EXPERIMENTAL AND COMPUTATIONAL MODELLING OF 3-D FLOW AND BED SHEAR STRESSES DOWNSTREAM FROM A MULTIPLE DUCT TIDAL BARRAGE

A thesis submitted to the University of Manchester

for the Degree of Doctor of Philosophy

in the Faculty of Engineering and Physical Sciences

2013

PENELOPE JEFFCOATE

School of Mechanical, Aerospace and Civil Engineering
# CONTENTS

LIST OF FIGURES ........................................................................................................... 8

LIST OF TABLES ............................................................................................................... 18

NOMENCLATURE ............................................................................................................. 19

ABSTRACT ......................................................................................................................... 23

DECLARATION .................................................................................................................. 24

COPYRIGHT ....................................................................................................................... 25

ACKNOWLEDGEMENTS .................................................................................................... 26

CHAPTER 1 - INTRODUCTION ......................................................................................... 27

1.1 INTRODUCTION ......................................................................................................... 27

1.2 TIDAL BARRAGE MODELLING .................................................................................. 27

1.3 RESEARCH OBJECTIVES .......................................................................................... 28

1.4 THESIS STRUCTURE .................................................................................................. 29

2. BACKGROUND AND LITERATURE REVIEW ................................................................ 30

2.1 RENEWABLE ENERGY SYSTEMS .............................................................................. 30

2.1.1 Wind Energy .......................................................................................................... 30

2.1.2 Marine Energy ....................................................................................................... 31

2.1.3 Wave Energy .......................................................................................................... 32

2.1.4 Tidal Energy .......................................................................................................... 35

2.1.5 Tidal Stream Turbines ............................................................................................ 38

2.1.6 Tidal Barrages ....................................................................................................... 40

2.1.7 Summary ................................................................................................................ 52

2.2 EXPERIMENTATION .................................................................................................. 53

2.2.1 Open Channel Flow ............................................................................................... 53
4.1.3 Barrage with no turbine representation .......................................................... 120
4.1.4 Barrage with turbine representation .................................................................. 120
4.1.5 Simulation Details .............................................................................................. 125
4.2 DEPTH-AVERAGED MODELLING ........................................................................... 126
4.2.1 Geometry and Mesh .......................................................................................... 126
4.2.2 Modelling Physics ............................................................................................ 127
4.2.3 Time Stepping .................................................................................................. 132
4.2.4 Simulation Details ............................................................................................ 132
4.2.5 Analysis ............................................................................................................ 132

5. RESULTS- NO TURBINE REPRESENTATION ......................................................... 133
5.1 FLOW CONDITIONS ............................................................................................. 134
5.2 WATER LEVEL VARIATION .................................................................................. 135
5.3 DUCT MIDHEIGHT RESULTS ............................................................................... 140
5.3.1 Velocity Vectors .............................................................................................. 140
5.3.2 Streamwise Velocity Profiles ......................................................................... 143
5.4 DEPTH-VARYING RESULTS ................................................................................ 145
5.4.1 X-Y Velocity Vectors in Horizontal Plane ...................................................... 145
5.4.2 Streamwise Velocity Profiles ....................................................................... 152
5.4.3 Vertical Profiles of Streamwise Velocity ....................................................... 157
5.5 DEPTH-AVERAGED RESULTS ............................................................................ 158
5.5.1 X-Y Velocity Vectors in Horizontal Plane ...................................................... 158
5.5.2 Streamwise Velocity Profiles ....................................................................... 159
5.6 CONCLUSIONS .................................................................................................... 163

6. RESULTS- BULB HOUSING WITH STATORS .................................................... 164
6.1 FLOW CONDITIONS ............................................................................................. 165
6.1.1 Flow rate and inlet velocity ............................................................................ 165
6.1.2 Water Level Variation ..................................................................................... 165
7.1 FLOW CONDITIONS ......................................................................................... 228
  7.1.1 Flow rate and inlet velocity ........................................................................... 228
  7.1.2 Water Level Variation .................................................................................. 229
  7.1.3 Swirl ........................................................................................................... 230
7.2 DUCT MID-HEIGHT RESULTS ......................................................................... 232
  7.2.1 Velocity Vectors .......................................................................................... 232
  7.2.2 Streamwise Velocity Profiles ...................................................................... 234
7.3 DEPTH-VARYING RESULTS ............................................................................ 236
  7.3.1 X-Y Velocity Vectors in Horizontal Plane .................................................. 236
  7.3.2 Y-Z Velocity Vectors in Vertical Plane ....................................................... 240
7.4 COMPARISON WITH STATORS ....................................................................... 244
  7.4.1 Velocity Ratio Profiles ............................................................................... 244
7.5 COMPUTATIONAL SIMULATIONS – STATOR MODELS .................................... 248
  7.5.1 AMF_{n,0} Model – Realisable k-ε ................................................................. 248
  7.5.2 Jet Profile Model – High Swirl Coefficient, C_{FB,5} .................................... 251
7.6 COMPUTATIONAL SIMULATIONS – FAN FUNCTION ....................................... 254
  7.6.1 Flow Conditions .......................................................................................... 254
  7.6.2 Midheight Vectors ....................................................................................... 255
  7.6.3 Velocity Ratio Profiles ............................................................................... 257
7.7 CONCLUSIONS ................................................................................................. 258

8. ANALYSIS OF BED SHEAR STRESS AND SEDIMENT TRANSPORT ............... 261
  8.1 METHODOLOGY ............................................................................................. 262
    8.1.1 Bed Shear Stress ....................................................................................... 262
    8.1.2 Coefficient of Friction Assessment ............................................................ 264
    8.1.3 Threshold of Motion for Sediment Transport ............................................. 264
  8.2 NO TURBINE REPRESENTATION ..................................................................... 267
    8.2.1 Bed Shear Stress ....................................................................................... 267
8.2.2 Log-Law Layer Assessment ................................................................. 272
8.2.3 Bed Shear Stress Ratio ...................................................................... 276
8.2.4 Full-Scale Threshold of Motion ......................................................... 279
8.3 BULB HOUSING WITH STATORS .......................................................... 281
  8.3.1 Bed Shear Stress .............................................................................. 281
  8.3.2 Log-Law Layer Assessment ............................................................... 285
  8.3.3 Bed Shear Stress Ratio ..................................................................... 288
  8.3.4 Full-Scale Threshold of Motion ......................................................... 290
8.4 BULB HOUSING WITH STATORS AND ROTORS .................................. 292
  8.4.1 Bed Shear Stress .............................................................................. 292
  8.4.2 Log-Law Layer Assessment ............................................................... 296
  8.4.3 Bed Shear Stress Ratio ..................................................................... 297
  8.4.4 Full-Scale Threshold of Motion ......................................................... 299
8.5 CONCLUSIONS .................................................................................... 301

9. CONCLUSIONS AND FURTHER WORK .................................................. 303
  9.1 SUMMARY OF RESEARCH ................................................................. 303
  9.2 FUTURE WORK .................................................................................. 309

LIST OF REFERENCES ..................................................................................... 312

APPENDIX ........................................................................................................ 327
  APPENDIX 1: LA RANCE BARRAGE AND CAISSON .................................. 327
  APPENDIX 2: SEVERN CAISSON DESIGN AND SCALED TURBINE CAISSONS .................................................................................................................................................................................. 328
  APPENDIX 3: MODIFIED TANK DIMENSIONS FOR TUBE OF 0.11m DIAMETER .................................................................................................................................................................................. 331
  APPENDIX 4: BARRAGE ENGINEERING DRAWINGS ...................................... 332
  APPENDIX 5: LA RANCE TURBINE ............................................................... 338
  APPENDIX 6: TURBINE ENGINEERING DRAWINGS ...................................... 339

58, 838 words
## LIST OF FIGURES

| Figure 2.1 | Annual mean wave power in UK territorial waters  | 33 |
| Figure 2.2 | Tidal current speed around the UK at mean spring tide | 37 |
| Figure 2.3 | Ebb flow generation | 42 |
| Figure 2.4 | Water levels and processes in ebb, flood and two-way generation | 44 |
| Figure 2.5 | Tidal range variation around the UK | 47 |
| Figure 2.6 | Proposed location of Severn barrage | 49 |
| Figure 2.7 | Proposed location of Mersey barrage | 50 |

| Figure 3.1: | NDV Probe transducers | 86 |
| Figure 3.2: | ADV Probe transducers | 86 |
| Figure 3.3: | Schematic of experimental apparatus | 87 |
| Figure 3.4: | Barrage with seven empty ducts | 91 |
| Figure 3.5: | Barrage with seven empty ducts and NDV | 91 |
| Figure 3.6: | Bulb turbine with guide vanes | 93 |
| Figure 3.7: | Barrage with seven turbines with stators and Vectrino ADV | 97 |
| Figure 3.8: | Bulb turbine with stators and rotor and cross-section of friction brake | 98 |
| Figure 3.9: | Barrage with seven bulb turbines with stators and rotors | 101 |
| Figure 3.10: | Point gauge measurement of probe depth | 102 |
Figure 4.1: StarCCM+ Model 105
Figure 4.2: StarCCM+ Model with Volume Mesh 106
Figure 4.3: StarCCM+ Model with Ducts Volume Mesh 107
Figure 4.4: Cell volume 110
Figure 4.5: Adjacent cells and node locations, CD 110
Figure 4.6: Adjacent cells and node locations, UD 111
Figure 4.7: Adjacent cells and node locations, LUD 112
Figure 4.8: Bulb turbine body in StarCCM+, a) Midheight plane b) 3-D Geometry 121
Figure 4.9: Bulb turbine body mesh refinement 122
Figure 4.10: Bulb turbine body with stators in StarCCM+, a) Plan view b) View along ducts 124
Figure 4.11: SW2D tank geometry 127
Figure 4.12: Adjacent cells and node locations, QUICK 129

Figure 5.1: Experimental set-up 133
Figure 5.2: Experimental velocimeter locations 134
Figure 5.3: StarCCM+ surface pressures upstream and downstream of barrage 135
Figure 5.4: Surface height variation downstream of barrage, StarCCM+ 136
Figure 5.5: Water depth variation 137
Figure 5.6: Water depth variation with high $c_f$ 139
Figure 5.7: StarCCM+ velocity vectors at duct midheight throughout flume 140
Figure 5.8: Velocity vectors at duct midheight in downstream region,
a) Experimental b) StarCCM+

Figure 5.9: Streamwise velocity at duct midheight

Figure 5.10: Velocity vectors at 0.04 m from the bed,
a) Experimental b) StarCCM+

Figure 5.11: Velocity vectors at 0.08 m from the bed,
a) Experimental b) StarCCM+

Figure 5.12: Velocity vectors at 0.12 m from the bed,
a) Experimental b) StarCCM+

Figure 5.13: Velocity vectors at 0.16 m from the bed,
a) Experimental b) StarCCM+

Figure 5.14: Velocity vectors at 0.195 m from the bed,
a) Experimental b) StarCCM+

Figure 5.15: Streamwise velocities at 1D downstream at different vertical, z, locations

Figure 5.16: Streamwise velocities at 2D downstream at different vertical, z, locations

Figure 5.17: Streamwise velocities at 5D downstream at different vertical, z, locations

Figure 5.18: Streamwise velocities at 10D downstream at different vertical, z, locations

Figure 5.19: Streamwise velocities at 20D downstream at different vertical, z, locations

Figure 5.20: Vertical profiles of streamwise velocity vectors, StarCCM+
Figure 5.21: SW2D velocity vectors 159
Figure 5.22: Depth-averaged streamwise velocity profiles 160

Figure 6.1: Water depth variation 166
Figure 6.2: Tangential velocity direction looking downstream 167
Figure 6.3: Velocity vectors at duct mid-height in downstream region 170
Figure 6.4: Streamwise velocity profile at duct mid-height 172
Figure 6.5: Streamwise velocity at duct mid-height 172
Figure 6.6: Velocity vectors at 0.04 m from the bed 174
Figure 6.7: Velocity vectors at 0.08 m from the bed 175
Figure 6.8: Velocity vectors at 0.12 m from the bed 176
Figure 6.9: Velocity vectors at 0.16 m from the bed 177
Figure 6.10: Velocity vectors at 0.185 m from the bed 178
Figure 6.11: Velocity vectors at 1D, 2D and 5D close to the bed (0.04 m), at the duct centreline (0.075 m) and close to the surface (0.185 m) 179
Figure 6.12: Velocity vectors at 1D downstream 180
Figure 6.13: Velocity vectors at 2D downstream 181
Figure 6.14: Velocity vectors at 5D downstream 182
Figure 6.15: Velocity vectors at 10D downstream 183
Figure 6.16: Velocity vectors at 20D downstream 184
Figure 6.17: Velocity vectors at 1D, 2D and 5D downstream 185
Figure 6.18: Streamwise velocities at 1D downstream at different vertical, z, locations 186
Figure 6.19: Streamwise velocities at 2D downstream at different vertical, $z$, locations 187

Figure 6.20: Streamwise velocities at 5D downstream at different vertical, $z$, locations 188

Figure 6.21: Streamwise velocities at 10D downstream at different vertical, $z$, locations 189

Figure 6.22: Streamwise velocities at 20D downstream at different vertical, $z$, locations 190

Figure 6.23: Lift and drag imposed on fluid by inclined flat plate 192

Figure 6.24: StarCCM+ upstream and downstream lids, 
a) Pressure b) Surface Heights 194

Figure 6.25: StarCCM+ midheight velocity vectors with swirl and drag 
from flat plate theory 196

Figure 6.26: StarCCM+ midheight velocity vectors of right-hand duct wake 196

Figure 6.27: StarCCM+ velocity vectors at 1D, 5D and 10D downstream 
from duct exits 198

Figure 6.28: Midheight streamwise velocities with swirl and drag 
from flat plate theory, StarCCM+ and Experimental 199

Figure 6.29: StarCCM+ upstream and downstream lids, 
a) Pressure b) Surface Heights 202

Figure 6.30: Midheight velocity vectors with swirl and drag 
from flat plate theory, StarCCM+ 204

Figure 6.31: StarCCM+ velocity vectors at 1D, 5D and 10D downstream 
from duct exits 205
Figure 6.32: Midheight streamwise velocities with fixed body force, StarCCM+ and Experimental

Figure 6.33: Midheight streamwise velocities with fixed body force, Experimental, Standard $k$-$\varepsilon$ and Realisable $k$-$\varepsilon$

Figure 6.34: Midheight streamwise velocities with fixed body force, Experimental, Standard $k$-$\varepsilon$ and $k$-$\omega$ SST

Figure 6.35: StarCCM+ upstream and downstream lids, Pressure

Figure 6.36: StarCCM+ upstream and downstream lids, Surface Heights

Figure 6.37: Midheight velocity vectors with high swirl constant, $C_{Fb,S}$.
   a) Full flume b) Downstream of left ducts

Figure 6.38: StarCCM+ velocity vectors at 1D, 5D and 10D downstream from duct exits

Figure 6.39: Midheight streamwise velocities with fixed body force, Experimental and Standard $k$-$\varepsilon$ with high swirl constant, $C_{Fb,S}$

Figure 6.40: Bulb body with stators in StarCCM+

Figure 6.41: StarCCM+ upstream and downstream lids,
   a) Pressure b) Surface Heights

Figure 6.42: Midheight velocity vectors with vane bodies

Figure 6.43: StarCCM+ velocity vectors at 1D, 5D and 10D downstream from duct exits

Figure 6.44: Midheight streamwise velocities, Experimental, Standard $k$-$\varepsilon$ with fixed swirl and drag momentum source and with vane bodies
Figure 7.1: Water depth variation

Figure 7.2: Velocity vectors at duct mid-height downstream of barrage

Figure 7.3: Streamwise velocity at duct mid-height

Figure 7.4: Velocity vectors at 0.04 m from the bed

Figure 7.5: Velocity vectors at 0.08 m from the bed

Figure 7.6: Velocity vectors at 0.12 m from the bed

Figure 7.7: Velocity vectors at 0.16 m from the bed

Figure 7.8: Velocity vectors at 0.20 m/0.195 m from the bed

Figure 7.9: Velocity vectors at 1D downstream

Figure 7.10: Velocity vectors at 2D downstream

Figure 7.11: Velocity vectors at 5D downstream

Figure 7.12: Velocity vectors at 10D downstream

Figure 7.13: Velocity vectors at 20D downstream

Figure 7.14: Ratio of streamwise velocity to inlet velocity at 1D downstream

Figure 7.15: Ratio of streamwise velocity to inlet velocity at 2D downstream

Figure 7.16: Ratio of streamwise velocity to inlet velocity at 5D downstream

Figure 7.17: Ratio of streamwise velocity to inlet velocity at 10D downstream

Figure 7.18: Ratio of streamwise velocity to inlet velocity at 20D downstream

Figure 7.19: Midheight velocity vectors with fixed body force and

realisable k-ε model

Figure 7.20: Midheight velocity ratios, Experimental stators/rotors,

Realisable k-ε model with stators only and Realisable k-ε model

with stators/rotors

Figure 7.21: Midheight velocity vectors with high swirl constant, $C_{Fb_S}$
Figure 7.22: Midheight velocity ratios, Experimental stators/rotors, Realisable $k$-$\varepsilon$ model with stators only and Realisable $k$-$\varepsilon$ model with stators/rotors

Figure 7.23: Midheight velocity vectors with fan momentum source, a) Experiment b) StarCCM+

Figure 7.24: Midheight velocity ratios, Experimental stators only, Experimental stators/rotors and Standard $k$-$\varepsilon$ model with fan momentum source

Figure 8.1: Bed shear stress vectors, a) Experiments b) StarCCM+ c) SW2D

Figure 8.2: Bed shear stress, a) Experiments b) StarCCM+ c) SW2D

Figure 8.3: Experimental, StarCCM+ and SW2D bed shear stresses at 1D, 2D, 5D, 10D and 20D

Figure 8.4: Experimental $z^+$ values

Figure 8.5: Experimental and StarCCM+ $z^+$ profiles at 1D, 2D, 5D, 10D and 20D (with dashed lines to depict $z^+ = 30$ and $z^+ = 200$)

Figure 8.6: Experimental and StarCCM+ bed stress profiles at 1D, 2D, 5D, 10D and 20D

Figure 8.7: Ratio of bed stress magnitude from to reference bed stress, a) Experiment b) StarCCM+

Figure 8.8: Experimental and StarCCM+ ratios of bed stress to reference bed stress (with duct locations shown by thick lines)
Figure 8.9: Full-scale ratios of sand Shields parameter to critical Shields parameter, a) Experiments b) StarCCM+ c) SW2D

Figure 8.10: Bed shear stress vectors, a) Experiment b) Realisable $k$-$\varepsilon$ turbulence model c) High swirl constant model

Figure 8.11: Bed shear stresses with stators, a) Experiment b) Realisable $k$-$\varepsilon$ turbulence model c) High swirl constant model

Figure 8.12: Experimental, StarCCM+ realisable $k$-$\varepsilon$ turbulence model and StarCCM+ high swirl constant model bed shear stresses with stators at 1D, 2D, 5D, 10D and 20D

Figure 8.13: Experimental $z^+$ values

Figure 8.14: Experimental and StarCCM+ $z^+$ profiles

Figure 8.15: StarCCM+ bed stress profiles

Figure 8.16: Ratio of bed stress magnitude to reference bed stress, a) Experiment b) Realisable $k$-$\varepsilon$ turbulence model c) High swirl constant model

Figure 8.17: Experimental and StarCCM+ ratios of bed stress to reference stress

Figure 8.18: Full-scale ratios of silt Shields parameter to critical Shields parameter with stators, a) Experiment b) Realisable $k$-$\varepsilon$ turbulence model c) High swirl constant model

Figure 8.19: Normalised experimental bed shear stress vectors, a) Stators b) Stators/rotors

Figure 8.20: Normalised experimental bed shear stresses, a) Stators b) Stators/rotors

16
Figure 8.21: Experimental bed shear stresses with stators and stators/rotors at 1D, 2D, 5D, 10D and 20D

Figure 8.22: Experimental $z^+$ values

Figure 8.23: Ratio of bed stress magnitude to reference bed stress,
a) Stators b) Stators/rotors

Figure 8.24: Experimental ratios of bed stress to reference stress

Figure 8.25: Full-scale ratio of silt Shields parameter to critical silt Shields parameter, a) No turbine representation b) Stators c) Stators/rotors
### LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2.1</td>
<td>Major world tidal barrage sites</td>
<td>46</td>
</tr>
<tr>
<td>Table 3.1</td>
<td>Table of flow conditions with no turbine representation</td>
<td>89</td>
</tr>
<tr>
<td>Table 3.2</td>
<td>Table of flow conditions with bulbs and stators</td>
<td>95</td>
</tr>
<tr>
<td>Table 3.3</td>
<td>Table of flow conditions with no turbine representation, bulbs/stators and bulbs/stators/rotors</td>
<td>103</td>
</tr>
<tr>
<td>Table 6.1</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment</td>
<td>169</td>
</tr>
<tr>
<td>Table 6.2</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and swirl and drag computational model</td>
<td>195</td>
</tr>
<tr>
<td>Table 6.3</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and fixed body force computational model</td>
<td>203</td>
</tr>
<tr>
<td>Table 6.4</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and high swirl constant computational model</td>
<td>212</td>
</tr>
<tr>
<td>Table 6.5</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and vane bodies computational model</td>
<td>219</td>
</tr>
<tr>
<td>Table 6.6</td>
<td>Summary of computational results</td>
<td>225</td>
</tr>
<tr>
<td>Table 7.1</td>
<td>Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and bulb/stator/rotor experiment at 1D</td>
<td>232</td>
</tr>
<tr>
<td>Table 7.2</td>
<td>Summary of computational results</td>
<td>259</td>
</tr>
</tbody>
</table>
NOMENCLATURE

