$ model well-predicted the turbulence level apart from in the area near the two ends of the cavity where the $k$ level was under-predicted. This is due to the existence of 3-dimensional flow conditions reported in the experimental study of the 15° stable test case. Despite these inconsistencies, there was reasonable agreement between the measured variation of the local Nusslet number along the centreline of the cold wall and that predicted in the two-dimensional computations using the $k$-$\varepsilon$-AWF. This leads to the conclusion that deviations from the two-dimensionality in the flow do not have a strong effect on the thermal characteristics. The 2D RSM-Basics predictions were similar to those of the $k$-$\varepsilon$ model, apart from improvements in the predictions of the turbulence levels near the two end walls of the cavity.

3D time-dependent simulations using the $k$-$\varepsilon$-AWF were performed to study the flow. The mean temperature comparisons showed close agreement between the numerical and experimental data. Comparisons of mean velocity showed large differences between the numerical and experimental data. The
results of the 3D numerical simulation were mostly two-dimensional. The 3D simulation using the $k$-$\varepsilon$-AWF produced similar but weaker movement near the top of the cavity, compared with the experimental data, but it was not strong enough to penetrate all the way down to the bottom of the cavity, as was seen in the experimental data. The numerical simulation also produces similar circulations at the bottom with the opposite direction. Comparisons of normal stresses showed that while near the end walls the normal stresses were severely under-estimated, over the rest of the cavity the $k$-$\varepsilon$-AWF returns the same fluctuation levels as those found in the measurements. The inconsistency is due to the existence of 3-dimensional flow conditions reported in the experimental study of the $15^\circ$ stable test case.

The version of the $k$-$\varepsilon$-AWF which included a limiter for the turbulent viscosity based on a realisability criteria was then employed to simulate the flow. The calculation without the turbulent viscosity limit produced steady-state results even in a time dependent simulation. The calculation with the turbulent viscosity limit, on the other hand, resulted in a time-dependent solution. The reason for this behaviour is that in the original model the level of turbulent viscosity was sufficient to damp out any instabilities which might have led to large-scale unsteady structure. The limiter became active in certain regions of the flow, acting to reduce the turbulent viscosity, and hence allowing such structure to develop. The turbulent viscosity limit is based on a physical constraint (Section 3.3.2). In general the flow pattern was similar to the results of the $k$-$\varepsilon$-AWF without the limiter. These comparisons revealed that the inclusion of the turbulent viscosity limit does not make significant changes to the temperature profiles. The comparisons showed that introduction of the viscosity limiter, in addition to enabling the $k$-$\varepsilon$-AWF to return unstable flow conditions, led to some improvements in the predicted time-averaged velocity field, but there were large deviations between the predicted and measured flow fields. Improvements in the predicted rms velocities were evident, most notably near the end walls, where the predicted levels were significantly higher than those of $k$-$\varepsilon$-AWF predictions without the viscosity limiter, and close to the measured values. These predictive improvements are most likely to result from the fact that the predicted rms velocities included both the modelled
Conclusions and Future Works

Component from the solution of the transport equation for $k$, which represents the contribution of the small-scale motion, and the resolved component, obtained by post processing the instantaneous velocity and which represents the contribution of the large-scale unsteady motion. This was consistent with the fact that the largest improvements in the predicted rms velocity field are observed near the two end walls, where the predicted flow is most unstable. The three-dimensional $k$-$\varepsilon$-AWF computations with the limiter returned the correct levels for the Nusselt number, but not the correct variation.

Since neither of the two versions of the $k$-$\varepsilon$ tested was successful in reproducing the three-dimensional flow features of the $15^\circ$ stable cavity, the Basic Reynolds stress model, RSM, with the analytical wall function was used to simulate the flow. Even though the computation was time-dependent, as well as three-dimensional, the solution reached steady-state conditions in the middle of the cavity, with some unsteadiness near the end walls. The comparisons showed that the Basic RSM, apart from some minor discrepancies, was able to reproduce the measured temperature field, and that introduction of the Basic RSM resulted in encouraging improvements in the flow field predictions, though not complete agreement with the experimental data. The Basic RSM returned a two-longitudinal-cell structure, with the fluid moving downwards within the centre of the cavity and upwards along the two side walls. Use of the RSM led to closer agreement between predicted and measured rms velocities than the deployment of the $k$-$\varepsilon$, but overall agreement was not as satisfactory as for the mean flow field. The Nu variation of the RSM model was in reasonable agreement with the measurements in terms of the Nu levels and variations.

Finally, it was concluded that any LES or DNS computations of strongly inclined cavities of finite spanwise aspect ratios, with both stable and unstable heating configurations need to fully resolve the spanwise direction instead of employing repeating flow boundary conditions.