Parameters used in the thesis are defined in this nomenclature list or within the text.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Cross-sectional Area</td>
<td>(m$^2$)</td>
</tr>
<tr>
<td>$AMF$</td>
<td>Angular momentum flux</td>
<td>(N m)</td>
</tr>
<tr>
<td>$AMF_{N-D}$</td>
<td>Non-dimensionalised angular momentum flux</td>
<td></td>
</tr>
<tr>
<td>$aturb$</td>
<td>Turbine area</td>
<td>(m$^2$)</td>
</tr>
<tr>
<td>$b$</td>
<td>Channel width</td>
<td>(m)</td>
</tr>
<tr>
<td>$b_s$</td>
<td>Surface width</td>
<td>(m)</td>
</tr>
<tr>
<td>$c_{d,L}$</td>
<td>Turbine drag, lift coefficient</td>
<td></td>
</tr>
<tr>
<td>$c_{f,0}$</td>
<td>Skin friction, Reference skin friction coefficient</td>
<td></td>
</tr>
<tr>
<td>$C_p$</td>
<td>Power coefficient</td>
<td></td>
</tr>
<tr>
<td>$C_{F_{b,D}}$</td>
<td>Drag constant for momentum source</td>
<td>(N m$^{-3}$)</td>
</tr>
<tr>
<td>$C_{F_{b,S}}$</td>
<td>Swirl constant for momentum source</td>
<td>(N m$^{-4}$)</td>
</tr>
<tr>
<td>$d$</td>
<td>Grain diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>$d^*$</td>
<td>Dimensionless diameter</td>
<td></td>
</tr>
<tr>
<td>$D$</td>
<td>Duct diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Hydraulic diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>$f$</td>
<td>Frequency</td>
<td>(Hz)</td>
</tr>
<tr>
<td>$F$</td>
<td>Force per unit volume</td>
<td>(N m$^{-3}$)</td>
</tr>
<tr>
<td>$F_b$</td>
<td>Body force per unit area</td>
<td>(N m$^{-2}$)</td>
</tr>
<tr>
<td>$F_{d,L}$</td>
<td>Drag, Lift force</td>
<td>(N)</td>
</tr>
<tr>
<td>$Fr$</td>
<td>Froude number</td>
<td></td>
</tr>
</tbody>
</table>
\( g \)  Gravitational acceleration  \((\text{m s}^{-2})\)

\( h \)  Channel height  \((\text{m})\)

\( H \)  Head difference  \((\text{m})\)

\( \bar{h} \)  Mean depth  \((\text{m})\)

\( h_1 \) or \( h_{up} \)  Upstream height  \((\text{m})\)

\( h_2 \) or \( h_{down} \)  Downstream height  \((\text{m})\)

\( k \)  Turbulent kinetic energy  \((\text{m}^2 \text{s}^{-2})\)

\( k_s \)  Roughness height  \((\text{m})\)

\( L \)  Length  \((\text{m})\)

\( l_h \)  Horizontal mixing length  \((\text{m})\)

\( \dot{m} \)  Mass flow rate  \((\text{kg s}^{-1})\)

\( P \)  Pressure  \((\text{Pa})\)

\( P_t \)  Turbine power  \((\text{W})\)

\( p^* \)  Piezometric pressure  \((\text{Pa})\)

\( P^{(k)} \)  Rate of production of \( k \)  \((\text{m}^2 \text{s}^{-2})\)

\( Q \)  Discharge  \((\text{m}^3 \text{s}^{-1}) \) \((\text{L s}^{-1})\)

\( Q_t \)  Discharge through each turbine  \((\text{m}^3 \text{s}^{-1}) \) \((\text{L s}^{-1})\)

\( r \)  Radius  \((\text{m})\)

\( Re \)  Reynolds number

\( R_h \)  Hydraulic radius  \((\text{m})\)

\( s \)  Scale factor

\( S \)  Slope

\( S_0 \)  Bed slope

\( S_f \)  Friction slope
\( S_{\phi} \) \hspace{1cm} \text{Integrand}
\( t \) \hspace{1cm} \text{Time} \hspace{1cm} (s)
\( TSR \) \hspace{1cm} \text{Tip speed ratio}
\( \bar{u} \) \hspace{1cm} \text{Mean velocity} \hspace{1cm} (m \ s^{-1})
\( u' \) \hspace{1cm} \text{Fluctuating velocity in x-direction} \hspace{1cm} (m \ s^{-1})
\( U \) \hspace{1cm} \text{Velocity} \hspace{1cm} (m \ s^{-1})
\( U_{avg} \) \hspace{1cm} \text{Depth-averaged velocity} \hspace{1cm} (m \ s^{-1})
\( U_{in} \) \hspace{1cm} \text{Inlet velocity} \hspace{1cm} (m \ s^{-1})
\( U_{x,y,z} \) \hspace{1cm} \text{Streamwise, Cross-stream, Vertical velocity} \hspace{1cm} (m \ s^{-1})
\( U(z) \) \hspace{1cm} \text{Velocity at z co-ordinate} \hspace{1cm} (m \ s^{-1})
\( U_{\theta} \) \hspace{1cm} \text{Tangential velocity} \hspace{1cm} (m \ s^{-1})
\( u_{r} \) \hspace{1cm} \text{Friction velocity} \hspace{1cm} (m \ s^{-1})
\( v' \) \hspace{1cm} \text{Fluctuating velocity in y-direction} \hspace{1cm} (m \ s^{-1})
\( V \) \hspace{1cm} \text{Volume} \hspace{1cm} (m^3)
\( x \) \hspace{1cm} \text{Streamwise distance, along channel} \hspace{1cm} (m)
\( y \) \hspace{1cm} \text{Cross-stream distance, across channel} \hspace{1cm} (m)
\( z \) \hspace{1cm} \text{Depth} \hspace{1cm} (m)
\( \Delta z \) \hspace{1cm} \text{Head difference} \hspace{1cm} (m)
\( z_0 \) \hspace{1cm} \text{Bed height} \hspace{1cm} (m)
\( z^+ \) \hspace{1cm} \text{Distance from the wall normalised using the viscous length-scale}
\( \alpha \) \hspace{1cm} \text{Angle} \hspace{1cm} (°) \hspace{1cm} (rad)
\( \beta \) \hspace{1cm} \text{Ratio of horizontal to vertical mixing length}
\( \gamma \) \hspace{1cm} \text{Elder constant}
\( \Gamma \) \hspace{1cm} \text{Circulation} \hspace{1cm} (m^2 \ s^{-1})
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Gamma_{N-D}$</td>
<td>Non-dimensionalised circulation</td>
<td></td>
</tr>
<tr>
<td>$\Gamma^{(\epsilon, (k))}$</td>
<td>Diffusivity of $\epsilon/k$</td>
<td>(m$^2$s$^{-1}$)</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Dissipation rate of turbulent kinetic energy</td>
<td>(m$^2$s$^{-3}$)</td>
</tr>
<tr>
<td>$\eta_t$</td>
<td>Turbine efficiency</td>
<td></td>
</tr>
<tr>
<td>$\theta$</td>
<td>Angle</td>
<td>($^\circ$) (rad)</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>von Karman’s constant</td>
<td></td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Boundary layer constant</td>
<td></td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>(kg s$^{-1}$ m$^{-1}$)</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulent dynamic viscosity</td>
<td>(kg s$^{-1}$ m$^{-1}$)</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>(m$^2$s$^{-1}$)</td>
</tr>
<tr>
<td>$\nu_t$</td>
<td>Turbulent kinematic viscosity</td>
<td>(m$^2$s$^{-1}$)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>(kg m$^{-3}$)</td>
</tr>
<tr>
<td>$\rho_s$</td>
<td>Sand density</td>
<td>(kg m$^{-3}$)</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Torque</td>
<td>(N m)</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Bed stress tensor</td>
<td>(N m$^2$)</td>
</tr>
<tr>
<td>$\tau_b$</td>
<td>Bed shear stress</td>
<td>(N m$^2$) (Pa)</td>
</tr>
<tr>
<td>$\tau_c$</td>
<td>Critical bed stress</td>
<td>(Pa)</td>
</tr>
<tr>
<td>$\tau_f$</td>
<td>Full-scale bed shear stress</td>
<td>(N m$^2$) (Pa)</td>
</tr>
<tr>
<td>$\tau_{visc}$</td>
<td>Viscous shear stress</td>
<td>(Pa)</td>
</tr>
<tr>
<td>$\tau_{turb}$</td>
<td>Turbulent shear stress</td>
<td>(Pa)</td>
</tr>
<tr>
<td>$\tau^*$</td>
<td>Shields parameter</td>
<td></td>
</tr>
<tr>
<td>$\tau_{crit}^*$</td>
<td>Critical Shields parameter</td>
<td></td>
</tr>
<tr>
<td>$\phi$</td>
<td>Scalar</td>
<td></td>
</tr>
<tr>
<td>$\omega$</td>
<td>Angular velocity</td>
<td>(rad s$^{-1}$)</td>
</tr>
</tbody>
</table>
ABSTRACT

The University of Manchester

Penelope Jeffcoate

Doctor of Philosophy (PhD)

Experimental and Computational Modelling of 3-D Flow and Bed Shear Stresses Downstream from a Multiple Duct Tidal Barrage

28th June 2013

The near-field depth-varying velocities and resulting bed stresses downstream from a tidal barrage have not previously been studied. The flow through and downstream of a row of seven open draft tubes in a barrage has been investigated through laboratory experiment in a wide flume, 3-D RANS CFD simulation and 2-D depth-averaged computation. When there is no turbine representation and hence negligible swirl in the draft tubes, agreement between the experiments and 3-D modelling is shown to be good, including the prediction of an asymmetric Coandă effect. With the addition of bulb bodies and vanes creating swirl in the draft tubes the velocity profiles are changed, with increased swirl directly downstream from the draft tubes and throughout the entire flume cross-section further from the barrage. The addition of rotors did not significantly alter the flow field patterns, for the stator/rotor combinations presented here. 3-D CFD could not accurately predict the velocity profiles resulting from the swirl in the ducts. The experiments and 3-D model shows that bed shear stress can be magnified markedly near the barrage particularly where the jets become attached to the bed. At full-scale this would result in a fully mobile bed with sand of typical grain size 1mm. One aim was to determine the distance downstream where depth-averaged modelling gives reasonable prediction and this is shown to occur around 20 tube diameters (20D) downstream of the barrage. Upstream of this, the depth-averaged modelling inaccurately predicts water level and bed shear as well as the 3-D flow field. The addition of swirl cannot be modelled using 2-D modelling, but by 20D downstream there is minimal velocity variation transversely and throughout the depth, indicating depth-averaged modelling would be applicable from this distance.
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.
1. 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.

2. 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.

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

4. 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://documents.manchester.ac.uk/DocuInfo.aspx?DocID=487), in any relevant Thesis restriction declarations deposited in the University Library, The University Library’s regulations (see http://www.manchester.ac.uk/library/aboutus/_regulations) and in The University’s policy on Presentation of Theses.
ACKNOWLEDGEMENTS

First of all I would like to thank my supervisors, Prof Peter Stansby and Dr David Apsley, for their guidance, advice and endless editing. It has been a pleasure to work with them and I have appreciated all the help that they’ve given me. Funding from the EPSRC is also gratefully acknowledged.

I would also like to thank the technical staff in the School of MACE, particularly Phil Oakes, Paul Townsend, Bob Wroe, Bill Storey and Alex Williams, for all their assistance in the design and construction of the barrage and turbines.

I’d like to thank some of the other people in our department for their advice, feedback and help. Dr Tim Stallard, Dr Ben Rogers, Dr Alistair Revell, Dr Nicolas Chini, Dr Alex Skillen, Dr James McNaughton, Dr Sarah Bellew and Alex Olczak have all helped me over the past three and a half years, even though there was no requirement to, so thank you.

My thanks to my friends and family for putting up with me through the ups and downs of my PhD, particularly during my many months of writing where they saved my sanity.

And finally, special mention goes to my proof readers – my Mum, Prof Nadina Lincoln, and Jack Banks - for not baulking when faced with 300 pages of a subject they’ve never encountered before!
CHAPTER 1 - INTRODUCTION

1.1 INTRODUCTION
This thesis details work carried out to determine the flow fields downstream from a tidal barrage and the resulting bed stresses, the applicability of depth-averaged modelling for near-field flow simulations and the ability to model the tidal barrage flow using a commercially available 3-D CFD modelling package. This work will be useful for tidal energy development, particularly for determining the effects of barrage installation for site selection and the most appropriate method for modelling the flow downstream of the barrage. The need for this research will be detailed in this Introduction and in further depth in the Literature Review.

1.2 TIDAL BARRAGE MODELLING
Tidal barrages have been proposed as a potential renewable energy device since the 1960s and barrages have been installed in France, Russia, China and Canada (Hammons 1993). However, despite many proposals for barrages in the UK, particularly in the Severn and Mersey estuaries, the objections against barrage installation has always prevented their development. The development of tidal barrages has been restricted by various factors. These are the high installation costs, because the barrage must be built in one unit unlike cheaper multiple unit devices such as wind farms or tidal stream turbines; high installation challenges, due to the blocking of an entire channel in strong tides and high tidal ranges; and environmental impact, due to the impediment to sediment motion and fish migration, the reduced tidal flow rates and reduced tidal range. There has been extensive modelling regarding the effect of a tidal barrage on the
sea levels and the resulting loss of inter-tidal mud flats, but the three-dimensional flow in the near-field has not been investigated. Within the near-field there are jet effects from the barrage culverts and swirl effects from the tidal turbines. These result in large scale vertical variations in velocity throughout the estuary, which affect bed stress patterns and sediment motion. Previous barrage modelling has only been two-dimensional, using the depth-averaged approximations, so these effects have not been investigated. 3-D analysis could lead to better understanding of the near-field barrage flow, the resulting bed stresses, the applicability of depth-averaged modelling and the ability to better model potential barrage sites.

1.3 RESEARCH OBJECTIVES

From this introduction it can be seen that there are several issues regarding tidal barrages that have not been fully investigated:

- The three-dimensional nature of the flow field downstream from a barrage;
- The changes to the bed stresses and, thus, sediment transport;
- The applicability of depth-averaged modelling for velocity, bed stress and sediment transport analysis.

Within this thesis these issues are addressed through the following research objectives:

- Investigation of the near-field flow downstream of a barrage through velocity profiles;
- Investigation of the effect of different types of stator/rotor configurations on the flow field;
• Determination of the velocity variation with depth in the flow;
• Investigation of the effect of a barrage on the bed stresses and, thus, sediment transport;
• Determination of where the flow field may be predicted by a 2-D depth-averaged model;
• Determination of whether the flow field can be accurately modelled using 3-D CFD.

1.4 THESIS STRUCTURE
The thesis is separated into nine chapters which cover background research of the subject area, methodology, results and conclusions. Chapter 2 includes a description of tides and tidal energy, with a literature review of experimental and computational research in the subject area. Chapter 3 details the experimental apparatus and methodology and Chapter 4 covers the computational models and methodology. The results are then separated into four chapters: Chapter 5 covers the flow results from the barrage model without turbine representation; Chapter 6 covers flow results from a tidal barrage with stators (guide vanes) in the ducts; Chapter 7 covers the flow results from a tidal barrage with stators and rotors (full turbine configuration); and lastly, Chapter 8 details the bed stress results. Chapter 9 contains the conclusions of the research aims and suggestions for future work.
2. BACKGROUND AND LITERATURE REVIEW

2.1 RENEWABLE ENERGY SYSTEMS
Research into renewable energy systems is growing extensively due to the increasing concern over climate change and the need to find carbon-free, sustainable energy technologies. Currently in the UK, biomass, biofuel, geothermal, solar energy and wind energy are all used in small scale applications and wind power is also used on a commercial scale. Tidal and wave power however have been utilised less due to various concerns, including high construction costs, disruption to shipping, aesthetic values and environmental impact, such as changes to sediment drift patterns and habitat alteration.

2.1.1 WIND ENERGY
Wind energy is renewable, plentiful and clean, with no harmful emissions. It is currently the most widely applied renewable energy in the UK. Currently approximately 197GW of wind turbines are installed worldwide, about 2.5% of world power usage (World Wind Energy Association 2011). An estimated 430 TWh of power is commercially viable worldwide annually; however, there are restrictions such as installation, access and efficiency, especially for offshore turbines.

Wind turbines operate by using a rotor disc to extract energy from the wind’s kinetic energy. For a given wind speed there is a speed of rotation which gives maximum power output; if the rotation speed is too low, some wind passes through the actuator disc without being disturbed. Alternatively, if the speed is too high then there is
excessive turbulence and vortex shedding, so energy is lost. The same theory is true for any turbine, whether in air or water.

Wind power has a number of major disadvantages: firstly, the visual and audio effect of onshore turbines reduces public support, especially in areas of natural beauty. Probably most importantly, the strength of the wind varies from 0 knots up to 65 knots, so the speed of the turbines and the amount of electricity being generated is constantly changing over a period of hours; these changes cannot be predicted far in advance so the energy generation is uncertain. There are also times when no electricity is being produced at all, so the supply patterns do not always meet demand patterns.

2.1.2 MARINE ENERGY

Marine energy, such as tidal and wave power, has been studied as a viable energy production system since the 1960s but no commercial scale marine energy electricity schemes have been implemented in the UK. Marine systems are now being investigated again due to the increasing need for sustainable, reliable energy sources. Wind power is not producing enough renewable energy to fulfil the UK’s future renewable energy obligations, so another system is required in addition, such as marine energy. There are several types of marine energy, including Ocean Thermal Energy Conversion (OTEC), salinity gradient power, wave power, marine current power and tidal power. OTEC uses the difference in temperature between deep and shallow water to drive a heat engine. Salinity gradient power utilises the difference in salt concentration between sea water and river water, and uses reverse electro-dialysis and pressure-retarded osmosis. The other types of power use the natural movement of water; wave power captures energy in wave motion that is created by wind and tide; marine current power use kinetic energy
in tidal flows; and tidal power uses the potential energy difference created by the tidal range. These technologies, which use the natural movement of the water, have been investigated further in Kerr 2005, because they are most likely to be used in the UK. Current proposals include implementing tidal stream turbines and wave devices off the North East coast of Scotland and Northern Ireland and several prototypes are currently operational.

### 2.1.3 WAVE ENERGY

Wave power harnesses the heave and surge of water particles created by wind passing over a water surface. Waves are generated by the wind if the wind speed is greater than the wave speed, leading to energy transfer from the wind to the waves. The wind speed also determines the wave height, although this is affected by other factors, such as the fetch, depth and topology of the seabed. The waves create oscillatory movement of the water particles throughout the water profile, though movements are greatest on the water surface; thus, wave devices can be placed in, or on, the water to convert the vertical and horizontal movement of the water particles to energy.

The total worldwide wave energy potential is very high, with the average wave power density for deep waters off the Northwest and Southwest coast of the UK reaching up to 50 – 60 MW km\(^{-1}\) (Kerr 2005) (Figure 2.1).
There are several wave energy converter (WEC) concepts that have been researched: point absorbers, attenuators, oscillating water columns and overtopping systems; these all use different characteristics of the wave to generate energy. Point absorbers, or buoys, are placed on the surface of the water and use the vertical and horizontal motion of the wave to generate energy, with power take-off systems varying from hydraulic rams to linear electrical generators. Attenuators are oriented parallel to the direction of wave propagation on the water surface and act in similar way to buoys, though they harness the wave energy over a greater distance. Oscillating water columns (OWCs) use the surge of waves to drive a turbine; the wave causes water to rise up a column placed in the water, forcing the air in the column to rise through a turbine, thus driving the turbine and generating energy. In overtopping systems, the wave breaks over the top of
the device and as the water drains off it exits through a turbine, driving a low-head hydro-turbine.

All of these concepts are currently under development and several prototypes have been installed. Development of wave energy devices in the UK is increasing, especially devices such as the Voith Hydro Wavegen, which is an oscillating water column currently installed off the island of Islay, the Pelamis, a hinged contour attenuator device being developed by Ocean Power Development, and Oyster, a hinged flap device by Aquamarine Power.

There are several advantages to wave energy devices; for example, point absorbers do not have a great impact on the close environment and sites can be selected where strong waves are fairly predictable over a short period. However, there are also major disadvantages. Firstly, like wind energy, wave production is irregular, and though the waves can be forecast with sufficient accuracy for energy yield, the amount of wave power available and the regularity of supply is inconsistent over a period of months. The amount of energy available i.e. the wave size, is also unpredictable, so how much energy can be produced in a certain time period cannot be known. There are strong regional and seasonal variations as well, with greater wave heights and storm conditions occurring in the winter and less extreme conditions occurring in the summer months. The sites with the largest waves also tend to be remote from the major electrical demand, so the cost of installing devices in these areas is high. The cost is increased by the need to build devices that can withstand the extreme environment, both mechanically from high seas and chemically from corrosion.
Wave energy converters have a role to play in energy generation in the UK and around the world. However, due to the unreliability of the waves and the amount of energy available for this method of generation additional types of marine energy device must be used for future energy generation.

2.1.4 TIDAL ENERGY

Tidal power is another form of marine energy conversion. Tidal power has several advantages over other renewable resources, such as advance predictability, reliability and consistent supply. Tidal power is a very dependable source of energy, because the tides are created by the interaction of the Sun, Moon and Earth. The tides can, therefore, be predicted accurately over periods of minutes to decades, are reliable and can produce a consistent supply due to the phase differences between different tidal barrage sites. These are desirable qualities for commercial electricity production and one that other renewable sources do not have.

There are several constituents that affect the tidal cycle; the largest effect comes from the principal lunar semidiurnal constituent, or M2 constituent, which is the interaction of the Earth’s rotation and the Moon’s orbit. The gravitational effect of the Moon is at its strongest when the distance between the Moon and the Earth is at a minimum; the tidal force, therefore, is stronger on the side of the Earth facing the Moon, resulting in the movement of ocean water across the Earth’s surface. This component of the tidal cycle is semidiurnal, so creates two high waters every day, where the tidal force is strong due to the close proximity of the Moon, and two low waters each day, where the tidal force is weak as the Moon is at its farthest. As the water rises between low and
high tide the tide is said to be flooding, and whilst falling from high to low tide, it is ebbing. The tidal cycle is also affected by the Moon’s orbit, with the tidal effect weakened when the Moon is at its apogee (the furthest point of orbit) and increased when at its perigee.

There is also a solar constituent to the tidal cycle, caused by the Sun’s position in relation to the Moon. At New Moon and Full Moon, the Sun is in alignment with the Moon, therefore combining the gravitational effect of the Sun and Moon, causing maximum tidal range called spring tide. At first and third quarter moon, the gravitational effect of the Sun and Moon counteract one another causing low tidal range, called neap tides.

The tidal cycle is also affected by the bathymetry of the ocean; areas that have narrow estuaries tend to have large tidal ranges as the flood tide is forced to accelerate and rise through a passage. This acceleration also occurs through narrow races between islands and around headlands. Figure 2.2 shows the peak flow at mean spring tides around the UK; this shows that the areas with the highest peak flows are around headlands or in channels.

There are some major advantages of tidal energy over other energy extraction methods: firstly, tides are predictable and reliable, unlike wind, wave and solar energy. Exactly how much energy can be generated and at what time can be calculated years in advance, which is essential for consistent and dependable energy supply to the grid. Secondly, tidal cycles run over a 12.45-hour period, so whether using the tidal stream velocity or
the tidal range, the generation window is long and regular, unlike wind and wave
generation. The resource is also theoretically inexhaustible, thus classifying it as
renewable, and no waste is generated. The timing of the tidal cycle also varies with
location around the UK, so when it is slack water in the South, it is high water in the
North; this, therefore, leads to a near-constant energy supply if considered nationally.

Figure 2.2: Tidal current speed around the UK at mean spring tide

There are several disadvantages associated with tidal power. Firstly, there is a high cost associated with building and installing devices, due to the hostile conditions and difficulty of installation in high velocity flows. Also, the environmental effects of tidal devices are relatively unknown. However, further research and technological advancements are bringing the economic and environmental costs down to competitive levels.

There are two main types of tidal energy device: Tidal Stream Turbines (TSTs) which utilise the kinetic energy of the tidal flow and Tidal Barrages which harness the potential energy of the tidal range. TSTs are placed in areas of peak flow, as they use the velocity of the tide to drive a turbine and generate power, similar to wind turbines. Tidal Barrages are placed in channels or estuaries where there is a large tidal range, so that a dam can be built to delay the tide and generate energy by allowing water at a higher level pass through a turbine within the barrage to the lower water level.

### 2.1.5 TIDAL STREAM TURBINES

Marine currents can be caused by tides, global oceanic circulation, seawater density variations and other factors; in the UK, currents are largely caused by the tidal effect. Tidal currents are strong enough to generate energy in areas where the flow is accelerated, either due to narrowing or shallowing of the channel. The sites with highest tidal flow occur where there are narrow straits between land masses or where the tide races around a headland, as shown in Figure 2.2. A method of harnessing this kinetic energy is to install Tidal Stream Turbines (TSTs).
2.1.5.1 TST OPERATION

TSTs are based on the wind turbine concept, with a turbine placed perpendicular to the flow, which turns due to the water passing over the turbine and the turbine blade design. The kinetic energy of the turbine is converted using a generator and transmitted back onshore. Energy extraction is very similar to wind power, however because water is 800 times denser than air, the energy that can be extracted from a wind flow of 9 m s\(^{-1}\) can be extracted from a water flow of 1 m s\(^{-1}\) (Kerr 2005). Tidal current sites with tidal velocities over 2.5 m s\(^{-1}\) are generally considered to be economically viable considering the large installation costs (Bryden and Crouch 2006; Bryden and Macfarlane 2000; Batten et al. 2007); this is because tidal devices are subjected to harsher environmental conditions than wind devices, including higher structural loads and corrosive seawater.

2.1.5.2 OPERATIONAL AND PROPOSED TSTS

There are several different types of TST, most based upon the same concept as a wind turbine, but other concepts are being developed. O Rourke (2010) details most of the promising tidal current turbines that are under development. Devices with similarities to wind and tidal barrage turbines are Atlantis AK-1000, Hammerfest Strom AS and Tidal Generation Limited detailed in Renewable UK (2011).

The first grid connected tidal current turbine was HS300, a single, three-bladed horizontal-axis turbine (HAT) developed and installed in Norway in 2003. The same company, Hammerfest Strom AS, have an agreement with Scottish Power to build new 1MW turbines, called HS1000, in Scottish waters. The Free Flow Turbine is also a three-bladed horizontal-axis turbine (HAT) being tested in New York City and it produces 1 MW h per day, 365 MW h per year. The Lunar Energy Tidal Turbine is a
HAT and is being developed in the UK. It has a 1 MW bi-directional turbine within a duct; this duct accelerates the flow, thus producing more energy.

Seaflow is a twin-rotor TST that was tested in the Bristol Channel and produced 300 kW in a flow of 2.7 m s\(^{-1}\) (Kerr 2005). SeaGen is a 1.2 MW turbine developed after the success of Seaflow by Marine Current Turbines in the UK. A trial model was installed in Strangford Lough, Northern Ireland and has operated at full power (Marine Current Turbines 2008). TidEl Stream Generator has two contra-rotating rotors. The system is also tethered to the seafloor by anchors rather than implanted into the bed.

O Rourke (2010) details the current issues with the development of tidal current turbines as: installation challenges, such as strong tidal races and short slack times; maintenance challenges, such as the ease of access; electricity transmission, because devices are offshore; loading conditions, due to the high density of seawater; and environmental impacts.

---

### 2.1.6 TIDAL BARRAGES

Unlike TSTs, a barrage uses a series of turbines and sluices to delay the flux of water as the tidal level changes, thus harnessing the tidal currents’ potential energy rather than the kinetic energy. A dam is placed across the entire estuary, so that the flow of water between the estuary and the basin can be controlled using sluice gates and ducts with turbines. This control mechanism can be used in three ways: ebb generation, flood generation and two-way (or dual) operation. Barrage operation is explained in detail in Baker (1991).
2.1.6.1 TIDAL BARRAGE OPERATION

In ebb generation, the sluice gates are left open as the tide rises, allowing the estuary and the basin to fill with water. At high tide, additional water can be pumped upstream to increase the difference in water levels (head) between the estuary and the basin; this is the filling phase of the system. The sluices are then dropped to stop the water from leaving the basin as the tide ebbs; this is the holding phase.

In order for the turbines to rotate and thus generate electricity, there must be a sufficient pressure difference between the estuary and the basin, which is created by a head difference. Once a sufficient head difference has been reached due to the ebbing tide the sluices are raised and the water passes through the ducts, driving the turbines as it ebbs from the basin. The system uses bulb turbines within the ducts; bulb turbines are propeller-type water turbines, which are installed horizontally so that they can be driven by both the pressure difference created by the head and the kinetic energy of the water flowing through the ducts. Once the tide in the estuary starts to turn i.e. it starts flooding, the sluices are dropped until the tide level reaches the same water level as the basin and the system repeats.

The period for ebb generation is approximately 5 hours, because a full tidal cycle lasts just over 12 hours, and the turbines can generate for most of the ebbing phase. The full cycle is shown in Figure 2.3. Figure 2.3 shows that the tidal range in the basin is smaller than that in the estuary, due to the holding phases of the cycle, especially at low water.
Flood generation uses the same process as ebb generation, but the rising flood tidal flow is delayed to create a head difference rather than the ebb flow. The head in the estuary, therefore, is higher than that in the basin. This also leads to a reduced tidal range, especially at high water. Figure 2.4 (Xia et al. 2009) shows all three modes of operation and the water levels in the basin and estuary at each stage i.e. filling, holding and generating. This shows that flood generation produces the least amount of energy; this is because there is a reduced bathymetry of the estuary i.e. the channel width is smaller when the turbines start generating, so fewer turbines are operational. As the tide comes in the number of operational turbines increases, but for the majority of the generation window there are fewer turbines driving than there are in ebb generation.
Two-way or dual-mode operation combines both of these methods of generation. The generation window for dual-mode operation is larger than for ebb or flood only, because there are two shorter generation windows over a tidal cycle (Figure 2.4), however the amount of energy produced is reduced, due to the altered water levels. Ebb-generation increases the low water height and flood generation reduces high water levels, thus reducing the tidal range in the basin and, therefore, creating a smaller head difference. This decrease in head difference reduces the pressure difference between the basin and the estuary and causes a smaller discharge rate through the turbines, so there is lower angular velocity and less power is produced.

The mode of operation, therefore, that will be most efficient is ebb generation. Ebb generation uses a larger bathymetry than flood generation, so more turbines are operational for a larger percentage of the generation window, plus the large head difference between the basin and estuary is retained, thus producing larger turbine velocities and increased power production.

The joint operation of several barrages at different locations means that there is a longer generation window, because the high and low tides occur at different times around the country. This means that generation at each location will occur at different times, which will produce a more consistent electricity supply. This joint operation is explained in detail in Burrows et al. (2009b).
Figure 2.4: Water levels and processes in ebb, flood and two-way generation

(Xia et al. 2009)
There are currently four operational barrages in the world: La Rance in France, Annapolis in Canada, Jangxia Creek in China and Kislaya Guba in Russia. The La Rance barrage near St Malo in France is the only large tidal power station in the world. The large tidal range of 13 m made the estuary a perfect site for barrage development and the barrage has been operational since 1966, though it was refurbished in the 1990s. The barrage can operate in both tidal directions, i.e. dual-mode production, but predominantly operates in ebb generation only, with pumping at high tide. The barrage geometry and caisson design of La Rance barrage from EDF Dossier de Presse (2009) are shown in Appendix 1. There are 24 bulb turbines of 10 MW capacity each and the system produces 544 GW h per year (Kerr 2005); this amount of electricity could supply a city of up to 300 000 people (Lowitz 2007). The barrage has also led to improved communication links by connecting two previously isolated towns, as well as increasing both trade and tourism. The estuary is also still navigable, with 20 000 boats a year travelling through the ship lock installed within the barrage (de Laleu 2009).

Kerr (2005) proposed that by introducing 17 more barrages across the world 63 TW h could be produced, whilst 80 Mt of CO\(_2\) would be saved by not using coal-fired generation. O Rourke (2010) proposed that by introducing 25 new barrages, approximately 580 TW h could be produced (Table 2.1), with the largest contributors being Minas-Cobequid, North America, White Sea, Russia and Mont St Michael, France.
Table 2.1: Major world tidal barrage sites (O Rourke 2010)

<table>
<thead>
<tr>
<th>Location</th>
<th>Mean range (m)</th>
<th>Basin area (km²)</th>
<th>Potential mean power (MW)</th>
<th>Potential annual production (GW h/ year)</th>
</tr>
</thead>
<tbody>
<tr>
<td>North America</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Passamaquoddy</td>
<td>5.2</td>
<td>262</td>
<td>1800</td>
<td>15,800</td>
</tr>
<tr>
<td>Cobequid</td>
<td>5.5</td>
<td>106</td>
<td>722</td>
<td>6320</td>
</tr>
<tr>
<td>Bay of Fundy</td>
<td>6.4</td>
<td>83</td>
<td>765</td>
<td>6710</td>
</tr>
<tr>
<td>Minas-Bahia</td>
<td>10.7</td>
<td>777</td>
<td>10,900</td>
<td>170,000</td>
</tr>
<tr>
<td>Anholt Fjord</td>
<td>10.7</td>
<td>10</td>
<td>736</td>
<td>22,180</td>
</tr>
<tr>
<td>Sherbey</td>
<td>9.8</td>
<td>117</td>
<td>520</td>
<td>6080</td>
</tr>
<tr>
<td>Cobh</td>
<td>10.1</td>
<td>75</td>
<td>1600</td>
<td>142,900</td>
</tr>
<tr>
<td>Pentlandia</td>
<td>10.7</td>
<td>31</td>
<td>794</td>
<td>6080</td>
</tr>
<tr>
<td>Meninnaesu</td>
<td>10.7</td>
<td>23</td>
<td>590</td>
<td>5170</td>
</tr>
<tr>
<td>South America</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>San Jose, Argentina</td>
<td>5.0</td>
<td>750</td>
<td>5870</td>
<td>51,500</td>
</tr>
<tr>
<td>United Kingdom</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Severn</td>
<td>9.8</td>
<td>70</td>
<td>1680</td>
<td>15,000</td>
</tr>
<tr>
<td>Mersey</td>
<td>6.5</td>
<td>7</td>
<td>110</td>
<td>1360</td>
</tr>
<tr>
<td>Solway Firth</td>
<td>5.5</td>
<td>60</td>
<td>1200</td>
<td>10,000</td>
</tr>
<tr>
<td>Thames</td>
<td>4.2</td>
<td>40</td>
<td>230</td>
<td>1400</td>
</tr>
<tr>
<td>France</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aber-Basert</td>
<td>5.2</td>
<td>29</td>
<td>18</td>
<td>158</td>
</tr>
<tr>
<td>Aber-Wrach</td>
<td>5.2</td>
<td>1.1</td>
<td>6</td>
<td>53</td>
</tr>
<tr>
<td>Argoatva</td>
<td>8.4</td>
<td>28</td>
<td>446</td>
<td>2910</td>
</tr>
<tr>
<td>Finnsay</td>
<td>7.4</td>
<td>12</td>
<td>148</td>
<td>1300</td>
</tr>
<tr>
<td>La Rosee</td>
<td>8.4</td>
<td>22</td>
<td>349</td>
<td>3060</td>
</tr>
<tr>
<td>Rochefort</td>
<td>8.1</td>
<td>1.1</td>
<td>18</td>
<td>140</td>
</tr>
<tr>
<td>Mont St Michel</td>
<td>8.4</td>
<td>616</td>
<td>9700</td>
<td>85,100</td>
</tr>
<tr>
<td>Somme</td>
<td>6.5</td>
<td>49</td>
<td>416</td>
<td>4090</td>
</tr>
<tr>
<td>Ireland</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strangford Lough</td>
<td>3.6</td>
<td>125</td>
<td>350</td>
<td>3070</td>
</tr>
<tr>
<td>Russia</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Kolaha</td>
<td>2.4</td>
<td>2</td>
<td>2</td>
<td>22</td>
</tr>
<tr>
<td>Lombocksk Bay</td>
<td>4.2</td>
<td>70</td>
<td>277</td>
<td>2430</td>
</tr>
<tr>
<td>White Sea</td>
<td>5.65</td>
<td>2000</td>
<td>14,400</td>
<td>126,000</td>
</tr>
<tr>
<td>Morecambe Estuary</td>
<td>6.6</td>
<td>140</td>
<td>270</td>
<td>12,000</td>
</tr>
<tr>
<td>Ascension</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Kimberley</td>
<td>6.4</td>
<td>600</td>
<td>630</td>
<td>5660</td>
</tr>
<tr>
<td>China</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bachalmou</td>
<td>2.4</td>
<td>No data</td>
<td>No data</td>
<td>No data</td>
</tr>
<tr>
<td>Jiangta</td>
<td>7.1</td>
<td>No data</td>
<td>No data</td>
<td>No data</td>
</tr>
<tr>
<td>Xinlanyang</td>
<td>4.2</td>
<td>No data</td>
<td>No data</td>
<td>No data</td>
</tr>
</tbody>
</table>

Burrows et al. (2009) found that the joint operation of 8 major barrages in the UK has the potential to supply 15% of the UK electricity and 5% more can be generated by tidal stream turbines. Implementing 5 barrages on the West Coast of the UK could also supply 33 TW h of electricity per year, which is approximately 10% of UK electricity demand.

For a site to be suitable for a tidal barrage system there must be a large tidal range and a fairly narrow estuary. Figure 2.5 shows the tidal range variation across the UK and it can be seen that the areas on the west coast with the highest range, and therefore highest energy potential are the Severn, Dee, Mersey, Morecambe Bay and Solway Firth. These
sites have ranges of 8-14 m and narrow estuaries suitable for dam construction, so are most suitable for barrage development.

**Figure 2.5: Tidal range variation around the UK**

There have been proposals for a tidal barrage in the Severn estuary since the 1980s. Three options have been investigated in detail: the Cardiff-Weston Barrage, the Fleming Lagoon (which does not span the entire width of the estuary) and the Shoots Barrage. Xia et al. (2009) found that the installation of the Cardiff-Weston barrage would cause a 50% decrease to upstream discharge, so would reduce mean water level in the basin. This could have a positive effect in reducing coastal flooding. The Fleming Lagoon was found to have no influence on hydrodynamic processes and the Shoots barrage was found to decrease upstream water levels, but could adversely increase downstream water levels.

The barrage that has been most widely considered is the Cardiff-Weston barrage. The proposed Severn barrage would be the largest in the UK and it is thought that it would be able to produce 17 TW h per year (Kerr 2005) because the tidal range is up to 14 m. The proposed barrage would be 16 km long, spanning from Lavernock Point to Brean Down, with 216 bulb-turbine generators, each turbine with a diameter of 9 m; the design is outlined in DEP Energy Paper 46 (1981). This barrage may be implemented in the UK, because it has the highest energy potential (Kerr 2005), therefore it was an ideal barrage to use as the basis for this study. Details of the barrage dimensions and the tidal range from the DEP Energy Paper 46 (1981) were used in this study.

There have been oppositions to the construction of a barrage in the Severn estuary by local groups and charities, due to the large scale of the project and the unknown potential environmental effects. However, Kirby (2010) describes the bed of the Severn estuary as an inter-tidal zone which is mainly mudflats, which are the source for sub-
tidal mud accumulations in various areas of the estuary. Kirby (2010) also states that due to its high turbidity the estuary is verging on becoming barren. Building a Cardiff-Weston barrage could create a rise in the abundance of fauna and species diversity. With the installation of the barrage there would be permanent settlement of sediment, leading to improved daylight penetration and dissolved oxygen concentration. This oxygenation would also be increased by the turbulent mixing of the water, both from ebb generation through the turbines and also from flood tide through the sluices. Improved water quality will lead to an increase in vertebrate fish. Flow modelling can be used to determine the extent of sediment settling and water oxygenation, by studying the turbulent flow downstream from the barrage and the sediment transport caused by the close-to-bed flow.

Figure 2.6: Proposed location of Severn barrage (Kerr 2005)
Another tidal scheme that has been proposed is the Mersey barrage, which spans the river between New Ferry and Dingle (Figure 2.7). The barrage will be 2 km long and it is estimated that it would produce 1.45 TW h a year (Kerr 2005). The tidal range is between 4 and 10 m at neap and spring tides respectively, therefore it has the second largest tidal range in the UK (Hammons 2008). The benefit of barrage construction in the Mersey instead of the Severn would be potentially less extreme environmental impacts, because the species are less diverse and abundant and the channel is narrower, so construction may be simpler.

Figure 2.7: Proposed location of Mersey barrage (Kerr 2005)
2.1.6.5 POTENTIAL DISADVANTAGES OF TIDAL BARRAGES

Tidal devices have many advantages over other technologies, for instance the predictability and reliability of supply, however there are issues that are currently hindering the development of barrages.

Firstly, high construction and installation costs make barrages less desirable than other marine energy technologies to potential developers. As with tidal current turbines the barrage construction must withstand high loads exerted by the reservoir, so a high amount of material is required, plus the barrage is packed into the seabed, leading to high construction costs. Maintenance, however, is a lesser problem than installation, due to the improvements in turbine design and ease of access. Also, the La Rance barrage has been operational since the 1960s and when refurbished in the 1990s, the blades were found to have very little signs of wear, which shows that the barrages have a long operational life.

Secondly, the environmental impact caused by barrages is relatively unknown, though the overall effect is perceived to be detrimental; this has led to little support from local residents and charities, such as the RSPB. There is currently little research into the environmental effect caused by barrages, though the La Rance barrage has led to an increased biodiversity within the basin and abundance of different species; for example, sea bass and cuttlefish have replaced sand-eels and plaice. Habitat change may have led to this change in species, possibly due to the progressive silting of the basin which has occurred; the cause of silting may be due to reduced velocities of the tides and therefore increased sediment deposition.
2.1.7 SUMMARY

The greatest impediment to the development of barrages is the uncertainty associated with the environmental impact that their construction and operation may cause. The barrage may alter flow patterns, causing changes to sediment drift and deposition, which could alter species environment and silting of the barrage. These effects could lead to species change and reduced operation of the barrage respectively. The extent of velocity change and sediment deposition is not fully known because there are few sites to analyse in the field and little in depth modelling of the area close to a barrage has been conducted. O Rourke (2010) states that “the impact of a barrage varies from site-to-site; however, there are very few projects available for comparison”.

In order to determine the impact of a barrage, modelling of the area affected by a barrage must be conducted. The velocity of the flow directly downstream of the barrage can be analysed to determine where sediment transport and sediment deposition occurs. This could in turn lead to appropriate analysis of the effect on the ecosystem and the species affected, plus the extent of silting of the estuary and effect on energy generation of the barrage. The development of a model that accurately predicts these features could then be altered for use in all potential barrage sites, rather than each site requiring individual extensive assessment.
2.2 EXPERIMENTATION

In order to create an accurate computational model that represents the barrage flow, the true flow field must be determined using experiments. Previously, there has been little experimentation on tidal barrages. The most similar type of experimentation, which is explored below, is on open channel flow, which would be similar to the estuary conditions, and water jets, since they examine water flowing from a pipe into a shallow-water reservoir. The instrumentation available for measuring the flow is also briefly investigated.

2.2.1 OPEN CHANNEL FLOW

Experimentation into estuarine flows has been conducted, for example to determine flow stratification and sediment transport. Stratification of the flow has been found to be apparent in many estuarine flows: Huang et al. (2003) found the Apalachiocha River estuary in Florida to be highly stratified in terms of salinity and current and Stephens and Imberger (1997) found the Swan River, Australia to be stratified in winter, due to excessive freshwater discharge. Circulation was also observed by Stephens and Imberger (1997) with vertical variation. Huang et al. (2003) found there was a stronger relationship between the bottom currents and the salinity gradient than there was between the bottom currents and the river flow, so the vertical variation of the flow was important for determining salinity. These flow features indicate that estuarine flows tend to be three-dimensional due to the tidal cycle influence, so three-dimensional modelling may be required for simulating tidal features.

Several studies have been conducted on the importance of the velocity distribution in predicting sediment transport have been conducted, for example Zhu and Hao (2011)
and Eyre (2009). Channel flows are found to greatly affect the sediment transport in an estuary, depending on the velocity distribution of the flow, and are an important factor to study with regards to barrages.

2.2.2 WATER JETS

Three-dimensional jets have been the subject of research for many years. Jets into unbounded quiescent water (reviewed in Lipari and Stansby 2011), as wall bounded jets, as buoyant jets with free surface interaction and both as single and multiple jets have all been studied. Circular jets and wall jets are most relevant here and have been reviewed in Law and Herlina (2002) and Pani (2012) and studied by Ead and Rajaratnam (2002), Raiford and Khan (2009), Wu and Rajaratnam (1990) and Rowland et al. (2009).

Law and Herlina (2002) found that the velocity profiles normal to the bed could be represented by that of a plane wall jet. Ead and Rajaratnam (2002) have studied plane wall jets entering shallow water. Their results showed that the velocity profiles were similar to those of deeply submerged jets. Two stages were also found for the wall jets: the first had a linear growth rate and the second stage also had a linear growth rate which was larger than the first. The depth of the jets therefore has a large effect on the velocity profile, so the full 3-D flow effects caused by a barrage must be assessed.

Rowland et al. (2009) analysed the effect that the turbulent characteristics of a shallow wall-bounded plane jet had on river mouth hydrodynamics. Jets entering still waters, such as a tributary entering an estuary, are different from most jet experiments due to
the presence of a solid bottom and a free surface. The impact of a boundary, such as the sea bed, must be investigated.

The effect of boundaries on jet flow was also shown to be significant by Johnston and Halliwell (1986) and Raiford and Khan (2009), producing less dilution than free jets. Johnston (1985) and Johnston and Halliwell (1986) examined the effects of shallow water on plane turbulent jets. They found that shallow jets, which interacted with the bed and free surface, produced less dilution than free jets, which had no boundaries affecting the flow. Since barrage ducts are typically located close to the bed, the effect of the bed may be strong and lead to less dilution and a significant Coandă effect; the Coandă effect causes a jet close to a wall to bend towards that wall due to lack of entrainment.

These four studies investigated plane jets, but the turbine has circular duct exits, so circular jets are more comparable. Pani (2012) reviewed a multiple circular free jet experiment; a relatively large centreline spacing (3D) caused jet merging by 100D downstream. Whilst flows through barrages may have similarities to previous jet experiments, the flows will differ due to the larger exit area compared to the channel cross section, proximity to the bed and the downstream flow eventually becoming almost uniform channel flow.