Based on the study of the tall cavities at different angles of inclination, it can be concluded that for the moderately inclined tall cavity with both thermally
stable and unstable configurations, $k-\varepsilon$ using LRN, AWF and SWF produced reliable data, although minor deficiencies were observed in the cases where the SWF was used. Implementation of more advanced turbulence models (RSM-Basic and TCL) not only did not make any improvement but also increased the inconsistencies compared with the experimental data. For the highly inclined cavity with thermally unstable configuration, the 3D time-dependent simulation of the flow by the $k-\varepsilon$-AWF produces very encouraging results. It can capture important features of this complicated flow. The RSM-Basic prediction is closer to experimental measurement regarding the position and strength of longitudinal vortices as well as stress field. For the highly inclined cavity with thermally stable configuration, employment of RSM-Basic to simulate the flow produced a significant improvement in capturing the 3D flow structure compared with the experimental data, while the $k-\varepsilon$-AWF produced an almost 2D structure.

**Annular horizontal penetration**

This case involves a horizontal annular passage, closed at one end, with the inner core maintained at a lower temperature. The inner core cools the fluid around it, which makes it heavier. The heavier fluid drops out of the penetration, from the open end, and new warm fluid enters to take its place. The facts that the thermally active surface is horizontal and the lower half of the penetration has unstable temperature stratification, were expected to lead to a flow with features similar to those of the $15^\circ$ unstable inclined cavity, which is what led to the use of the EVM-AWF model in both steady state and time-dependent simulations. Time-averaged predictions (resulting from the time-dependent computations) were therefore compared with those obtained in steady state computations. The simulations were carried out for two different Rayleigh numbers ($Ra = 4.5 \times 10^8$ and $Ra = 3.1 \times 10^{13}$). In the lower Ra case, in the steady-state simulation, it has been observed that results were not symmetrical, specially inside the tube away from the open end of the tube. That was why time dependent calculations were employed for this simulation, to obtain a better understanding of how the flow develops in this horizontal penetration test case. The time-dependent numerical results showed that downward
flow which separates from the cold tube was not uniform all along the tube. The shape of the downward flow changed with time. The component of velocity along the tube was time dependent as well. Both sets of mean velocity contours showed that over most of the penetration there were strong oscillations. By the exit, however, these oscillations appeared to have died down. The comparisons between the steady state and time-dependent simulations revealed that there were noticeable differences in the two predicted flow fields in terms of the vertical velocity. In contrast to those of the steady-state predictions, the time-averaged contours of the vertical velocity were symmetric over the entire length of the penetration. Even at the exit of the annulus, where both approaches resulted in symmetric conditions, the two sets of contours had some differences. The corresponding comparisons of the axial velocity contours were perhaps of greater significance. They confirmed that the time-dependent simulation returned a stronger inward flow within the penetration than the steady state simulation. The total turbulent kinetic energy, which consists of the modelled component plus the component resolved by the time-dependent computations, was compared with the the modelled component, which essentially is the time-averaged solution of the time-dependent transport equation for the turbulent kinetic energy. Within the penetration there was a substantial difference between the total turbulent kinetic energy and its modelled component, which means that the greatest contribution to turbulence arises from the resolved, large-scale instabilities. These were, of course, important characteristics of this flow and these comparisons suggested that, at least for this Rayleigh number, it is necessary to employ a time-dependent RANS approach to reproduce it reliably.

In the higher Ra case, the contours of the instantaneous vertical velocity, in common with those at the lower Rayleigh number, showed flow oscillations within the penetration, but perhaps with smaller amplitude. In common with the lower Ra behaviour, these oscillations also considerably weaken at the exit of the penetration. The axial flow field was reasonably symmetric, but the inward and inevitably outward flow were stronger than those at the lower Ra. The temporal variation of inflow and outflow at the exit plane was also greater in the higher Ra case. As far as the vertical velocity field within the
penetration was concerned, the time-averaged predictions were again close to those of the steady RANS computations and the differences between the time-averaged and steady computations were notably smaller than for the corresponding lower Rayleigh number comparisons. This suggested that in the higher Rayleigh number case, though present, the large scale flow oscillations within the penetration were not as influential. This conclusion, however, was only partially confirmed by the comparisons between the time-averaged and steady flow predictions of the axial flow field. In terms of the distribution of the axial velocity, the time-averaged and steady flow predictions were in close agreement over the entire length of the penetration. When it came to the level of the axial velocity on the other hand, the steady RANS predictions returned a stronger axial flow within the penetration than the time-averaged unsteady RANS. So for at least one important parameter the use of unsteady RANS does make a difference. As in the lower Rayleigh number case for the unsteady RANS, both the modelled and the total turbulent kinetic energy were included. In contrast to what was observed at the lower Ra level, there was little difference between the modelled and total levels of the time-averaged predictions. This means that the resolved turbulence, due to large-scale oscillations, was less significant at this higher Rayleigh number, as a result of which, use of unsteady RANS had only a small effect on the predicted turbulence field.

9.2 Suggested Future Work

In the 60° stable and unstable cases, introduction of an RSM not only did not improve the predictions compared with the EVM’s results, but also produced more inconsistencies in comparison with the experimental data. The RSMs tested over-predicted turbulence level near the ends of the cavity where impingements occur. Further studies could, therefore be carried out to improve the RSM. It could be investigated how the process of redistribution of turbulent stresses could be improved near the impingement regions. RSM predictions for the 60° stable and unstable cases might be improved compared with the EVM after obtaining more accurate prediction of turbulent stresses near the impingement regions.
In the $60^\circ$ stable and unstable cases, implementation of more elaborate algebraic equation than the GGDH for the calculation of turbulent heat fluxes, in the RSM framework, produced predictive improvements near the two ends of the cavity. Within such a modelling framework one needs to solve extra transport equations for $\overline{\theta^2}$ and $\varepsilon_\theta$. The comparisons of $\overline{\theta^2}$ results with the experimental data showed that $\overline{\theta^2}$ was considerably over-predicted in the near-wall regions, especially near the impingement regions. Further study could therefore be carried out on the modification of the transport equations of $\overline{\theta^2}$ and $\varepsilon_\theta$ to overcome the problem. The flow field may be simulated more accurate by obtaining more accurate $\overline{\theta^2}$ and $\varepsilon_\theta$ data, especially near the impingement regions.

Three near-wall treatments were used in the present study. They were the LRN, SWF and AWF. The AWF was developed more recently than the other mentioned near-wall treatments in the University of Manchester and also called UMIST-A. Another near-wall treatment was developed at the same time in the University of Manchester which is called UMIST-N. Their concepts are similar, with the main difference being that UMIST-A solves the near-wall flow by analytical integration of the equations while UMIST-N solves it by numerical integration. A study could be carried out to test whether or not UMIST-N can produce any predictive advantage in simulation of buoyancy-driven flows compared with the UMIST-A wall treatment.

The horizontal penetration case was simulated in the present work by employing the EVM-AWF. Inside the lower half of the penetration there is thermally unstable stratification. In the $15^\circ$ unstable rectangular cavity test case, use of an RSM produced predictive improvements compared with the EVM results. It might, therefore, be useful to employ an RSM in the horizontal penetration case in order to assess how it predicts the flow structure and unsteadiness in the horizontal penetration case.

The cavities which were studied in the present research have the high longitudinal aspect ratio of 28.7. Baïri et. al.[1] experimentally investigated natural
convection within rectangular cavities with longitudinal aspect ratios as low as 0.75 and 1.5. They considered different angles of inclination from $0^\circ$ to $360^\circ$. Therefore another avenue for investigation would be to test the URANS approach further through calculation of buoyancy-driven flows within inclined cavities with these lower aspect ratios.

A further interesting question is what would happen if highly inclined cavities (unstable and stable $15^\circ$) were studied but with a spanwise aspect ratio substantially higher than the one which was examined in the present study, which was around 6.8. If the aspect ratios were higher (eg 15, 20), the question would be whether the flow exhibits a two-dimensional region or more vortices form than those found in the case with aspect ratio 6.8. These are questions that will be difficult to answer through experiments, because a huge experimental model would be needed. LES and DNS would also have problems, because of the increased grid requirements, so practically it is only possible through URANS using high-Re models and the AWF.

Given the long time-scales involved in some of the natural convection cases which were computed in the present study, it is likely that low frequency temperature fluctuations will propagate into the solid walls. This suggests another possible research direction, the generation of conjugate heat transfer data either through experiments, or LES/DNS, and the subsequent testing of RANS and URANS models.
Bibliography


[58] P.G. Huang and M.A. Leschziner. An introduction and guide to the computer code TEAM. *UMIST*.


Appendix A

Grid sensitivity - 2D test cases

Figure A.1 shows mean temperature, velocity and rms velocity fluctuations in the 60° stable cavity obtained on different grids. The computations were carried out using the $k$-$\varepsilon$-AWF. They show that all the computations produce almost identical results. Based on the comparisons 40x220 was chosen for the remaining analysis even though 20x220 and 40x180 produced almost the same solutions. The reason for selecting the slightly finer grid was to allow the same grid to be used for all the cases, including those at different angles of inclination and unstable heating configurations, where the flow structure might be more complex than in the 60° stable inclination case.

The same grid was used for the computations using SWF to assess the performance of these near-wall strategy. For the LRN near-wall treatment, similar grid sensitivity test was carried out to select a suitable grid for the computations.
Figure A.1 – Mean temperature, velocity and velocity fluctuations in the 60° stable cavity using $\kappa$-$\varepsilon$-AWF with different number of grids.