Raiford and Khan (2009) also investigated circular jets in shallow water, so there were interactions with the bed and the surface, unlike Pani’s experiment; this is a similar problem to that at the downstream end of the barrage. Because the mixing is affected by
the presence of boundaries, the combined influence of the free surface and the bed were investigated using a physical model study. The results showed the growth rate of the velocity profiles in the horizontal plane had two regions, whereas the growth rates in the vertical plane were linear within the near zone. The local maximum velocity decay rate was found to follow the free circular jet in the near zone and the wall or surface jet in the far zone. The velocity profiles in both the horizontal plane and vertical plane were analysed using a Prandtl tube. Madnia and Bernal (1994) studied circular jets and also found that the velocity decay was slower than in a free jet.

Wu and Rajaratnam (1990) examined circular jets, at the wall and on rough boundaries. Velocity profiles of the jets were found to be similar in both the central plane and the transverse direction. In the vertical plane the roughness affected the length scales; however, in the transverse plane the expansion was mostly unaffected. The boundary roughness also affected the shear stress at the bed; the bed shear stress results caused by the circular jets downstream from a barrage must therefore be assessed in future barrage experimentation.

The experimental results in Rajaratnam and Humphries (1984) also showed that the centreline velocity decay was slower for plane and circular surface jets than for free jets. Rajaratnam and Pani (1974) also found that the circular surface jets had a growth rate of about half that of a wall jet. Anthony and Willmarth (1992) found similar results for a round jet beneath the surface. Surface jets are very dissimilar to barrage jets, since barrage ducts are close to the bed in order to maximise generation time, so the barrage results will probably be very different from the surface jet results.
The key features of the jet experiments are that in the presence of boundaries the velocity decay is less than that of a free jet; the velocity decay and jet dilution is affected by the distance of the jet from a boundary; and that the bed stress is affected by the jets, depending on the distance of the jet from the bed. The jets caused by a barrage are circular, close to the bed and 3-D; therefore, the bed stress is expected to be significantly affected by the jets.

2.2.3 INSTRUMENTATION

Several methods have been used for measuring flow in experimental conditions. Particle-tracking velocimetry (PTV) has been used for open-channel flows (for example, Nezu and Sanjou 2011); this method uses individual particle tracking to calculate velocity vectors within a flow. However, tracking of individual particles in highly concentrated flows is particularly difficult, so the method can only be applied in low-particulate-density flows. Particle-image velocimetry (PIV) uses a laser light to trace the velocity vectors of particles in the flow and “the statistical cross-correlation of sequential narrow-window images” (Nezu and Sanjou 2011), so can trace velocity vectors much more easily; several PIV instruments and techniques are reviewed in Adrian (1991). PTV and PIV could be considered better methods than probe sampling techniques because they do not interfere with the flow, but require greater equipment.

Acoustic Doppler Velocimeters (ADVs) have been widely used, such as in Chanson et al. (2008) and Rowland et al. (2009). Chanson et al. (2008) found that the ADVs were good for measuring field estuarine flows, but post-processing was required to eliminate data spikes and noise caused by fish, birds and slack water. There were also some
erroneous results in the vertical direction, possibly due to the interference of the probe stem. The results were filtered and de-spiked, leading to an approximate mean, which showed reasonable results. The error in the results may have also been due to the low sampling frequency (25Hz) in a highly turbulent flow. Experimental velocities were measured using Vectrino ADVs in Rowland et al. (2009). The results were filtered to remove Doppler noise and minimize aliasing, using the Gaussian smoothing function detailed in Biron, Roy and Best (1995) and Lane et al. (1998), thus creating improved velocity measurements.

The velocity profiles throughout the flow can be assessed using a pitot tube or a velocimeter; however in order to obtain the velocities in all three directions a 3-D velocimeter, such as a Nortek ADV, rather than a pitot tube (Rhaiford and Khan 2009), must be used.
Vertical variation and circulation were found to be a key feature of estuarine flows, so the experimental results are expected to be 3-D. The depth of the jets affects the flow, since in shallow water flows shallow jets were found to interact with the bed and free surface (Law and Herlina 2002; Johnston 1985; Johnston and Halliwell 1986). Wu and Rajaratnam (1990) also found that circular jets affected the bed shear stress, especially those close to the bed.

Plane and circular jets have been studied in various conditions, including surface, wall and shallow water jets; however the jets have all been caused by narrow nozzles. The same principles and velocity profiles have not been examined for the jets caused by a wide nozzle (when compared with the depth), such as may occur in the flow exiting a barrage.
2.3 COMPUTATIONAL MODELLING

2-D and 3-D computational models have commonly been used to study hydrodynamic processes; several types of models and their applications are reviewed below.

2.3.1 SHALLOW WATER MODELLING

Shallow water modelling has been used extensively in analysing flows, especially in open channels. There are some key features of the barrage flow that can be seen in other studies to be significant and must therefore be addressed.

Firstly, whether the flow downstream from the barrage is steady or unsteady has a large impact on the flow results. Ball et al. (1996) used PTV to study the velocity vectors and far wake instabilities of flow that passes through pile groups in shallow water. Wake oscillation occurred downstream of the piles, but as the spacing between the piles reached eight-diameters the oscillation was intermittent, so may not occur if spacing is increased further. If the flow reaches steady state then analysis is simpler; when comparing with experiments, the results can be time-averaged rather than time-varying. Stansby (2006) found that the steadiness of the wake depended on the ratio of vertical mixing length to horizontal mixing length in subcritical flow for both a 3-D boundary layer model (Stansby 2003) and a 2-D depth-averaged model. The stability factor also affected whether the wake was vigorous and vortex shedding or steady and recirculating. The depth-averaged model in Stansby (2006) was used for further analysis, so the horizontal to vertical length scale and Froude number of the flow must be assessed to determine whether the flow is steady.
Wu (2004) also studied unsteady flow in open channels using a depth-averaged two-dimensional numerical model and also analysed the non-uniform sediment transport in open channels. One of the cases studied was Saiedi’s 1997 experiment of channel degradation, which showed scour downstream from a channel opening due to the jet which formed. The comparison between the computational result and experimental results was good, but the model tended to over-predict scour and under-predict accretion. Wu (2004) suggested that the difference between the results may “attribute to the development of sand dunes under unsteady flow conditions”. However, it may be due to the bed roughness calculation, which Ball et al. (1996) found to be over-predicted in the model; the Manning’s $n$ used for bed friction was twice the value calculated for a single pile from drag representation. The bed friction assumptions in the SW2D model in Stansby (2006) must, therefore, be assessed for accuracy.

There are, however, limitations of depth-averaged modelling for shallow wakes, as detailed in Stansby (2006). Stansby found that the critical value for a stable wake to form behind a structure in shallow water was over-predicted by the depth-averaged model. Also, the length of the stable wake bubble was under predicted by the model. Stansby stated that this is likely to be due to the fact that the amplification of the friction coefficient due to horizontal strain rates was not incorporated in the model.

Knight et al. (2007) studied depth-averaged velocity and boundary shear modelling in trapezoidal channels with secondary flows, using the Shiono and Knight Method (SKM). The lateral distributions of depth-averaged velocity and boundary shear stress in straight prismatic channels were calculated, incorporating bed shear, lateral shear and
secondary flow effects. The model was validated using a wide range of data from several sources and the results that were produced were found to have good accuracy. There were problems with the calibration of the model, but because it is a simple model, it can produce velocities and boundary shear stresses for real problems in less computational time than more traditional CFD models. The simplicity of the trapezoidal channel, however, is not very applicable to other channel conditions; if the channel is altered in any way, e.g. by adding a barrage to the flow, the depth-averaged velocities may no longer be accurate and the bed shear calculated from them may be incorrect. As with Wu (2004), with a more complicated geometry or flow condition the accuracy of the depth-averaged results, and therefore the bed shear stress results, may require validating using three-dimensional modelling or experimental results.

Ghamry and Steffler (2005) and Anh and Hosoda (2007) investigated slightly more complicated geometries: Ghamry and Steffler (2005) investigated the flow through curved open water channels and Anh and Hosoda (2007) studied the flows over complex terrains. Both studies used depth-averaged models to determine the velocity profiles and showed good agreement with experimental results. However, the Anh and Hosoda (2007) model is only applicable to channels with curved beds so is not appropriate for regular tank geometry and Ghamry and Steffler (1995) suggested that finer meshing close to the channel walls would improve the results.

Liang et al. (2006) also found that grid refinement was important for accurate results. Liang et al. (2006) suggested three potential improvements that can be made to a numerical model which is used for predicting depth-averaged shallow water flows.
Firstly, an enhanced grid layout can be used, second the method for simulating flooding and drying can be improved and third, a better irregular wall boundary representation could be used. These improvements can potentially be used for long-term, large scale simulations, since they better predict the effect of irregular beds and different beach configurations. For simple-sided flows the most important feature for modelling is therefore the grid layout and, thus, mesh size.

A two-dimensional unstructured hybrid mesh was developed by Lai (2010) and a finite-volume method was used to determine depth-averaged flows in open channels. The model uses the surface elevation as the main variable and is suitable for steady and unsteady flows, plus it can be used for subcritical, trans-critical and supercritical flow. Lai (2010) found that a triangular cell mesh was less accurate than quadrilateral cell or hybrid cell meshes and the velocity distribution was found to be sensitive to the mesh. Mesh sensitivity studies for 2-D modelling must therefore be conducted, though the model in Stansby (2006) uses a quadrilateral cell mesh, which is assumed to be more accurate than a triangular cell mesh (Lai 2010).

2.3.2 TIDAL BARRAGE MODELLING
Depth-averaged modelling has also been used for modelling tidal and coastal flows. Coastal flow models that cover large domains have been developed to include barrage schemes, in order to determine their impact on the far-field environment, and are generally based on depth-averaged shallow-water equations for computational efficiency. For example, Burrows et al. (2009a) and Burrows et al. (2009b) used two-dimensional modelling to model the entire Irish Sea and estuary approaches, in order to determine the far-field effects of five proposed barrages on the West Coast of the UK.
Depth-averaged modelling has also been used by Xia et al. (2010) to predict large-scale barrage-induced hydrodynamics and by Ahmadian et al. (2010) to predict the associated hydro-environmental parameters of water quality, and sediment and bacterial concentrations.

Previous modelling of the hydrodynamic processes affected by a tidal barrage has been conducted, particularly by Falconer et al. (2009, 2010), Xia et al. (2010a, b, c), Carroll et al. (2008) and Kirby (2010). The models have assessed the impact of the installation of a barrage on the water levels upstream and downstream, resulting loss of inter-tidal zones, decreased turbidity, improved water quality and altered sediment transport. The model regions range from relatively near-field, e.g. whole estuaries (Xia et al. 2010a) with grid sizes of approx 40 m, to far-field e.g. the entire Irish Sea (Burrows et al. 2009b) with grid sizes of approx 4-5 km in open sea and 50 m within the estuaries. Some studies have found that the installation of a barrage in an estuary will affect the sediment transport, accretion and deposition (Ahmadian et al. 2010), though no studies have looked at the near-field effect, within a few hundred metres of the barrage. Within this region there are expected to be jets from the barrage duct exits, swirling water produced by the turbines and flow expansion and deceleration as the flow exits the barrage into the still water downstream. These effects will lead to complex flows and that might affect the sediment transport and turbidity, the effects of which may have dissipated by the distances downstream previously modelled.

Previously models have used cell sizes from 40 m to 5 km (Xia et al. 2009, 2010a, 2010b and Burrows et al. 2009a), with grid areas of 57000 km² (Xia et al. 2010a, b) or
larger (Burrows et al. 2009a); this modelling was appropriate for this scale of project, however, to analyse the flow within the near-to-barrage region, i.e. within a couple of hundred metres of the barrage, a much smaller cell size is required. The models have also used unstructured triangular meshes, which were found to be suitable for this type of modelling.

The tidal barrage models have been two-dimensional, or depth-averaged, so have been assumed to ‘not display any significant vertical velocity or stratification’ (Xia et al. 2010c). Modelling of other estuarine flows has found this not to be the case; for example, Wai et al. (1996) investigated the 3-D flow along coastlines and Liu (2006) investigated the vertical circulation of estuarine flows. These studies found that there was significant vertical variation in the flow. The three-dimensional flow effects of a tidal barrage are therefore important. Particularly important is the need for assessment of the close-to-bed flows, as this will affect the sediment transport (as discussed in Wai et al. 1996 and Kirby 2010) and the flow field, and thus bed stresses, may differ from that calculated using depth-averaged assumptions.

Other key features of the flow were found to be altered by the installation of barrages, such as the water levels and discharge rate. Xia et al. (2010a) analysed the effects of the three different proposed barrages for the Severn estuary: the Cardiff-Weston barrage, the Fleming lagoon and the Shoots barrage. The Cardiff-Weston barrage was found to reduce discharge rates into the basin by up to 50%, but also to reduce maximum basin water levels by up to 1.2 m which could reduce the risk of flooding. The Shoots barrage was also found to decrease upstream levels by up to 1 m, but the downstream levels
were found to increase by 30 cm, thus increasing the flood risk. Xia et al. (2010b) found that the maximum discharge was predicted to decrease by up to 50% at some cross-sections with the implementation of a barrage, plus the water level upstream of the barrage could reduce by up to 1.5 m.

The velocity fields in the estuary and basin (Xia et al. 2010b) were also found to become more complex as the oscillatory tidal flow combines with the jet flow from the barrage, plus lateral flows were found to occur in the middle of the channel. Since the flow caused by the oscillatory tidal flow and jets is complex then the flow closer to the barrage must be analysed and further mesh refinement is required. Burrows et al. (2009a) also used two-dimensional modelling, but focussed on the whole Irish Sea, incorporating five barrage areas rather than just one. The bathymetry of the bed up to 3 nodes from the barrage was artificially lowered so that the model had improved stability and a more stable time step. Burrows et al. (2009a) stated that this artificial lowering has no effect on flow rates, turbine characteristics or external hydrodynamics, but since the grid size is so large the near field effects are not known. The near-field tidal amplitudes are stated to be accounted for in the two-dimensional modelling; however with a grid size of 50 m the flow within this distance of the barrage is unknown and may alter the flow downstream from the barrage.

The power output of barrages has also been modelled, for example in Xia et al. (2009) and Burrows et al. (2009a). Xia et al. (2009) created a two-dimensional hydrodynamic model to determine which type of generation produced the highest power output, plus its effect on the flood inundation extent. Xia et al. found that ebb-generation and two-
way generation would produce high levels of power, with a reduced flood risk. They also found that as the discharge coefficient for the sluices was increased the maximum power output was increased, but the total electricity generated was only increased slightly. The effect of the starting and minimum heads was also investigated, and it was found that as the starting head was increased above 4 m the generation window decreased, thus causing problems with trying to achieve a continuous power supply. If the starting head was kept as 4 m and the minimum water head was approximately 2 m, then the operation of the barrage at mean spring tide in ebb generation mode was improved.

Zero-dimensional modelling was also used by Burrows et al. (2009a) to model the barrage energy generation. The model assumed flat water surfaces either side of the barrage, which can improve performance calculation of the turbine, but does not take into account the hydrodynamics upstream or downstream of the barrage. The Generation program was used to model the barrage operation and to calculate the potential power output and basin water level. As the head difference across the barrage changed, the Turgency program (described in the paper) was used to model the through flow of water and the amount of power being produced.

Previous modelling of tidal barrage flows has been conducted; however, the models have been far-field with large grid sizes, so cannot account for near-field flow complexity and jet effects. The models have also been used to study the water quality and sediment transport, but neglect the jet effect on the bed, that was found to be important in jet experiments. The models have also been depth averaged, so have not
accounted for the three-dimensional flow complexity and circulation evident in estuarine flows. Three-dimensional modelling must therefore be used to model the near-field flow effects of barrages.

2.3.3 3-D MODELLING OPEN CHANNEL FLOW

Three-dimensional modelling has been conducted for other open channel flows, such as Wai et al. (1996). Wai et al. (1996) developed a water-sediment predictive model to study the effect that different coastlines geometries have upon the 3-D flow and sediment patterns. Vertically-averaged layers were used to represent the non-uniformity of the flow and the sediment distribution. Time-marching was used, so the time increment was relatively small; short-life activities such as sediment deposition could therefore be studied. The model results and the field measurements showed good correlation and variation throughout the depth was evident in both sets of results; therefore, in order to predict sediment patterns the velocity close to the bed required analysis. The model studied coastlines and whether these characteristics are still true for channel flow is unknown.

Czernuszenko and Rylov (2002) and Blumberg et al. (1992) found that three dimensional modelling produced reasonable results for open channel flow and both models gave good results for both primary velocity flows and secondary flows.

Czernuszenko and Rylov (2002) used a model to calculate the three-dimensional, stationary velocity field in a regular channel. The model was used to calculate the streamwise velocities and gave good secondary flow patterns in the cross-section of the channel. An anisotropic turbulence model was developed from a basic $k-\varepsilon$ model, which
contained a sub-model for shear stresses and a sub-model for normal stresses. The numerical solution also gave good flow patterns for the secondary flow as well as the longitudinal velocity. The flow patterns for the secondary flows showed vortices developed in the flow, which indicate the importance of three-dimensional modelling.

Blumberg et al. (1992) also used a three-dimensional hydrodynamic model with a turbulence-closure model \((k-\ell_l, \text{Mellor and Yamada 1982})\), which has proven effective for simulating turbulent flows. The model performance was compared with experiments, where laser-Doppler anemometer measurements were taken along the centreline of open-channel flows, and they were found to show good agreement.

All three models showed variation throughout the depth of the open water channel and were accurate at predicting secondary or vertical flows; this shows that three-dimensional modelling was required for this type of analysis since depth-averaged modelling would not have given the accurate flow profiles throughout the modelled region.

3-D models have also been developed to investigate the effect of estuarine flows on sediment transport. For example, Ma et al. (2009) investigated the effect of the Mersey flow on sediment transport processes. The numerical model results found that the sediment transport was affected by the tidal level and river discharge, which may vary significantly with the installation of a barrage.
Several simulations use turbulence models to close the RANS (Reynolds-Averaged Navier Stokes) equations; a summary of different turbulence models for predicting stratified flow and salinity are given in Liu et al. (2010). The study compared the results of $k-\epsilon_l$ (Mellor and Yamada 1982), $k-\epsilon$, $k-\omega$ and UB (Umlauf and Burchard 2003) turbulence models. The $k-\epsilon_l$ turbulence model was found to produce the most accurate results, depending on the background diffusivity, but the $k-\epsilon$ turbulence model results were still fairly comparable. Yang et al. (2011) compared the standard $k-\epsilon$, RNG $k-\epsilon$ and Reynolds Stress Model (RSM) turbulence models with experimental results for flow in an open channel junction. The models all assume fully-developed turbulent flow has formed, whereas the experiment may still be showing transition to turbulence, which may be a factor to consider in barrage flows. The $k-\epsilon$ turbulence models produced better results than the RSM model, with a fully developed vortex forming.

Eddy-viscosity models, such as the $k-\epsilon$ turbulence model, are commonly used for open channel flows, such as in Molls and Chaudhry (1995) and Cotton et al. (2005). The model detailed in Molls and Chaudhry (1995) is used for unsteady, sub-critical and super-critical flows. The turbulent Reynolds stresses were determined using an eddy-viscosity model, but the stresses formed by depth-averaging were neglected. Recirculating flows were also excluded when determining whether the effective stresses affect the converged solution; this assumption may make the results inaccurate, as recirculation within the flow may be an important flow feature of the results, shown in Liu (2006). Further modelling must take into account the recirculation within the flow and stresses formed by this.
A $k-\varepsilon$ turbulence model was used in Cotton et al. (2005) for analysing open channel flows with low-to-moderate bulk flow rates and a low Reynolds number. The eddy viscosity vertical profiles were determined using a limiting free surface condition, which produced improved results from the zero gradient form for the upper-channel section. Two more free surface boundary conditions were considered for the dissipation rate equation, however despite these improvements the model still under-predicted the turbulence levels throughout the channel depth. This under-prediction, however, occurred when a low Reynolds number was used, so may not occur for a high Reynolds number simulation. Alternatively, a different surface condition could be used for modelling the flow, thus reducing the under-prediction of the turbulence. The $k-\varepsilon$ turbulence model was found to produce reasonable results, so could be used for similar modelling conditions.

### 2.3.5 TURBINE MODELLING

The incorporation of turbines in the models could affect their performance. There are a few methods of simulating turbines, some of which include swirl and some of which do not.

#### 2.3.5.1 TURBINE WAKES

Aubrun (2007) described using porous discs to represent wind turbines and the measurement of the effects to the wake. It was found that the disc reliably created a velocity deficit, plus any amount of deficit could be produced by altering parameters, such as the disc diameter and the porosity level. Varying the material homogeneity produced a non-uniform velocity deficit, which is more like a real flow. Like a wind turbine, the porous disc concept extracts kinetic energy from the wind and creates shear-
generated turbulence. This porous disc method could therefore be used for representing turbines within ducts, since the drag on the flow imposed by the porous region would achieve an appropriate level of head difference and velocity decrease. The effect the disc has on the streamlines however is unknown and the swirl of the turbine stators and rotors is not incorporated.

Rotating porous discs have been previously investigated in Sharma and Mishra (2005), Verma and Sharma (1988) and Khan (1968), which include the restriction to the flow imposed by the turbines/porous disc as well as the swirl generated. However, only laminar flow between two discs has been analysed in these studies, rather than turbulent flow in the downstream region.

Gant and Stallard (2008) modelled a tidal turbine in unsteady flow, using computational-fluid-dynamics (CFD). The CFD simulations were run in time-dependent mode using two different inflow models: the von Karman spectral approach, which is commonly used in industry, and the Synthetic Eddy Method (SEM, Jarrin et al 2009). The unsteady flow simulations used a porous disc to represent the tidal turbine and results indicated that the wake produced was shorter than in steady flow simulations. Also, the turbulence produced by the SEM model was slightly less vulnerable to decay between inflow and disc downstream than the von Karman approach. The porous disc method also had similar flow results to that produced by Aubrun’s 2007 study and appear to be a reasonable representation of a turbine.
Porous discs, or a similar method of resistance to the flow, could reasonably be used with the incorporation of swirl to simulate the turbines within the ducts. These turbine methods have, however, been in open channel flow, whereas the swirl in ducts or pipelines could be different.

2.3.5.2 PIPELINES

Several papers have investigated the effect of swirl in pipe lines, specifically for assessing heat transfer effects. Ahmadvand, Najafi and Shahidinejad (2010) described the effect of adding vanes to a pipe inlet to generate decaying swirl with in the pipe using vane bodies and RSM turbulence model; this proved an effective method of generating swirl and led to the effective analysis of the swirl pattern on the velocities within the pipe, plus the wall stresses and turbulence intensity. The inclusion of bodies results in complex mesh generation and high numbers of cells, leading to expensive simulations. Including a mathematically induced swirl may be quicker and easier than a physically induced swirl.

Escue and Cui (2010) also analysed the swirl in pipelines, comparing the effectiveness of two turbulence models at predicting flow effects. The RNG \( k-\varepsilon \) model was found to give more accurate results than the Reynolds Stress Model (RSM) when comparing with experimental results for low amounts of swirl. The experimental swirl was created by rotating a tube bundle about the pipe axis (Rocklage-Marliani et al. 2003) and the computational swirl was specified by using the experimental axial and tangential velocity profiles, plus the Reynolds stress, as inlet conditions.
Susan-Resiga et al. (2011) also investigated the ‘mathematical modelling of swirling flow in hydraulic turbines’, specifically a Francis turbine. The swirl produced by this type of turbine is complicated as it has three elements: swirl from the stay vanes, the guide vanes and the rotor blades. The model developed by Susan-Resiga et al. related the velocity upstream from the guide vanes to that downstream, thus generating swirl mathematically. The results correlated strongly with the experimental results, which shows that accurate turbine modelling can be conducted and the influence of inclined guide vanes can be simulated mathematically.

Previous swirl modelling has been conducted, but several parts require further development. The swirl can be generated by stationary vanes, stators, or a rotating component such as a propeller, rotors, so the difference in the two methods could be investigated. Also, numerical modelling can accurately predict swirl flow features, including a $k$-$\varepsilon$ model, so this could be used for further swirl research.
2.3.6 SUMMARY

Previous large scale modelling of coastlines, estuaries and rivers has been conducted using depth-averaged or two-dimensional modelling, however the mesh size used was very large and the variation throughout the water depth cannot be taken into account. Mesh sizes of 40 m to 1 km have been used to cover areas of up to 5700 km$^2$ (Xia 2009, Xia 2010a and Xia 2010b), but a finer mesh should be used to model the area close to the barrage, within 20 diameters, to measure the velocities throughout the flow accurately. A head of 2 m was found to be sufficient for tidal barrage use (Xia 2009, Xia 2010a and Xia 2010b), so this head difference will be required for future barrage modelling. In order to model the near-field flow, several aspects of computational modelling were analysed. Two-dimensional modelling was found to be insufficient for modelling open channel flows (Czernuszenko and Rylov 2002 and Blumberg et al. 1992); so both 2-D and 3-D modelling must be assessed to determine the limit of applicability of depth-averaged modelling and the mesh used for the model must be finer than those previously used. Unstructured meshes can be used (Lai 2010), however polyhedral meshes were found to produce better results than triangular meshes (Cotton et al. 2005). $k$-$\varepsilon$ turbulence modelling produces reasonable results when modelling turbulent flows (Cotton et al. 2005 and Lai 2010), but turbulence model studies must be conducted to determine the effect of different turbulence models. The boundary conditions may also have to be assessed, since free surface boundaries led to under-prediction of turbulence levels (Cotton et al. 2005). Whether the model reaches steady state was assessed to determine whether wake oscillation occurs (Ball et al. 1996 and Stansby 2003). The bed shear stress, and thus the sediment transport, were determined using the close-to-bed flow since previously calculations from depth-averaged results
may not be accurate (Wu 2004 and Knight et al. 2007). Turbines have been previously modelled in several studies (Aubrun 2007 and Gant and Stallard 2008). Swirl modelling has also been conducted, including swirl generated by turbines (Susan-Resiga et al. 2011), however different swirl generation methods were employed, including swirl generation from vanes and numerical swirl generation. These different methods require further investigation, both experimentally and computationally to determine the most appropriate modelling method. Porous discs can be used in conjunction with numerically induced swirl to simulate the drag imposed on the flow. Within some CFD programs, for example StarCCM+, the model generation method for porous discs is more complex than numerically inducing the drag as well as the swirl. The effectiveness of numerically simulating drag compared to using a porous disc will be tested and the most effective method used.

2-D and 3-D models were used to model the flow through a turbine and the distance at which each method is most appropriate was determined. The model region was selected in order to analyse the flow close to the barrage and the mesh was refined to analyse the flow conditions close to the barrage as accurately as possible. The model was run with an appropriate turbulence model, such as $k$-$\varepsilon$ turbulence model, and the most suitable boundary conditions were selected. Whether the flow reaches steady state was also analysed, plus whether this is affected by different flow conditions. Different swirl generation methods were investigated, such as stator swirl and rotor swirl, to determine the most accurate modelling method. The velocities close to the bed were also assessed in order to determine the bed shear stress patterns and the sediment transport.
2.4 CONCLUSIONS

From this literature it can be seen that three-dimensional modelling and depth-averaged modelling have been used far from the barrage to determine water heights, estuary effects and sediment drift. However, there has been no modelling of the flow immediately upstream, downstream or through the barrage. This near-to-barrage modelling could provide information about the turbulence and swirl immediately downstream of the barrage, caused by the turbines within the barrage. This could lead to more detailed information about environmental effects such as sediment transport and deposition.

In open channel flows, the velocity flow field was found to vary with depth, however previous barrage modelling, which is only 2-D, does not account for this. The variation throughout the depth was found to decrease as distance from disturbances in the flow increased, leading to the assumption that 2-D modelling may become more accurate further from a barrage. The distance at which this method is most appropriate can be determined using result comparison with experimental results.

Swirl components which act upon pipe flows have been found to affect the flow field considerably, so the swirl components induced on the flow by the turbines within a barrage may affect the downstream flow field. Previous small-scale modelling of the swirl within a barrage has not been conducted, so should be analysed to determine whether it has any effect on the flow downstream.
The bed shear stress caused by a barrage and jet flow has been assessed; however, the velocity components used for this analysis were either depth-averaged throughout the whole channel or depth-averaged across a layer close to the bed. Previous open channel flow modelling shows a large variation in the velocity profile at different depths throughout the channel, so the bed shear stresses predicted from depth-averaged velocities may be inaccurate. The bed shear stress patterns, and thus the sediment transport patterns, caused by the velocity if the flow close to the bed must be assessed to determine whether depth-averaged velocities can be used to accurately predict the bed shear stress.

The key questions with regard to barrage modelling and experimentation that must be addressed are:

1. Can 3-D computational modelling accurately model the flow that develops in the near-field of a tidal barrage?
2. How far downstream does 2-D modelling become accurate?
3. How are the results affected by the swirl from the turbines within the barrage?
4. How are the bed shear stresses and the sediment transport altered by the barrage?
3. EXPERIMENTAL METHODOLOGY

Three analysis methods were used to study the flow characteristics of the multi-tube barrage: experimental work, three-dimensional (3-D) and two-dimensional (2-D) computational modelling. The three methods were used to determine the flow features upstream, through and downstream of the barrage.

The experiments were conducted using a pre-existing flume, so the dimensions and inlet conditions of the flow were determined by the flume size and pumping system; the experimental rig was, therefore, analysed first. The key characteristics defining the flow conditions were the flume dimensions, the inlet velocity and the water levels upstream and downstream of the barrage; Froude modelling criteria were selected to determine these parameters.

3.1 MODELLING CRITERIA

3.1.1 GEOMETRIC SCALING

Geometric scaling was applied to the dimensions of the proposed Severn barrage detailed in DEP Energy Paper 46 (1981) to specify the experimental model. The proposed turbine caissons had basic dimensions of 19 m width, 40 m height and 77 m streamwise length (shown in Appendix 2). The flume that was originally intended for use on the project was 0.31 m in height and 0.3 m in width, so a scale factor was used to fit a single barrage caisson to this flume. In order to fit the flume, a scale factor of $7 \times 10^{-3}$ was used (scaling also shown in Appendix 2). This gave a caisson of 0.133 m width, 0.287 m height and 0.539 m length. A larger flume was then selected for testing,
but the same scale factor was used. The flume had dimensions of 1.22 m width, 0.61 m height and 4.3 m length, so seven scaled caissons could fit across the flume width, with 10 turbine diameters (10D) distance upstream and 20 turbine diameters (20D) downstream. An upstream distance of 10D was selected to ensure uniformity of the inlet flow throughout the breadth and depth of the flume and 20D downstream was selected as the anticipated distance for the jet-like flow to return to uniform channel flow. The duct dimensions from the proposed barrage were thus scaled and the flow characteristics were determined by Froude scaling.

3.1.2 FROUDE NUMBER

The Froude number, $Fr$, is a dimensionless parameter that describes the flow characteristics of open channel flow, in terms of the ratio of inertial and gravitational forces. The Froude number for a general cross section is defined by the flow velocity and the mean depth:

$$Fr \equiv \frac{U}{\sqrt{gh}}$$

where $\bar{h} = \frac{A}{b_s}$.

The mean depth, $\bar{h}$, is calculated using the channel cross-sectional area, $A$, and the surface width, $b_s$; for a rectangular cross section the mean depth is the actual depth. When Froude number is less than 1 the flow is said to be subcritical, whereas when the Froude number is greater than 1 it is supercritical. When the flow is subcritical, the flow is controlled from a downstream point and information is transmitted upstream. This is the expected upstream flow condition in barrage-controlled flows.
At the proposed Severn barrage location, Xia et al. (2010a) determined that if a barrage is constructed within the estuary, the upstream channel is approximately 16 km wide and has an average depth, $\bar{h}$, of 32 m. The discharge, $Q$, in the estuary would be $6.5 \times 10^5$ m$^3$ s$^{-1}$, so the mean velocity was calculated using:

$$V = \frac{Q}{A},$$

(2)

where area, $A$, is $5.12 \times 10^5$ m$^2$. The average velocity is, therefore, approximately $1.27$ m s$^{-1}$ and the Froude number is 0.072, so the flow is subcritical.

The criticality of the flow in the experiments was maintained by altering the inlet velocity and the corresponding Froude number was calculated. Equality in the Froude number in model-scale and full-scale ensures gravitational forces and wave resistance are correctly scaled and the flow conditions in the model are accurate.

The Froude number was also used to set the scaling parameters. The length-scale was 0.007, so using Froude scaling, the velocity-scale was $0.007^{0.5}$ and the flow rate-scale was $0.007^{2.5}$.

### 3.1.3 REYNOLDS NUMBER

The Reynolds number, $Re$, is the ratio of inertial forces to viscous forces in a fluid regime and determines, for example, whether a flow is laminar or turbulent. In a turbulent regime, the flow is dominated by the inertial terms causing flow instabilities, such as eddies; therefore turbulence occurs at high Reynolds numbers.
Turbulence typically occurs in open channel flows where \( Re > 1000 \) and in pipe flows where \( Re > 4000 \). The Reynolds number in open channel flow is defined by the viscosity of the water, \( \mu \), the density of the water, \( \rho \), the domain size, \( L \), and the water velocity, \( U \):

\[
Re \equiv \frac{\rho UL}{\mu} \equiv \frac{UL}{v}.
\]  

(3)

For open channel flow with a rectangular cross section, the domain size is the hydraulic radius, where \( A \) is the cross-sectional area and \( P \) is the wetted perimeter:

\[
L = R_h = \frac{A}{P} = \frac{bh}{(2h + b)}.
\]  

(4)

Using the values for the Severn Estuary, as detailed in Section 3.1.2, the Reynolds numbers for normal flow is approximately \( 7 \times 10^7 \). This value cannot be scaled, as is generally the case in model studies. The Reynolds number was calculated and used to determine whether the flow was turbulent and the bed was hydrodynamically rough, but was not therefore used to set the flow conditions. The Reynolds number within the ducts was also calculated using the following formula and hydraulic diameter, \( D_H \):

\[
Re \equiv \frac{\rho UD_H}{\mu} \equiv \frac{UD_H}{v}.
\]  

(5)
3.2 CONFIGURATION

The experimental barrage was scaled from the Severn barrage dimensions given in Appendix 2. A scale factor of 0.007, 1 in 143, was used to produce a barrage that fitted the flume; the scaled turbine caisson dimensions are also shown in Appendix 2. The barrage dimensions were 1.22 m width by 0.287 m height by 0.539 m length. The full-scale ducts were tapered to cause flow acceleration at the turbine location but these were changed to cylindrical pipes for simplicity and ease of construction; cylindrical Perspex pipes were used for the turbine ducts. The outer diameter of the ducts was chosen for the pipe diameter (16 m scaled to 0.112m, Appendix 2), so that the exit area at the downstream end of the barrage was comparable to the directly scaled barrage dimensions. The scaled duct diameter at the turbine location was therefore proportionately larger than that in the Severn proposal, and the flow acceleration and deceleration due to the duct shape was not considered, but the effects of this were not considered to significantly alter the flow effects. The scaled duct radius was 0.056 m. However, due to the unavailability of Perspex pipes of this diameter, the duct radius was changed to 0.055 m. Similarly, for manufacturability, the gap between each pipe and the distance between the pipes’ base and the bed were increased from 0.021 to 0.035 m and 0.0175 to 0.02 m respectively. The centre-to-centre spacing of the turbine duct units, or caissons, was 0.145 m, so the barrage could accommodate seven duct units. The duct inlets were curved to avoid flow separation, as shown in the La Rance caisson design in Appendix 1. The modified flume dimensions are shown in Appendix 3.
The barrage was placed in the flume so that the flow conditions twenty duct diameters (20D) from the barrage could be assessed, with the exit further downstream. The duct diameter was 0.11 m; therefore, the downstream length was 2.2 m. The upstream tank length was 10D, 1.1 m, and the barrage length was 0.539 m; the total length required was 3.84 m, less than the overall length of 4.3 m of the tank.

### 3.3 FLOW CONDITIONS

The desired water depths were obtained by controlling the inlet discharge and the downstream water depth. A weir of 0.15 m height was used to keep the downstream water depth at a constant level; the weir was placed over 20D from the downstream wall of the barrage, at 2.8 m downstream. A discharge was selected to submerge the ducts completely and generate a head difference. The discharge, or inlet velocity, required to satisfy these conditions, i.e. to submerge the ducts completely without overtopping the barrage, varied with the amount of resistance imposed by the different turbine representations.

The water depths were measured using a metre rule and, therefore, had an accuracy of 1 mm. Average values were recorded for the upstream and downstream water heights on one side of the flume; the right side was selected for ease of measurement and the values were averaged due to possible human error when reading the depths and slight fluctuations in the water surface.

The system used a re-circulating flow, thus creating a constant flow rate and inlet velocity. The flow rate was calculated by timing the rate at which a measuring tank downstream from the flume was filled. Several discharge values were averaged to
account for error in the timing and measurement. The inlet velocity was, therefore, calculated using the discharge rate and the upstream water depth, as described in Equation 2. The inlet velocity was assumed to be uniform throughout the upstream section of the tank. The experiment was also highly repeatable as the discharge was kept constant using a control valve.

3.4 INSTRUMENTATION

The stream-wise, cross-stream and vertical velocities of the water at various locations in the tank were recorded to produce a three-dimensional description of the flow throughout the model. Two velocity meters were used: a 2-D Nortek Doppler Velocity meter (NDV) and a 3-D Nortek Vectrino Acoustic Doppler Velocity meter (ADV). Initially the NDV probe was used to record the stream-wise and cross-stream velocities, until the ADV probe was purchased.

The 2-D NDV probe uses the Doppler Effect to calculate the volume-averaged velocity of the flow within a certain volume; the transmit transducer emits an acoustic pulse, which passes through the sample volume and any particulates present create an echo, which is received by two parallel receive transducers (seen in Figure 3.1). The echo is processed to determine the Doppler shift and thus the velocity vector in a single plane. The sample volume was located 0.05 m from the probe, to provide measurements undisturbed by the probe, and the sample rate of the probe was 25 Hz. The velocities were recorded for a period of 120 s, so approximately 3000 results were recorded at each probe location; a convergence study was conducted to show that this time length was sufficient for the parameters of interest. The time-averaged value of these recordings was used for analysis.
The 2-D NDV was replaced with a 3-D ADV probe which operated using the same theory. It also recorded vertical velocities as it had receivers in the vertical direction, as shown in Figure 3.2:

The sample volume was also at 0.05 m from the probe; the sample volume was 6 mm in diameter, 7 mm high and the wavelength of the emitted beam was 1.8 mm. The sample rate was increased to 200 Hz, however this increased the variation in the velocities recorded and reduced the signal to noise ratio, possibly due to lack of particulates at all the sampling instances. Due to this high sample rate, seeding material was required to
improve the signal; a micro-bubble diffuser was added 1m upstream from the barrage, with pressurised air at 2 bar feeding in to the diffuser, which increased the signal-to-noise ratio and the correlation of the velocities. The velocities at each probe location were recorded for 180 s, so approximately 36000 samples were recorded; a time length convergence study was also conducted for this probe. To guarantee that the probes were fully operational the transducers were all submerged, to ensure there was no interference from the air or fluid surface.

Figure 3.3 shows the experimental apparatus used to produce the flow features described. The full engineering drawings for the barrage are shown in Appendix 4.

Figure 3.3: Schematic of experimental apparatus
3.5 BARRAGE WITH NO TURBINE REPRESENTATION

Initially, a barrage with no turbines was analysed; the seven ducts within the barrage were left uncovered and empty. The flow rate was set to completely submerge the ducts and create a head difference across the barrage. The recorded discharge, $Q$, was 0.0291 m$^3$ s$^{-1}$ and the water levels upstream and immediately downstream were 0.233 m and 0.216 m respectively; the water depths at one turbine diameter (1D=0.11 m), 2D, 3D, 5D, 10D, 15D and 20D downstream from the barrage where also recorded to determine the head loss over this distance. The width of the flume is 1.22 m, so the inflow velocity was calculated as 0.1025 m s$^{-1}$, using the recorded discharge and upstream water depth in Equation 2.

The average depth upstream was 0.233 m, so the flow becomes critical when the Froude number is 1, i.e. when the velocity is 1.511 m s$^{-1}$ (Equation 1). Since the upstream velocity is 0.1025 m s$^{-1}$, the Froude number is less than 1, so the upstream flow is subcritical. The exact Froude number in the upstream region was 0.068; the full scale value was 0.072 (based on the computational results from Xia et al. 2010a), so the values are extremely similar, indicating that the conditions are correctly scaled. The slight discrepancy between the values may be due to a reduced resistance to the flow within the ducts because no turbine bodies are present, unlike that assumed in the computational model.

The Reynolds number is defined in Equation 3. Upstream from the barrage $U=0.1025$ m s$^{-1}$, $L=0.1684$ m (hydraulic radius, Equation 4) and $\nu=1\times10^{-6}$ at 20°C, therefore, $Re=1.7\times10^4$. Directly downstream from the barrage $U=0.5$ m s$^{-1}$, $L=0.1593$ m.
and $\nu=1\times10^{-6}$ at 20°C, therefore, $Re=8\times10^4$. These values show that the regime was fully turbulent, which is consistent with the full scale condition. Within the pipes, the velocity could not be directly recorded, however, the velocity can be assumed to reach $0.875 \text{ m s}^{-1}$ by using the flow rate and cross-sectional area of the pipes. The Reynolds number within the pipes was assumed to be approximately $9.6\times10^4$; this shows that the flow was fully turbulent within the pipes also. The flow conditions are shown below in Table 3.1.

<table>
<thead>
<tr>
<th>Flow condition</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q_{in}$</td>
<td>$\text{m}^3 \text{ s}^{-1}$</td>
<td>0.0291</td>
</tr>
<tr>
<td>$h_1$</td>
<td>m</td>
<td>0.233</td>
</tr>
<tr>
<td>$h_2$</td>
<td>m</td>
<td>0.216</td>
</tr>
<tr>
<td>$H$</td>
<td>m</td>
<td>0.017</td>
</tr>
<tr>
<td>$U_1$</td>
<td>$\text{m s}^{-1}$</td>
<td>0.1025</td>
</tr>
<tr>
<td>$U_{duct}$</td>
<td>$\text{m s}^{-1}$</td>
<td>0.8750</td>
</tr>
<tr>
<td>$U_2$</td>
<td>$\text{m s}^{-1}$</td>
<td>0.5000</td>
</tr>
<tr>
<td>$Re_1$</td>
<td></td>
<td>1.7$\times10^4$</td>
</tr>
<tr>
<td>$Re_{duct}$</td>
<td></td>
<td>9.6$\times10^4$</td>
</tr>
<tr>
<td>$Re_2$</td>
<td></td>
<td>8.0$\times10^4$</td>
</tr>
<tr>
<td>$Fr_1$</td>
<td></td>
<td>0.068</td>
</tr>
<tr>
<td>$Fr_2$</td>
<td></td>
<td>0.343</td>
</tr>
</tbody>
</table>

*Table 3.1: Table of flow conditions with no turbine representation*
The 2-D NDV probe was traversed across the tank at a range of depths to obtain the stream-wise and cross-stream velocities of the water at various distances downstream. For each profile the velocities at 0.04 m intervals were recorded, so values at 30 points across the tank at each depth and distance were obtained. Within 0.35 m (i.e. 35 cm) of the wall, the wall produces an echo that the receive transducers detect; the distance of the probe from the wall can, therefore, be detected to an accuracy of 0.1 mm. Probe locations within this range were more accurate than at other points across the tank, where the location was measured using a ruler, to an accuracy of 1 mm.

The velocity profiles across the tank at one turbine diameter (1D=0.11 m) downstream, 2D, 5D, 10D and 20D were recorded. The profiles were recorded at depths with 0.04 m intervals; however, since the water depth was 0.2156 m, the probe was not fully submerged at 0.2 m from the bed, so the close-to-surface profile was recorded at 0.195 m. Also, the velocity of the water at 0.195 m from the bed at 20D could not be recorded because the probe was not fully submerged, due to decreasing water depth, so at this distance the sample height was 0.185 m. The vertical profiles at each distance downstream were also recorded at 1 cm intervals, using the 3-D ADV. The vertical location was measured using a point gauge, with an accuracy of 0.1 mm. The barrage and NDV are shown below in Figures 4 and 5.
Figure 3.4: Barrage with seven empty ducts

Figure 3.5: Barrage with seven empty ducts and NDV
3.6 BARRAGE WITH TURBINE REPRESENTATION

Turbine representation was incorporated into the model with the addition of bulb turbines. The bulb turbines were developed based on drawings from the La Rance barrage (EDF Dossier de Presse 2009 and de Laleu 2009, shown in Appendix 5). The full turbine incorporates both guide vanes, stators, to direct the flow in the desired direction, and a rotor that is driven by the flow, thus creating rotation to convert to electricity. The swirl generated by the turbines in the barrage was assessed in two ways. Firstly, bulb bodies with guide vanes were made so that the swirl created by stators could be analysed. Secondly, rotors were added downstream of the guide vanes, thus incorporating the moving component of the turbines.

3.6.1 BULB TURBINE WITH GUIDE VANES

Initially an aluminium bulb turbine with guide vanes only was constructed to enable the analysis of the restriction to the flow imposed by the bulb and the rotation of the flow induced by the angled guide vanes/stators, without any moving elements, as shown in Figure 3.6.

The bulbs were scaled based on the turbines in DEP Energy Paper 86 and the La Rance turbines shown in Appendix 5. The bulbs covered half the cross-sectional area, with radii of 0.0275m. The length of the bulb was 0.105 m and the turbine was supported by a structure placed at the middle of the cylindrical part of the bulb. The vanes were mounted downstream from the support structure, similarly to the La Rance turbines, with a pointed downstream bulb end to avoid flow separation and cavitation (Appendix 6).
The La Rance turbines housed 24 guide vanes each; however, due to the limitations of construction and the size of the model, the experimental bulbs had half the number of vanes. The twelve 0.015 m × 0.022 m vanes were set to an angle of 30° from the stream-wise direction. The vanes were sized so that they covered the entire cross-sectional area of the flume with a 1mm allowance at the duct edge. The length of the vanes was scaled so that the proportional area of the vanes was similar to that at full scale based on the La Rance turbines shown in Appendix 5. The stator angle is within the blade inclination range of the La Rance turbine (-5° to +35°, Appendix 5) and was selected in the absence of any further attainable data. The vanes were placed so that they directed the flow in the same direction as the rotor, to initiate movement when the rotor was added. The rounded end of the bulb faced upstream, with the guide vanes and nose cone facing downstream. The support structure fitted between the upstream and downstream sections of the duct to hold the turbine in position. The full engineering drawings are shown in Appendix 6.

![Figure 3.6: Bulb turbine with guide vanes](image)
Due to the increased restriction to the flow imposed by the bulb and guide vanes, the inlet velocity was reduced to maintain the same upstream and immediate downstream water depths that occurred with empty tubes. The resulting water depths were recorded using a ruler. The average recorded discharge, \( Q \), was 0.02224 m\(^3\) s\(^{-1}\) and the water levels upstream and downstream were 0.2326 m and 0.2154 m respectively. The width of the flume is 1.22 m, so this created an inflow velocity of 0.07837 m s\(^{-1}\).

The average depth upstream was 0.2326 m, so the flow becomes critical when the Froude number is 1, i.e. when the velocity is 1.511 m s\(^{-1}\); since the upstream velocity is 0.07837 m s\(^{-1}\) the upstream flow is subcritical with a Froude number of 0.052. This Froude value is similar to the full scale, indicating that the scaling was still accurate; however, the value is slightly less than the empty duct Froude number due to the reduced inlet velocity.

Upstream from the barrage the Reynolds number was 1.3×10\(^4\) and the region was fully turbulent. Directly downstream from the barrage \( Re=4.9\times10^4 \) and the flow was fully turbulent. Within the pipes the velocity can be assumed to reach approximately 1.3373 m s\(^{-1}\) as the flow is accelerated past the bulb turbines. Using the discharge and cross-sectional area in Equation 2; this gives \( Re=7.3\times10^4 \) so the flow was assumed to be fully turbulent within the ducts as well as upstream and downstream. The full scale flow was also fully turbulent, so the flow condition is maintained. The flow conditions are shown below in Table 3.2.
<table>
<thead>
<tr>
<th>Flow condition</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q_{in}$</td>
<td>m$^3$ s$^{-1}$</td>
<td>0.0222</td>
</tr>
<tr>
<td>$h_1$</td>
<td>m</td>
<td>0.233</td>
</tr>
<tr>
<td>$h_2$</td>
<td>m</td>
<td>0.215</td>
</tr>
<tr>
<td>$H$</td>
<td>m</td>
<td>0.018</td>
</tr>
<tr>
<td>$U_1$</td>
<td>m s$^{-1}$</td>
<td>0.0783</td>
</tr>
<tr>
<td>$U_{stators}$</td>
<td>m s$^{-1}$</td>
<td>1.3373</td>
</tr>
<tr>
<td>$U_2$</td>
<td>m s$^{-1}$</td>
<td>0.3100</td>
</tr>
<tr>
<td>$Re_1$</td>
<td></td>
<td>1.3×10$^4$</td>
</tr>
<tr>
<td>$Re_{stators}$</td>
<td></td>
<td>7.3×10$^4$</td>
</tr>
<tr>
<td>$Re_2$</td>
<td></td>
<td>4.9×10$^4$</td>
</tr>
<tr>
<td>$Fr_1$</td>
<td></td>
<td>0.052</td>
</tr>
<tr>
<td>$Fr_2$</td>
<td></td>
<td>0.213</td>
</tr>
</tbody>
</table>

Table 3.2: Table of flow conditions with bulbs and stators

The three-component downstream velocities were recorded using the 3-D ADV probe at 1D, 2D, 5D, 10D and 20D from the barrage. The flow velocities at depths of 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.185 m from the bed were measured; the probe was not fully submerged at 0.195 m from the bed, due to the vertical transducers on the 3-D ADV probe, so the close-to-surface profile was recorded at 0.185 m. Similarly, the water depth reduced with distance from the barrage, therefore, at 20D the sample height was 0.175 m. The close-to-bed profile at 0.01 m was also recorded.
The velocity profiles at 5D, 10D and 20D were recorded with 0.02 m intervals, whereas at 1D and 2D a finer grid was required due to the high amounts of swirl close to the duct exits, so an interval of 0.01 m was used. When the probe was within 36 cm of the wall, the wall distance was recorded, giving a probe location to accuracy of 0.1 mm; thereafter, a ruler was used to determine the probe locations, with an accuracy of 1 mm. The vertical profiles at each distance downstream were also recorded at 1 cm intervals, using the 3-D ADV. Figure 3.7 shows the barrage with seven turbines with stators fitted and the 3-D ADV probe.
Figure 3.7: Barrage with seven turbines with stators and Vectrino ADV
Finally, rotors were added to the bulb turbines directly downstream from the guide vanes/stators (Figure 3.8); these were four-bladed, axial-flow, model boat propellers, 0.1016 m in diameter.

Figure 3.8: Bulb turbine with stators and rotor and cross-section of friction brake
The rotors were chosen to resemble the La Rance turbines as closely as possible; four bladed rotors were used on the La Rance turbines (Appendix 5) and were mounted downstream from the inclined guide vanes. The rotors were selected so that the swept area of the blades covered the majority of the duct area. The blades were angled at approximately +23° slope, which was within the range used by the La Rance turbines (−5° to +35°). The propellers were not designed to be accurate representations of tidal turbines, but a reasonable approximation that would produce swirl in a similar manner. The swirl produced may, therefore, not be entirely accurate for full-scale predictions, but gives an example of the flow effects experienced downstream from the barrage.

The rotors imposed a greater drag force on the flow than the turbines with stators alone, which required an alteration to the inlet flow rate to prevent overtopping and thus, the head difference. In order for the maximum rotational velocity of the rotors to be achieved, the maximum inlet velocity which did not result in overtopping of the barrage was selected. The average discharge was 0.03833 m³ s⁻¹ and the upstream water depth was 0.287 m; therefore, the inlet velocity was 0.1095 m s⁻¹. The immediate downstream water height was 0.2328 m, resulting in a head difference of 0.0542 m.

The rotors were free to turn (however, due to the friction brake assembly there is some mechanical friction), creating an additional rotating component to the flow, which is similar to that which occurs in a barrage. There was no power output recorded, but the approximate power of each turbine can be calculated using the flow rate, as described in Xia et al. (2012). The power generated by each turbine, $P_t$, was calculated using Equation 6.
where the density of seawater, \( \rho \), is 1.025 \( \text{tm}^{-3} \) (values and units were stated in Xia et al. 2012), the discharge per turbine, \( Q_t \), is the overall discharge divided by 7 (number of turbines) and equals 5.5\( \times 10^{-3} \), \( H \) is the head difference across the barrage and the turbine efficiency, \( \eta_t \), is estimated to be 0.33 (based on predictions from various previous turbine models). This produced a turbine power of 1 W, which when scaled using scale factor \( s^{3.5} \) was equivalent to a full-scale power of 35 MW. The turbines proposed for the Severn barrage were rated at 40 MW, so this value is reasonable for energy generation.

Due to the low power generation of each turbine and the relatively high amounts of friction in the system, the power output was not recorded, but the turbines speeds were controlled with the use of friction brakes (shown in Figure 3.8 and Appendix 6). The brakes were entirely sealed in the bulb casing, ensuring that the brake remained dry, thus allowing the rotational speed to be more effectively controlled. The brakes can be controlled by tightening the nut on the end of the turbine shaft, which in turn compresses the spring in the system. The compressed spring exerts more force on the casing, thus restricting the movement of the rotor shaft and rotor. The speed can be controlled using this mechanism and the desired speed was calculated by scaling the La Rance turbines.
The rotational speed of the La Rance turbines is 93.75 rpm (de Laleu 2009) with a maximum over-speed of 260 rpm. Froude scaling dictates that the rotational velocity scale factor is \( s^{0.5} \), thus producing target scaled speeds of 1120 rpm. With no imposed braking, but with braking due to friction only, the maximum speed obtained by the turbines was 153 rpm, with an average rotational velocity of 138 rpm. Though the ideal scaled speeds could not be obtained, the addition of a moving component to the turbine system was investigated (Figure 3.9).

*Figure 3.9: Barrage with seven bulb turbines with stators and rotors*
The Froude and Reynolds numbers were again calculated to determine whether the scaled flow conditions were reasonable. The average depth upstream was 0.287 m, so the flow becomes critical when the Froude number is 1, i.e. when the velocity is 1.678 m s\(^{-1}\); the upstream velocity is 0.1095 m s\(^{-1}\) so the flow is subcritical \(Fr= 0.065\).

Upstream from the barrage the Reynolds number was \(2.1\times10^4\) and directly downstream from the barrage \(Re=10.1\times10^4\). Within the pipes the velocity can be assumed to reach approximately 2.3030 m s\(^{-1}\), so \(Re=12.7\times10^4\). The flow is, therefore, fully turbulent within the ducts as well as upstream and downstream. The flow conditions with bulbs, stators and rotors are shown in Table 3.3, along with those from the experiments with no turbine representation and bulbs with stators.

The downstream velocity profiles at 0.04 m, 0.08 m, 0.12 m, 0.16 m and close to the surface at 0.2 m were recorded using the 3-D ADV; an interval if 0.01 m was used at 1D and 2D and 0.02 m thereafter. The close-to-bed (0.01 m) velocities were also recorded. The location of the probe with respect to the bed was measured using a point gauge, as shown in Figure 3.10; the accuracy of the gauge is 0.1 mm.

\[\text{Figure 3.10: Point gauge measurement of probe depth}\]
<table>
<thead>
<tr>
<th>Flow condition</th>
<th>Unit</th>
<th>No turbine</th>
<th>Bulbs/Stators</th>
<th>Bulbs/Stators /Rotors</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q_{in}$</td>
<td>m$^3$s$^{-1}$</td>
<td>0.0291</td>
<td>0.0222</td>
<td>0.0383</td>
</tr>
<tr>
<td>$h_1$</td>
<td>m</td>
<td>0.233</td>
<td>0.233</td>
<td>0.287</td>
</tr>
<tr>
<td>$h_2$</td>
<td>m</td>
<td>0.216</td>
<td>0.215</td>
<td>0.233</td>
</tr>
<tr>
<td>$H$</td>
<td>m</td>
<td>0.017</td>
<td>0.018</td>
<td>0.054</td>
</tr>
<tr>
<td>$U_1$</td>
<td>m$^3$s$^{-1}$</td>
<td>0.1025</td>
<td>0.0783</td>
<td>0.1095</td>
</tr>
<tr>
<td>$U_{ducts, stators or rotors}$</td>
<td>m$^3$s$^{-1}$</td>
<td>0.8750</td>
<td>1.3373</td>
<td>2.3030</td>
</tr>
<tr>
<td>$U_2$</td>
<td>m$^3$s$^{-1}$</td>
<td>0.5000</td>
<td>0.3100</td>
<td>0.6000</td>
</tr>
<tr>
<td>$Re_1$</td>
<td></td>
<td>1.7×10$^4$</td>
<td>1.3×10$^4$</td>
<td>2.1×10$^4$</td>
</tr>
<tr>
<td>$Re_{ducts, stators or rotors}$</td>
<td></td>
<td>9.6×10$^4$</td>
<td>7.3×10$^4$</td>
<td>12.7×10$^4$</td>
</tr>
<tr>
<td>$Re_2$</td>
<td></td>
<td>8.0×10$^4$</td>
<td>4.9×10$^4$</td>
<td>10.1×10$^4$</td>
</tr>
<tr>
<td>$Fr_1$</td>
<td></td>
<td>0.068</td>
<td>0.052</td>
<td>0.065</td>
</tr>
<tr>
<td>$Fr_2$</td>
<td></td>
<td>0.343</td>
<td>0.213</td>
<td>0.397</td>
</tr>
</tbody>
</table>

*Table 3.3: Table of flow conditions with no turbine representation, bulbs/ stators and bulbs/stators/rotors*
4. COMPUTATIONAL METHODOLOGY

The experimental barrage and flow conditions were replicated using both 3-D and 2-D computational models, so that an appropriate model for assessing the flow downstream of a barrage, and the limits of applicability for that model, could be determined. 3-D CFD modelling was conducted using StarCCM+ and 2-D depth-averaged modelling was conducted using the SW2D model detailed in Stansby 2006. Both models were used for assessing the flow resulting from a barrage with no turbine representation and the advantages/disadvantages of each model were evaluated. 3-D CFD only was used for modelling the flow with swirl created by stators and rotors, because the swirl could not be modelled in a 2-D domain. The model geometries, meshes, boundaries and modelling physics are described in this chapter, plus the details of the swirl modelling in StarCCM+.

4.1 CFD MODELLING

In order to analyse the depth-varying flow upstream, through and downstream of a barrage, three-dimensional computer modelling was required. The program selected for this modelling was StarCCM+ because it is a widely available, commercial package, which may be used by other parties in future barrage modelling.

A StarCCM+ model was created to replicate the conditions of the laboratory, including the barrage dimensions, the flume geometry and the flow conditions. The first runs were based upon a barrage with seven empty ducts and then stator and rotor representation were added to determine the effect of swirl.
4.1.1 GEOMETRY AND MESH

The geometry that was created for analysis represented the upstream section of the flume, the seven ducts and the downstream section of the flume. The cuboid upstream tank was 1.1 m (10D) and the downstream tank was 2.8 m (over 20D) long, with the barrage separating the two regions. The water depths upstream and immediately downstream from the barrage were set to the measured experimental values. The free surface upstream was treated as a rigid horizontal plane boundary and downstream as a plane boundary sloping at a constant gradient; the slope angle was calculated using the depth on the downstream side of the barrage and the depth at 2.8 m downstream, which was extrapolated from the value at 2.2 m (to avoid the local influence of the weir in the computation). The barrage contained seven ducts, which were cylindrical with fillets at the upstream end; the radius and length of the ducts were 0.055 m and 0.539 m respectively. Figure 4.1 below shows the model components:

*Figure 4.1: StarCCM+ Model*
The mesh was created using StarCCM+; a polyhedral mesher was used to create a volume mesh, with a surface remesher to improve surface quality. Polyhedral meshes are relatively easy and efficient to create, because they contain approximately five times fewer cells than tetrahedral meshes, and tend to have better accuracy and efficiency than tetrahedral meshes (Tu et al. 2008). A prism layer mesher was also used, which created two layers of orthogonal prismatic cells along the wall boundaries; these are important for accurate predictions of turbulence.

The target cell size was varied between 0.05 m and 0.01 m; the results changed when the base size was 0.05 m, 0.04 m and 0.03 m, however at 0.02 m and 0.01 m there was no variation in the results. The base cell size was specified as 0.02 m because further refinement did not affect the results, however since the ducts were 0.11 m wide only 5 cells would cover the ducts, which would be insufficient for resolving internal pipe flow and jet exit. On the barrage walls and ducts the target cell size was 0.005 m, approximately 5 % of pipe diameter, with the mesh size growing to the base size at the inlet and outlet boundaries. The thickness of the prism layer, or layer of orthogonal cells along the wall, was specified as a third of the base size, so extended to 0.007 m from the boundaries. The mesh, with 333,540 cells, is shown in Figures 4.2 and 4.3.

Figure 4.2: StarCCM+ Model with Volume Mesh
4.1.2 MODELLING PHYSICS

The model based on Navier-Stokes and continuity equations was specified as three-dimensional, steady and for a single, constant density, fluid (water). The steady state was chosen because repeat recordings of the velocities at certain points in the flume showed that there was little variation in the flow over time; because the flow was predicted to remain steady over time, a small time-step was unimportant. To reach steady state, the model governing equations are iterated until the residuals reach convergence; the tolerance of the generic transport equation residuals was set to be $10^{-4}$. The governing equations, discretisation scheme, convection scheme and turbulence models used in the simulations are described and justified below.

4.1.2.1 GOVERNING EQUATIONS

The Continuity Equation, or Principle of Mass Conservation, states that in any steady state process, the rate of mass entering a system is equal to the rate of mass leaving a system.
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} + \frac{\partial (\rho u_j)}{\partial x_j} + \frac{\partial (\rho u_k)}{\partial x_k} = 0.
\] (7)

The flow is considered to be incompressible, because there are no significant density changes in the flow, which leads to several simplifications: there are no thermodynamics in the system, so changes in the internal energy are assumed to be insignificant and the governing equations are driven by pressure. For incompressible flows the density is constant along a streamline, so the equation reduces to:

\[
\frac{\partial u_j}{\partial x_j} = 0.
\] (8)

The Navier-Stokes equation is the momentum equation for viscous fluids; this states that the rate of change of momentum is equal to the force imparted on the fluid and can therefore be expressed as:

\[
\frac{\partial (\rho u_i)}{\partial t} + \nabla \cdot (\rho uu_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j},
\] (9)

where \( \tau_{ij} \) is the stress tensor, which viscous and turbulent stress parts:

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho u_i' u_j'.
\] (10)
The Reynolds-averaged form of the Navier-Stokes (N-S) equations are used for turbulent flow; these decompose each flow variable into mean or time-averaged flow component and a fluctuating, turbulent component. The turbulent fluxes in the mean flow are called Reynolds stresses. These stresses generate a net transport of momentum.

A discretisation scheme and an equation solver were required to solve the flow equations and a turbulence model was required to represent the turbulent stresses in the mean-flow equations. These are described in Section 4.1.2.2 and 4.1.2.3.

### 4.1.2.2 DISCRETISATION METHOD

The finite volume method (FVM) for discretisation is generally applied for fluid flows because it enforces conservation of mass, incorporates fluid phenomena and is relatable to physical quantities. FVM is used to solve the scalar-transport equation, which is used to determine the transported physical quantities (such as momentum and energy), over a finite volume. The equation comprises the rate of change term, advection term, diffusion term and source term:

\[
\frac{\partial}{\partial t}(\rho \phi) + \sum C \left( \phi - \Gamma \frac{\partial \phi}{\partial n} A \right) = SV, \tag{11}
\]

where \( V \) is the cell volume, \( A \) is the area of the cell face, \( \phi \) is the scalar to be solved (shown in Figure 4.4) and \( C \) is the mass flux. The equation is integrated over each cell, resulting in a balance between advective and diffusive fluxes across the control volume faces and any source terms are integrated over the cell volume. The transient term is not required for steady-state calculations, because the discretisation method uses the
solution from previous time steps, which are not applicable in a steady-state case. Advection schemes are applied over a control volume and are used to approximate the value of a variable, $\phi$, at a cell face or the derivative of a scalar for diffusive terms.

Advection schemes commonly used in flow modelling are Central Differencing (CD), First-Order Upwind Differencing (UD), Second-Order or Linear Upwind Differencing (LUD), Bounded-Central Differencing, Hybrid Differencing and Hybrid-BCD Differencing.

The Central Differencing (CD) scheme is commonly used for the discretisation of the diffusive terms because this produces stable results which are second order ($O$) accurate with respect to the distance between cell centroids. A solution is stable if errors in the calculation do not increase with steps. From Taylor series:

Figure 4.4: Cell volume

Figure 4.5: Adjacent cells and node locations, CD
\[
\frac{du}{dx} = \frac{u_E - u_P}{\Delta x} + O((\Delta x)^2)
\]  

(12)

The CD scheme for advective terms is also second order accurate, but suffers from a lack of boundedness. This is the concept that numerical solutions lie within correct physical bounds, i.e. \( \phi \) at \( P \) lies between \( \phi_W \) and \( \phi_E \). Due to the truncation of the Taylor series there are errors in the results, but the errors are reduced by refining the mesh i.e. reducing \( \Delta x \). Without an extremely fine mesh, there will be unacceptable errors in the result and the unbounded scheme gives rise to oscillatory solutions.

As a result of this a First Order Upwind Differencing (UD) scheme could be used; this scheme relies on the fact that advection causes transport only in the flow direction.

\[
u_e = u_P + O(\Delta x)
\]  

(13)

UD is only first order accurate, so less accurate than CD, but has the advantage of being bounded. UD also has the problem of numerical diffusion, where the Peclet number is
equal to the numerical diffusion over the real diffusion, and the magnitude of the problem decreases as $\Delta x$ reduces:

$$\text{Truncation Error} = \frac{\Delta x}{2} \left( \frac{du}{dx} \right)_e$$  \hspace{1cm} (14)

$$PecletNo. = \frac{\text{NumericalDiffusion}}{\text{RealDiffusion}} = \frac{(u_e A_y) \left( \frac{\Delta x}{2} \right) \left( \frac{du}{dx} \right)_e}{v \left( \frac{du}{dx} \right)_e} = \frac{u_e \left( \frac{\Delta x}{2} \right)}{v}$$  \hspace{1cm} (15)

The numerical diffusion is at its worst where the flow is skewed with respect to the mesh, so the best results are produced if the flow is perpendicular to the cell face. This is achieved by using a structured orthogonal mesh where possible. The StarCCM+ model has a polyhedral mesh, however, so UD may not be an accurate advection scheme.

An alternative is the Second-Order or Linear Upwind Difference (LUD) scheme which uses the node immediately upwind and the node upwind of that.

**Figure 4.7: Adjacent cells and node locations, LUD**
By applying the Taylor series a result is produced that is second order accurate, though order-of-accuracy is also only applicable for Cartesian meshes:

\[ u_e = u_p + \frac{u_p - u_w}{\Delta x} \frac{\Delta x}{2} + O\left(\Delta x^2\right) \]  \hspace{1cm} (16)

This second-order spatial discretisation scheme is used for unstructured meshes, because UD only works if each cell face is aligned with the adjacent cell. When an unstructured mesh is used the Green-Gauss method is required, which states that “the surface integral of a scalar function is equal to the volume integral of the gradient of the scalar function”:

\[ \int_V \nabla \phi dV = \int_A \phi \nabla \cdot dA. \]  \hspace{1cm} (17)

This formula can be used to calculate the reconstruction gradients, by rewriting the equation as:

\[ \nabla \phi_e = \frac{1}{V_p} \sum_e \phi_e A_e, \text{ where } \phi_e = \frac{\phi_p + \phi_e}{2}. \]  \hspace{1cm} (18)

The reconstructed face value from cell \( P \) on face \( e \) is:

\[ \phi_{e,P} = \phi_p + s_0 \cdot (\nabla \phi)_{e,P}. \]  \hspace{1cm} (19)
where $s_0$ is the distance from the cell centroid to the face centroid.

The reconstruction gradients are limited to ensure boundedness, so that in the convective flux in the second-order upwind scheme can be calculated using:

$$
(\dot{m}\phi)_e = \frac{\dot{m}_e \phi_{e,E}}{\dot{m}_e \phi_{e,p}} \quad \text{for} \quad \dot{m}_e \geq 0 \quad \dot{m}_e < 0
$$

(20)

The LUD scheme has higher accuracy than the UD scheme, but may have poorer convergence because there is higher numerical dissipation. The model has an unstructured grid, however, necessitates its use over the UD scheme.

Three other possible convection schemes are the Bounded-Central-Differencing (BCD), the Hybrid and the Hybrid-BCD; the BCD scheme is a bounded central-differencing scheme, but is used in large eddy simulations. The Hybrid scheme, which is a combination of the second-order upwind scheme and the central-differencing scheme, and the Hybrid-BCD scheme, which is a combination of the second-order upwind and the Bounded-Central-Differencing scheme, can only be used with detached eddy simulations. The LUD scheme was, therefore, used.

The Segregated Flow Model was selected for solving the velocity and pressure, because it is quicker and uses less memory than the Coupled Model. Coupled models are used for flows with high density fluctuations e.g. supersonic, but not for these flows. Within the Segregated Flow Model either a first order or second order scheme can be selected. The first order scheme, Upwind Differencing, was not used because the mesh was
unstructured so the nodes and cell faces were not aligned. The Linear Upwind Differencing (LUD) scheme was second order accurate (unlike the Upwind Differencing (UD) scheme) and was bounded (unlike the Central Differencing (CD) scheme), so was selected.

The Segregated Flow Model in StarCCM+ also uses Rhie-Chow Velocity Interpolation (Rhie and Chow 1983) with Semi-Implicit Method for Pressure Linked Equations (SIMPLE, Patankar and Spalding 1972) as the solution algorithm. The Rhie-Chow algorithm separately interpolates the pressure and non-pressure parts of the velocity to determine the advection velocities on the cell faces, so that the gradient of the pressure is taken into account and not assumed to be linear; it provides pressure smoothing and prevents “odd-even decoupling” that may occur with co-located storage of velocity and pressure. SIMPLE is widely used in constant-density flows; it is a pressure-correction method that is used to derive the velocity and pressure fields which satisfy the Navier-Stokes mass and momentum equations.

4.1.2.3 TURBULENCE MODELS

There are three commonly used methods for modelling turbulence within StarCCM+: Reynolds-averaged Navier-Stokes (RANS) modelling, large eddy simulation (LES) and detached eddy simulation (DES). LES and DES can only be used with extensive investigation into the grid resolution and also the computational costs are much higher than the RANS models, because small time steps and length scales are used. For these reasons a RANS model was used. Closure of the RANS equations is achieved by modelling the fluctuating stress components of the flow in terms of the mean flow quantities. There are several models that achieve this which can be categorised as either
eddy viscosity models or Reynolds stress transport models. Eddy viscosity models use additional transport equations to determine the turbulent viscosity, which is used to model the Reynolds stresses as a function of the mean flow. The Reynolds Stress Transport Model solves transport equations to close the RANS, but it solves an equation for each individual Reynolds stress. Eddy viscosity models are commonly used for modelling turbulence because they are relatively simple to code, the extra viscosity component improves the model stability and they have a good theoretical background for simple flows. However, because only a single viscosity value is modelled, some of the Reynolds stresses may be inaccurate and better predicted by a Reynolds Stress Transport Model. For simplicity, however, some commonly used eddy viscosity models were used for the barrage analysis. Two different types of eddy viscosity models were assessed - the $k$-$\varepsilon$ model and $k$-$\omega$ SST model - and some of the variations of the $k$-$\varepsilon$ model were also assessed.

The $k$-$\varepsilon$ model (Launder and Spalding 1974) is a two-equation eddy-viscosity model and is very widely used for turbulence modelling. It uses the following equations to calculate the turbulent viscosity:

$$
\mu_t = \rho \nu_t, \quad \text{where} \quad \nu_t = C_\mu \frac{k^2}{\varepsilon}.
$$

(21)

$C_\mu$ is a constant ($\sim 0.09$, Launder and Spalding 1974), $k$ is the turbulent kinetic energy and $\varepsilon$ is the rate of dissipation of turbulent kinetic energy. $k$ and $\varepsilon$ are determined from the transport equations, shown below, where $P^{(k)}$ is the production rate of $k$: 

116
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j k - \Gamma_{(k)} \frac{\partial k}{\partial x_j}\right) = \rho \left(P^{(k)} - \varepsilon\right)
\]  
(22)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j \varepsilon - \Gamma_{(\varepsilon)} \frac{\partial \varepsilon}{\partial x_j}\right) = \rho \left(C_{\varepsilon 1} P^{(k)} - C_{\varepsilon 2} \varepsilon\right) \frac{\varepsilon}{k}
\]  
(23)

\[
P^{(k)} = -u_{i j} \frac{\partial U_j}{\partial x_j}
\]  
(24)

Diffusivities of \( k \) and \( \varepsilon \) are related to the molecular and turbulent viscosities:

\[
\Gamma_{(k)} = \mu + \frac{H_i}{\sigma_k} \quad \Gamma_{(\varepsilon)} = \mu + \frac{H_i}{\sigma_\varepsilon}
\]  
(25)

The constant in the standard \( k-\varepsilon \) model (Launder and Spalding 1974) are: \( C_\mu = 0.09 \), \( C_{\varepsilon 1} = 1.44 \), \( C_{\varepsilon 2} = 1.92 \), \( \sigma_k = 1 \) and \( \sigma_\varepsilon = 1.3 \).

The seven \( k-\varepsilon \) models provided by StarCCM+ are Standard, Standard Two-Layer, Standard Low-Re, Realizable, Realizable Two-Layer, Abe-Kondoh-Nagano Low-Re, V2F Low-Re, Multiphase Standard and Multiphase Standard Two-Layer.

The original \( k-\varepsilon \) models are applied with wall functions, which are used to obtain the boundary conditions for the continuum equations. There are two alternative methods that resolve the viscous sub-layer; Low-Re models usually use damping functions and incorporate the wall distance and Two-Layer models treat the close to wall region.
separately and only this layer uses functions of wall distance. The advantage of using a model with wall functions is that less near-wall refinement is required.

The Standard model (Launder and Spalding 1974) was selected for modelling the flow, but sensitivity studies were conducted with other models; the Realisable model (Shih et al. 1994) and the V2F Low-Reynolds model were compared in the barrage with no turbine representation results. The results from these two models were found to be almost the same as the results produced by the Standard \( k-\varepsilon \) model, so the Standard model with a high Reynolds number (log law) wall treatment was deemed acceptable to use. This model used wall functions which better represent the flow close to the wall. The high \( y^+ \) wall function states that the turbulence model is only valid outside the viscous sub-layer of the flow and the cell close to the wall is assumed to lie within the log-law layer of the boundary layer.

The \( k-\omega \) model was also analysed when comparing the results with swirl imparted by stators only. The \( k-\omega \) model is also a two-equation model, but the transport equations are solved for \( k \) and \( \omega \), where \( \omega \) is the specific dissipation rate or dissipation rate per unit of turbulent kinetic energy:

\[
\omega \approx \frac{\varepsilon}{C_\mu k}. \tag{26}
\]

The model is fully detailed in Wilcox (1998). The boundary layers are, however, very sensitive to the specific dissipation rate in the free-stream, which is not a problem in the \( k-\varepsilon \) model. The shear-stress transport (SST) \( k-\omega \) model developed by Menter (1994)
effectively uses a $k-\varepsilon$ model far from the wall and a $k-\omega$ model at the boundary, thus reducing this effect. This model was used for the stator model and the results compared with the Standard $k-\varepsilon$ model.

4.1.2.4 BOUNDARY CONDITIONS

There were 11 boundaries in the model: upstream floor, sides and surface; downstream floor, sides and surface; barrage upstream end, downstream end and ducts; and the upstream inlet and downstream outlet. The boundary conditions in StarCCM+ were specified to match the experimental conditions as closely as possible.

The floor upstream and downstream and all the sides were specified as wall boundaries with rough, no-slip shear stress condition. The roughness of the walls, $k_s$, was specified as 0.001 m, appropriate for painted wood. The barrage ends and ducts were also specified as rough wall boundary conditions.

The surface upstream and downstream were modelled as symmetry planes to represent the free surface; this means that there is zero normal velocity at this boundary, but also that there is no shear stress or friction imposed on the flow.

The upstream face was specified as a velocity inlet, because the experimental inflow is assumed uniform across the whole of this plane and the inlet velocity was equal to that in the experiment. The downstream boundary was set as a constant pressure/head outlet, to simulate the water level of the weir in the experiment.
4.1.3 BARRAGE WITH NO TURBINE REPRESENTATION
The upstream water level was set to 0.233 m, with a discharge of 0.029 m$^3$ s$^{-1}$; thus the inflow velocity was 0.103 m s$^{-1}$. The downstream surface sloped from 0.216 m at the barrage to 0.203 m at 2.8 m downstream. All seven ducts were included in the model and the flow upstream, through and downstream from the barrage was studied. The velocity profiles at the same locations as the NDV probe locations at one turbine diameter (1D=0.11 m), 2D, 5D, 10D and 20D were recorded. The flow velocities at depths 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.195 m/0.185 m from the bed and at the duct mid-height were assessed. For each profile the velocities at 0.04 m intervals were recorded, so values at 30 points at each depth and distance were obtained; data were extracted from equi-spaced points along a line. The velocities at 1 cm from the bed were also extracted from the model for use in the bed shear stress assessment. The velocities at these heights and spacing were selected for direct comparison with the experiment, but the velocities at any height or spacing could be extracted from StarCCM+ simulations.

4.1.4 BARRAGE WITH TURBINE REPRESENTATION
The swirl generated by the turbines in the barrage was modelled in two ways; firstly, a bulb turbine with stators was created so the swirl created by stationary components could be analysed. Secondly, a rotor was added to the model, thus incorporating the moving element of the turbine. The computation models were calibrated to replicate the experimental flow conditions and the rotational component imposed by the stators and the rotors, as well as the acceleration of the flow around the bulb bodies.
4.1.4.1 BULB TURBINE WITH STATORS

The StarCCM+ model was altered to represent the new flow conditions in the flume; the upstream height was changed to match that in the experiments, as was the downstream water height and the inlet velocity. Due to the increased resistance to the flow imposed by the bulbs and stators, the flow rate was reduced to 0.0222 m$^3$s$^{-1}$ and the resulting upstream depth was 0.233 m, so the inflow velocity was reduced to 0.0781 ms$^{-1}$. The downstream depth was 0.215 m at the barrage and 0.197 m at 2.8 m downstream of the barrage, so the surface sloped at a negative gradient of 1:156.

Bulb bodies were added to the model with the same dimensions as the experimental bodies but without the stators, by altering the duct profile, as shown in Figure 4.8. The stators were not initially added to the model, in order to reduce mesh size and therefore computational time.

![Figure 4.8: Bulb turbine body in StarCCM+, a) Midheight plane b) 3-D Geometry](image)
A momentum source body force density, $F_b$, was added to the model to simulate the drag and the swirl imposed by the stators on the flow. The momentum source applied a drag constant, $C_{Fb,D}$, to the model along the $x$-axis in the reverse direction, a vertical swirl constant, $C_{Fb,S}$, along the $y$-axis and a horizontal swirl constant, $C_{Fb,S}$, along the $z$-axis, thus creating a restriction to the flow and clockwise swirl when looking downstream:

$$F_b = [- C_{Fb,D}, - C_{Fb,S}(z - z_{ref}), C_{Fb,S}(y - y_{ref})].$$

To capture the swirl detail the mesh was refined from 1D upstream of the stators to 1D downstream from the barrage. The target cell size in this region was 0.01m, approximately 9% of the duct diameter, with refinement at the wall from the prism layer meshing, and further refinement in this region did not affect the results (Figure 4.9).

*Figure 4.9: Bulb turbine body mesh refinement*
The momentum source field function was added to the model along the straight part of the bulb sides, so that it covered several cells. The function was applied from 0.245 m to 0.314 m from the upstream end of the duct, covering approximately 15 cells in the streamwise direction. The amount of swirl, $C_{Fb,S}$, and drag, $C_{Fb,D}$, that was applied to the model was altered and the details are specified in Chapter 6.

The accuracy of adding a field function to the flow to simulate the effect of the vane bodies on the flow was compared to the experimental result and also to a model with the vane bodies included. A model was created with the vanes themselves incorporated, to impart a drag force and swirl component to the flow (Figure 4.10). The 0.015 m by 0.022 m vanes were inclined at 30° to the streamwise direction, causing swirl in the clockwise direction when looking downstream. The mesh required further refinement from the conditions with only a bulb body, due to the complexity of the flow around the blades and the resulting swirl flow. The base cell size was 0.02 m; from 1D upstream of the bulb to 1D downstream of the barrage the cell size was 0.004 m, 20% of the base cell size; from 1D downstream to 2D the cell size was 0.008 m, 40% of base, and up to 5D downstream the cell size was 0.01 m, 50% of base.

The results from the field function momentum source models and the guide vane body model were compared with the experiment, with respect to the flow conditions, the swirl created, the midheight velocity vectors and the midheight velocity profiles throughout the flume.
4.1.4.2 BULB TURBINE WITH STATORS AND ROTORS

Rotors were then added to the model to represent the complete turbine configuration. The StarCCM+ model was altered to represent the new flow conditions in the flume. The upstream height was increased, due to the increased discharge (0.0383 m$^3$s$^{-1}$) and inlet velocity (0.1095 m s$^{-1}$), to 0.287 m. The downstream surface sloped at a negative gradient of 1:165 from was 0.233 m at the barrage and 0.216 m at 2.8 m downstream of the barrage.

The field function simulation method used for modelling the stators only was again employed for simulating the rotor flow, to determine how much of an effect the rotor had on the swirl. Another method was also compared: using an inbuilt fan function momentum source. The fan source uses actuator disc methodology to model the forces imposed by the blades and the detailed geometry of the blades is, therefore, not required. If a fan performance curve is available for the fan that is modelled then this can also be used, but is not a requirement; the rotors used in the experiments do not have a performance curve so the basic parameters of the model were used. The momentum source requires the input of the rotation rate and blade angle. The average
rotor rotation rate was 138 rpm, so the operating rotation rate was specified as 14.5 rad s\textsuperscript{-1}. The rotor blades have an average slope of approximately 23° pitch, so the blade angle in the model is 0.401 rad. The model also requires an upstream and downstream boundary; these were applied at the upstream and downstream end of the straight side of the bulb, giving a length of 0.0682 m that the fan source was applied over.

The results from the field function momentum source models and the fan function momentum source model were compared with the experiment, with respect to the flow conditions, the swirl created, the midheight velocity vectors and the midheight velocity profiles throughout the flume.

### 4.1.5 SIMULATION DETAILS

The models were run on an Intel(R) Core(TM) i7 CPU at 2.93 GHz, with 4 GB RAM. Convergence of the $x$- $y$- and $z$-momentum and continuity residuals in the model with no turbine representation occurred when the residuals dropped below $10^{-4}$, after approximately 1 hour. When more complex geometries were added to the model, with mesh refinement around the guide vanes, the simulation run-time increased to approximately 3 hours, though convergence was not guaranteed (this is described later in the Results).
4.2 DEPTH-AVERAGED MODELLING

Depth-averaged modelling of the barrage was performed using in-house Stansby shallow water 2-D codes (SW2D), detailed in Stansby 2006. A simplified model of the experimental downstream tank was created using SW2D codes in FORTRAN. The model uses a finite volume formulation of shallow water equations with a horizontal turbulent mixing coefficient.

4.2.1 GEOMETRY AND MESH

The program created a rectangular downstream region, with the upstream boundary representing the downstream wall of the barrage, the side boundaries representing the downstream tank sides and the downstream boundary representing the weir and outlet. The dimensions from the experimental apparatus were input into the program; the depth at the barrage was specified as 0.216 m and the outlet depth was set to 0.205 m. The centreline spacing of the turbines was 0.145 m, so the seven ducts were evenly distributed across the width and the duct exit points were specified. The duct areas were depth-averaged using the water depth at the barrage to give an accurate representation of the discharge and duct exit area. The discharge was also divided between the seven ducts, so that the inlet velocity at each duct was accurate. The inlet velocity was applied over the cells covering the depth-average duct area, shown in Figure 4.11.

The flume geometry was symmetrical about the centreline, as for the 3-D CFD. The rectangular mesh had an X-resolution (streamwise) of 280 cells, 0.01 m base size (9% of duct diameter), and a Y-resolution (cross-stream) was 140 cells, 0.009 m base size (8% of duct diameter); a mesh sensitivity study was undertaken which showed further refinement had no effect on the results.
4.2.2 MODELLING PHYSICS

Depth-averaged, or Shallow-Water, assumptions can be used where a single, constant density, fluid (water) with a free surface is modelled and the depth of the fluid is negligible compared to the domain size. The governing equations, boundary conditions, fluid parameters and time conditions used in the simulation are described below.

4.2.2.1 GOVERNING EQUATIONS

The model used depth-averaged assumptions to simulate the 2-D flow through the flume. The governing continuity and x- and y-momentum equations were depth-averaged, using depth-integrated streamwise and cross-stream velocities, \( \bar{u} \), as shown below.

Continuity:

\[
\frac{\partial \eta}{\partial t} + \frac{\partial (h \bar{u})}{\partial x} + \frac{\partial (h \bar{v})}{\partial y} = 0
\]  

(28)
\[\begin{align*}
\frac{\partial (hu)}{\partial t} + \frac{\partial (hu^2)}{\partial x} + \frac{\partial (huv)}{\partial y} &= -gh \frac{\partial \eta}{\partial x} - \frac{\tau_{bx}}{\rho} + \frac{\partial}{\partial x} \left( 2\nu_k h \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left[ \nu_k h \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] \quad (29) \\
\frac{\partial (hv)}{\partial t} + \frac{\partial (huv)}{\partial x} + \frac{\partial (hv^2)}{\partial y} &= -gh \frac{\partial \eta}{\partial y} - \frac{\tau_{by}}{\rho} + \frac{\partial}{\partial x} \left[ \nu_k h \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left( 2\nu_k h \frac{\partial v}{\partial y} \right) \quad (30)
\end{align*}\]

The bed stresses used in the model are derived using the equation below, the Blasius formula for \(C_f\) and the Reynolds number based on depth, \(Re\).

\[
(\tau_{bx}, \tau_{by}) = \rho C_f (u, v) \sqrt{\frac{u^2}{u^2 + v^2}}, \text{ where } C_f = 0.0559 Re^{-0.25} \quad (31)
\]

The eddy viscosity is a result of the horizontal and vertical strain rates:

\[
v_j = \left\{ l_h^2 \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 \right] + (\nu u, h)^2 \right\}^{1/2} \quad (32)
\]

The parameters used to determine the eddy viscosity are the friction velocity, \(u_r\), the Elder constant, \(\gamma \sim 0.067\) and the horizontal mixing length, \(l_h\). \(l_h\) is calculated using the boundary layer constant, \(\lambda = 0.09\), and the ratio of horizontal to vertical mixing length, \(\beta\):

\[
l_h = \beta \lambda h. \quad (33)
\]
4.2.2.2 DISCRETISATION METHOD

The finite volume method was again used for modelling the SW2D code. In order to minimize the effects of wave damping and numerical diffusion, the model was semi-implicit with second-order time stepping. The model uses the Crank-Nicolson method (Crank and Nicolson 1947) for time stepping; this is a second-order accurate method and is numerically stable. The method requires a small-time step; the model ran with a 0.0002 s time step which was sufficient.

The model used a QUICK (Quadratic Upstream Interpolation for Convective Kinematics) advection scheme and CGS (Conjugate Gradient Solver) in the coupled solution for surface elevation. QUICK determines the values on a cell face by using 3 nodes: the downwind, $\phi_E$, upwind, $\phi_W$, and upwind upwind nodes, $\phi_{WW}$:

$$\phi_e = -\frac{1}{8}\phi_{WW} + \frac{3}{4}\phi_W + \frac{3}{8}\phi_E$$

(34)

Figure 4.12: Adjacent cells and node locations, QUICK

The QUICK scheme uses a quadratic polynomial to relate these values and determine the value at the face:
The scheme is 3rd-order accurate, so more accurate than the StarCCM+ schemes. It is also conservative, because it is used within the finite volume method. The scheme is still upwind-biased, even though it uses the values at the downwind node, so is also transportive. It is not, however, bounded, which can cause problems in turbulence modelling, but because it is 3rd-order accurate it is still a good scheme to use.

The velocity-pressure coupling in the model is modelled using spatial discretisation on a staggered mesh (Harlow and Welch 1965), as opposed to the Rhie-Chow method used in StarCCM+. The velocity components of the flow are stored halfway between the nodes that are used for other scalar properties, such as depth. The advantages of this system are that no interpolation is required and, because the mesh is Cartesian, the velocity nodes do lie upstream of the pressure nodes that are driven by them. This can be a problem in more complex geometries.

4.2.2.3 BOUNDARY CONDITIONS

At the upstream boundary, which is the downstream end of the barrage, the depth-averaged width of a turbine was calculated to give the same area as the tube and the depth-averaged velocity that gave the correct tube discharge was input at cells covering the depth-averaged width. The discharge was 0.0291 m$^3$ s$^{-1}$, so the discharge and velocity per duct were 4.16×10$^{-3}$ m$^3$ s$^{-1}$ and 2.13×10$^{-3}$ m s$^{-1}$ respectively. The rest of the upstream edge inlet velocities were set to zero. The side walls were modelled as vertical slip planes, i.e. with zero normal velocity (symmetry plane). The downstream boundary was defined as a pressure boundary, with a fixed depth of 0.205 m (the extrapolated depth at the end of the region) and a zero normal velocity gradient.
4.2.2.5 FLUID PARAMETERS

Several parameters were input into the program for use in the running of the program models; the horizontal mixing coefficient, Crank-Nicolson theta, roughness height, von Karman constant and bed friction coefficient.

The ratio of horizontal to vertical mixing, used in Equation 33, was altered to determine its effect on the results. As the parameter was increased the jets merged closer to the barrage, resulting in reduced jet definition; this excessive merging led to the SW2D results becoming less comparable with the experimental results. A horizontal-to-vertical mixing ratio of 1 led to the most accurate results, where accuracy was determined by the relation to the experimental results.

The Crank-Nicolson theta was set to 0.5, because the model is semi-implicit; a value of 0 gives the fully explicit forward Euler method and a value of 1 gives the fully implicit backward Euler method. The von Karman constant was defined as 0.41, since this is a typical value; the constant is used for describing the velocity profile near a no-slip boundary for turbulent flows. The boundary layer profile uses the roughness height, $k_s$, which was defined as 0.001 m. The bed friction coefficient, $c_f$, was calculated using the rough wall log law, the inlet depth and the roughness height:

$$c_f = \frac{2}{\left(6.2 + 2.5\log \frac{h_{in}}{k_s}\right)^2} = \frac{2}{\left(6.2 + 2.5\log \frac{0.216}{0.001}\right)^2} = 0.0138.$$  \hspace{1cm} (35)
4.2.3 TIME STEPPING

The program used a ¼ sine variation ramp up for flow rate with a period of 100 s, so the inlet velocity was reached after 25 s. The run time was set as 100 s in order to allow the step-up to be achieved and for the model to run with the target inlet velocity for a decent length of time. The time step was set to 0.0002 s, so there were 500 000 time steps; this was the largest time-step that allowed the program to run. The results produced at 75 s and 100 s were identical, so the solution was assumed to reach steady state.

4.2.4 SIMULATION DETAILS

The models were run on an Intel(R) Core(TM) i7 CPU at 2.93 GHz, with 4 GB RAM. 100 s of run time was completed after 3 hours, but steady state may have been reached earlier, so this time could be reduced with further analysis.

4.2.5 ANALYSIS

The 2-D computational velocity vectors were compared with the velocity vectors at duct mid-height produced by the 3-D computation in StarCCM+. The 2-D velocity profiles at 1D, 2D, 5D, 10D and 20D were compared with the depth-averaged experimental results and depth-averaged 3-D computational results. Swirl could not be incorporated into the model, so the experiments with swirl from stators and stators/rotors were not modelled.
5. RESULTS- NO TURBINE REPRESENTATION

The first set of experiments conducted had no turbine representation within the seven ducts and is described in Chapter 3, in particular in Section 3.5. The experimental set-up is shown below in Figure 5.1:

![Experimental set-up](image)

*Figure 5.1: Experimental set-up*

The experimental conditions were represented as a 3-D CFD model using StarCCM+, as detailed in Chapter 4, Section 4.1. The 2-D SW2D model described in Section 4.2 was used to compare the depth-averaged results from the experiments and StarCCM+ to 2-D results.

The water depths were measured at 1D, 2D, 3D, 5D, 10D, 15D and 20D downstream, where the tube diameter \((D)\) is 0.11 m. The velocities across the tank were recorded both at the duct midheight \((0.075\text{ m from the bed})\) and at depths of 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.195 m from the bed and at 0.185 m from the bed at 20D, using the
2-D NDV probe; the probe locations (excluding those at duct midheight) are shown in Figure 5.2. The streamwise and cross-stream velocities were recorded at 1D, 2D, 5D, 10D and 20D downstream. The velocity vectors, streamwise velocity profiles and depth-averaged velocities were used to compare the experimental results with 3-D and 2-D computational results.

![Experimental velocimeter locations](image)

**Figure 5.2: Experimental velocimeter locations**

### 5.1 FLOW CONDITIONS

The discharge was set so that the tubes were fully submerged; the inlet discharge, $Q$, was $0.0291 \text{ m}^3 \text{ s}^{-1}$, the average upstream height close to the barrage, $h_1$, was $0.233 \text{ m}$, giving an average upstream inlet velocity, $U_{in}$, of $0.102 \text{ m s}^{-1}$. The average height immediately downstream, $h_2$, was $0.216 \text{ m}$, giving an average head difference across the barrage, $H$, of $0.017 \text{ m}$. This corresponds to a head difference of $2.43 \text{ m}$ at full scale, which was found by Xia et al. (2010) and Shaw and Watson (2003) to be optimal for energy generation.

The head drop over the barrage, as predicted by StarCCM+, was derived from the surface pressure difference between the upstream and downstream lids. The pressure on each lid is shown below in Figure 5.3. The average higher pressure of approximately
95 Pa is shown immediately upstream of the barrage and the average lower pressure of approximately -32 Pa is shown across the surface immediately downstream.

Figure 5.3: StarCCM+ surface pressures upstream and downstream of barrage

The pressure drop, $P$, across the barrage was approximately 127 Pa; the head change was calculated using the following formula:

$$H = \frac{P}{\rho g}$$

where $\rho = 1000$ kg m$^{-3}$ and $g = 9.81$ m$^2$ s$^{-1}$. The head change predicted by StarCCM+ was 0.0130 m, which is within 24% of the experimental value; therefore StarCCM+ slightly under predicted the pressure difference, and thus head loss, across the barrage.

### 5.2 WATER LEVEL VARIATION

The experimental water depths downstream of the barrage at 1D, 2D, 3D, 5D, 10D, 15D and 20D from the barrage were recorded on one side (right side looking downstream) of the flume. The 3-D water levels were derived from the surface pressures recorded on the downstream lid. The surface pressure was approximately zero and converged on this average towards the downstream end of the tank. Where the pressure was lower, the
water height predicted by the model was slightly less than that measured in the experiments and where the pressure was higher, so was the water height.

The 3-D surface variation from the rigid lid was derived from the surface pressure difference from zero, assuming hydrostatic pressure, using Equation 36; the surface fluctuations varied from -6 mm to 0.6 mm (Figure 5.4). The areas of low pressure and reduced water height at the flume sides close to the barrage occurred at the centre of the eddies, which is expected. The low pressure close to the barrage occurs where there is flow in the cross-stream and negative vertical directions, resulting in a lower surface depth than the rigid lid.

![Surface height variation downstream of barrage, StarCCM+](image)

*Figure 5.4: Surface height variation downstream of barrage, StarCCM+*

The surface elevations were calculated using these surface fluctuation values added to the lid level. The depths at 1D, 2D, 3D, 5D, 10D, 15D and 20D along the flume centreline and on one side (right side looking downstream), were calculated. These
locations were selected to correspond with the experimental depth locations. The depth at 2.5 m downstream was also recorded so that the depths at the end of the flume in the 3-D model and 2-D model could be compared.

The surface heights in the depth-averaged model were directly output and the depths at the flume centreline and the right side of the flume were extracted. The longitudinal water depth variations from the experiment, the 3-D CFD StarCCM+ model and the 2-D SW2D model are shown in Figure 5.5.

![Variation of water depth along tank](image)

*Figure 5.5: Water depth variation, Head = 0.017 m*

The experimental depth immediately downstream of the barrage decreased with distance downstream; the water level at the barrage was 0.216 m and at 2.2 m from the barrage was 0.206 m, a drop of 0.01 m.
The 3-D model also shows decreasing water depth with distance from the barrage, though the water depths predicted are slightly less than those shown in the experiments; close to the barrage there is an average 2.7 mm discrepancy, though there is lateral variation in the depth and this value is dependent on the averaging method, but at 2.2 m downstream the depths are equal. The StarCCM+ model also predicted a slight variation across the tank (between values at the centreline and near the flume sides), but at greater distances from the barrage this difference becomes negligible. Close to the barrage the water depth at the centreline is lower than at the sides, where there is stagnant water.

The depth-averaged model showed an almost horizontal downstream surface with depth very slightly increasing with distance downstream, probably associated with higher local velocities close to the barrage. The equivalent effect of bed friction may be analysed by considering the gradually varied flow equation for channel flow:

$$\frac{dh}{dx} = \frac{S_0 - S_f}{1 - F_e^2} \tag{37}$$

where $h$ is depth and $x$ is downstream distance. The bed slope, $S_0$, is zero and the friction slope, $S_f$, is calculated using:

$$S_f = \frac{\tau_b}{\rho gh} = \frac{1}{2} \frac{U_{avg}}{gh} c_f \tag{38}$$

where $U_{avg}$ is velocity magnitude (depth-averaged).
Since the average downstream depth is 0.216 m and the discharge is 0.0291 m$^3$s$^{-1}$, the average velocity is 0.11 m s$^{-1}$. With the reference coefficient of friction, $c_f$, defined by the rough wall log law (Chapter 4, Page 131, Equation 35) as 0.0138, the friction slope was calculated as 3.94x10$^{-5}$ using one step in the gradually varied flow equation. The Froude number, $Fr$, was calculated to be 0.0756, using Equation 1 from Chapter 3. The depth change from 0D downstream to 20D downstream of the barrage was found to be -9.71x10$^{-5}$ m; the surface was almost horizontal. However with the experimentally measured depth decrease input into the gradually varied flow equation, $S_f$=4.34x10$^{-3}$, a $c_f$ value of 1.52 or 110 $c_{f0}$ was calculated. Using this value in the 2-D model the water depths were found to be close to the experimental results (Figure 5.6). While this $c_f$ value is unphysical, and in reality most of the energy losses are associated with the jets rather than bed friction, it does demonstrate the value needed to replicate the effect of marked head losses induced by the 3-D jets.

![Variation of water depth along tank](image)

*Figure 5.6: Water depth variation with high $c_f$*
5.3 DUCT MIDHEIGHT RESULTS

The experimental and CFD velocities were recorded at the duct midheight plane, where \( z = 0.075 \) m. The experimental velocity vectors in the streamwise and cross-stream directions were computed and the three-component StarCCM+ vectors were extracted from the model.

5.3.1 VELOCITY VECTORS

The computational velocity vectors at the duct midheight are shown in Figure 5.7. The inlet velocity, on the left boundary, is 0.103 m s\(^{-1}\) and the maximum flow reached within the ducts is 0.5 m s\(^{-1}\), so the flow accelerates as it enters the ducts; the flow decelerates as it expands into the downstream tank. The velocities within each of the ducts are not uniform, due to the asymmetry that forms in the downstream tank. The flow features in the downstream tank can be compared with those recorded in the experiments.

![StarCCM+ velocity vectors at duct midheight throughout flume](image)

Figure 5.7: StarCCM+ velocity vectors at duct midheight throughout flume

The velocity vectors in the downstream tank at duct midheight in both the experiments and the StarCCM+ model are shown in Figure 5.8. Jets are evident directly downstream from the duct exits and extend into the downstream tank. Minima in the velocity occur
between each of the jets close to the barrage, but as distance from the barrage increases the jets merge and the velocity magnitude becomes more uniform across the flume.

Asymmetric eddies form on both sides of the flume, leading to reversed flow at the tank sides. The asymmetry is due to the Coandă effect causing the jets to incline towards one side of the flume (described in Reba 1966); the Coandă effect occurs where the presence of the flume wall does not allow entrainment of fluid from one side of a jet, thus pulling the jet towards the flume wall. The experimental asymmetry is always in one direction, due to minor imperfections in the flume; however, the asymmetry in the StarCCM+ flow could be triggered in either direction because the geometry is symmetrical and only the direction matching experiments is shown here.

The eddy formation also forces the outer jets to incline towards the centre of the flume, causing enhanced merging of the jets. Areas of very low velocity form at the eddy centres and at the end of the eddies’ area of influence. Towards the end of the flume the velocity becomes more uniform across the tank, with slightly decreased velocity along the right side of the flume.

The flow features in both the experimental and computational results appear analogous; however, the accuracy of the results can be assessed using the streamwise velocity profiles. The magnitude of the jet velocities, extent of the eddies and the distance at which the flow becomes more uniform across the flume was analysed by assessing the streamwise velocity profiles throughout the flume depth and width.
Figure 5.8: Velocity vectors at duct midheight in downstream region,

a) Experimental b) StarCCM+
5.3.2 STREAMWISE VELOCITY PROFILES

The streamwise velocity profiles across the flume at the duct midheight were recorded at 1D, 2D, 5D, 10D and 20D. The experimental and StarCCM+ results are shown below in Figure 5.9:

![Streamwise Velocity at Duct Midheight](image)

*Figure 5.9: Streamwise velocity at duct midheight*

The seven jets are evident at 1D directly downstream from the duct exits, with maximum experimental and computational velocities of 0.48 m s\(^{-1}\) and 0.52 m s\(^{-1}\) respectively; the StarCCM+ model, therefore, predicts an 8% error in the maximum velocity and the maximum velocities are approximately 466 % of the inlet velocity. As distance from the barrage increases and the flow tends towards the inlet condition, velocity decreases until at 20D it is approximately 0.12 m s\(^{-1}\), 117 % of the inlet velocity, in both the experimental and computational results. The jets also merge with
distance, so that at 2D the jets are still evident but they are indistinguishable at 5D and beyond.

The eddies are evident at the flume sides, with reversed flow observed at 1D, 2D and 5D. The reversed flow on the left side of the flume reaches maximum width, 0.2 m, and magnitude, $-0.17 \text{ m s}^{-1}$, at 5D; however, the reversed flow on the right decreases in size and strength with distance from the barrage. This asymmetry is due to the Coandă effect.

By 10D downstream the flow appears to be mostly uniform across the flume, with a 4% error between the maximum velocities predicted by the two methods, with slightly higher velocity in the tank centre, which reduces further by 20D downstream. Whether the flow is uniform throughout the depth and, therefore, 2-D modelling would be applicable, can be assessed by analysing the vectors and profiles throughout the depth.

Again, the 3-D CFD modelling results are in very close agreement with the experimental results and show similar velocity magnitudes, jet merging and eddy formation. Whether the model is accurate at all depths was also assessed.
5.4 DEPTH-VARYING RESULTS

The velocity vectors were analysed in the $z=0.04\,\text{m}$, $0.08\,\text{m}$, $0.12\,\text{m}$, $0.16\,\text{m}$ and $0.195/0.185\,\text{m}$ planes to determine the flow features present throughout the flume depth. The streamwise velocity profiles at these depths at 1D, 2D, 5D, 10D and 20D downstream were also examined and the accuracy of the 3-D model was investigated.

5.4.1 X-Y VELOCITY VECTORS IN HORIZONTAL PLANE

The measured experimental velocities and computed velocities from StarCCM+ are shown to be in close agreement and the velocity vectors are shown in Figures 5.10-5.14. Figure 5.10 shows the velocity vectors close to the bed, at $0.04\,\text{m}$; the experimental results show the streamwise and cross-stream velocities at 1D, 2D, 5D, 10D and 20D, whereas the StarCCM+ results show the three-component velocities throughout the downstream tank.

The velocity vectors at this depth (0.04m) are similar to those at the duct midheight, though the maximum velocity in the jets has reduced and there are larger areas of low velocity flow between the duct exits and downstream from the eddies. The large areas of low velocity close to the bed may result in sediment deposition, though this will be studied in Chapter 8. The asymmetric eddies are also evident at this depth, due to the Coandă effect. The experimental and computational results have a strong correlation at this depth. The velocity vectors at 0.08 m are extremely similar to those at duct midheight, because the centreline is at $z=0.075\,\text{m}$; these are shown in Figure 5.11. The ducts extend to 0.13 m from the bed, therefore, the velocity vectors at 0.12 m are near the top of the duct exits; they still show jets forming downstream from the ducts though at this depth they are narrower and weaker than at the duct midheight (Figure 5.12).
Figure 5.10: Velocity vectors at 0.04 m from the bed, a) Experimental b) StarCCM+
Figure 5.11: Velocity vectors at 0.08 m from the bed, a) Experimental b) StarCCM+
Figure 5.12: Velocity vectors at 0.12 m from the bed, a) Experimental b) StarCCM+
The jets do not extend as far downstream, because they expand into the ambient flow above the duct exits, causing uniform flow across the flume much closer to the barrage. The eddies are also smaller and weaker as they merge with the ambient water.

At the levels within the duct exit area, the vectors show that jets form. Close to the surface, however, the jets are no longer evident, with marked cross-stream flow occurring; these are evident at 0.16 m (Figure 5.13) and 0.195 m/0.185 m from the bed (sampling depth varied with distance from the barrage, Figure 5.14).

Close to the surface strong cross-flow forms, especially close to the barrage at 1D and 2D, and the jets are no longer apparent. The asymmetry in the flow is highly influenced by the eddies; high velocity flows form inboard of the eddies, particularly on the left side of the flume (looking downstream). This causes asymmetry to extend down the flume, though the flow is now predominantly in the streamwise direction (Note the colour scale of the plots has changed, so the asymmetry is no stronger than that exhibited at the lower depths).

These flow features are also evident close to the surface at 0.195 m from the bed (0.185 m at 20D in the experimental case). These are shown in Figure 5.14. The flow features vary considerably throughout the depth, especially close to the barrage, with correlation between the experimental and computational results. The magnitudes of the streamwise velocity and the eddy formation at each depth were examined using the streamwise velocity profiles at 1D, 2D, 5D, 10D and 20D.
Figure 5.13: Velocity vectors at 0.16 m from the bed, a) Experimental b) StarCCM+
Figure 5.14: Velocity vectors at 0.195 m from the bed, a) Experimental b) StarCCM+
5.4.2 STREAMWISE VELOCITY PROFILES

The variation of streamwise velocity throughout the depth was compared at 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.185 m/0.195 m from the bed, at 1D, 2D, 5D, 10D and 20D downstream. At 1D downstream from the barrage (Figure 5.15), jets are clearly apparent at 0.04 m to 0.12 m from the bed, directly downstream of each draft tube (the tubes have lower and upper points at 0.02 m and 0.13 m from the bed). Both the experimental and 3-D computational results predict a similar peak streamwise velocity of approximately 0.46 m s\(^{-1}\) at 0.08 m, which is close to the duct centre level (0.075 m). This is ~350\% of the inlet velocity and there is only a 10\% error in the StarCCM+ results.

![Streamwise Velocity at 1D Downstream](image)

*Figure 5.15: Streamwise velocities at 1D downstream at different vertical, z, locations*

These jets are also evident at these levels at 2D downstream (Figure 5.16), though the peak velocity has reduced to ~0.43 m s\(^{-1}\) (320\% of the inlet velocity) and there is some
merging of the outer jets, leading to less defined minima between the jets. Close to the surface at both 1D and 2D downstream the jets are not discernible, with marked vertical and cross-stream flow, as seen in the velocity vectors.

The eddy sizes at the flume sides predicted by the experiments and the CFD model are quite similar; the streamwise velocity component becomes positive at the same cross-stream location. The experimental reversed flow reaches a maximum of -0.09 m s\(^{-1}\) at 1D and 2D; the computational results are within 10\% of the experimental results.

As distance from the barrage increases the difference between the experimental and computed eddy sizes increases somewhat, although the maximum reverse velocity is still similar; the velocity is -0.13 m s\(^{-1}\) close to the bed and -0.11 m s\(^{-1}\) close to the surface, a variation of 15\%.

![Streamwise Velocity at Two D Downstream](image)

*Figure 5.16: Streamwise velocities at 2D downstream at different vertical, z, locations*
The StarCCM+ results differ from the experimental results at 2D downstream, particularly near the outermost jet locations, probably due to the under predicted spreading of the jets. There is only, however, a 5% difference in the maximum velocities and 7% difference in the minimum velocities. At 5D, however, the results are quite similar. The jets have merged further by 5D downstream (Figure 5.17) and are not evident even at the tube centre level. The experimental and StarCCM+ results are very similar, both in terms of the jet locations, jet merging and the velocity magnitudes.

![Streamwise Velocity at Five D Downstream](image)

*Figure 5.17: Streamwise velocities at 5D downstream at different vertical, z, locations*

The eddy width also reaches a maximum at 5D downstream, with reverse flow occurring within 0.2 m of the left tank wall with a maximum reversed velocity of -0.2 m s\(^{-1}\); the computational result is also accurate to 10% close to the bed, but drops to 20% closer to the surface. At these three distances downstream, eddies occur throughout the depth.
As the Coandă effect causes the central jets to incline towards one side of the tank, the left-hand eddy increases in size, whilst the right-hand eddy decreases, producing large-scale asymmetry in the flow. This is particularly obvious at 5D downstream. The asymmetry is depth dependent: close to the bed the central jets are merged, with fairly constant velocity across the centre of the flume and asymmetric eddies on either side, whereas close to the surface there is strong streamwise and reversed flow on the left side of the flume and close-to-zero streamwise flow on the right. Some asymmetry in the results across the flume is evident at 10D downstream from the barrage in Figure 5.18, with a merged jet forming. The merged jet location is depth dependent; close to the bed the peak velocities occur on the right side of the flume, while closer to the surface the jet moves to the left side. The variation in the velocities throughout the flume at this distance are lower than previously exhibited, so the flow is becoming uniform, however slight variation in the velocity profiles still occurs.

![Streamwise Velocity at Ten D Downstream](image)

**Figure 5.18**: Streamwise velocities at 10D downstream at different vertical, z, locations
At 20D downstream (Figure 5.19) the velocity profiles are fairly constant across the width and depth, with slight residual asymmetry leading to a slight decrease in velocity on the right side of the flume. The average streamwise velocity at all depths at 20D is 0.11 m s⁻¹, so the flow is returning to uniformity at a velocity similar to that at the inlet, \( U_{in} = 0.103 \text{ m s}^{-1} \); the streamwise velocity is within 7% of the inlet velocity. There is only \(~7\%\) error between the maximum velocity in the StarCCM+ and experimental results. At this distance downstream there is very little change in the velocity profile throughout the depth, suggesting that depth-averaging at this distance will be a reasonable approximation. The experimental and computational results are highly comparable; the errors in the maximum velocity are approximately 10% throughout the flume and the velocity profiles are extremely similar. This shows that the 3-D CFD StarCCM+ model predicts the flow features throughout the flume very well.

![Streamwise Velocity at Twenty D Downstream](image)

**Figure 5.19:** Streamwise velocities at 20D downstream at different vertical, z, locations
5.4.3 VERTICAL PROFILES OF STREAMWISE VELOCITY

The computational velocity vectors at the flume centreline are shown in Figure 5.20; the streamwise velocities at 1D, 2D, 5D, 10D and 20D are shown throughout the depth. The vectors again show that close to the barrage the maximum streamwise velocities occur directly downstream from the duct exit area, i.e. a jet forms between 0.02 m and 0.13 m from the bed. The velocities above and below this are significantly lower and close to the surface become reversed, as seen in the horizontal velocity vectors.

![Vertical Velocity Vectors](image)

*Figure 5.20: Vertical profiles of streamwise velocity vectors, StarCCM+

As distance from the barrage increases the jet bends towards the bed, due to the Coandă effect, which causes a jet close to a wall to bend towards that wall due to lack of entrainment. At 10D downstream there is slightly increased velocity flow close to the bed, due to the jet merging, but by 20D the normal open channel flow profile has been established.
5.5 DEPTH-AVERAGED RESULTS

The flow features described have varied considerably with depth and the variation has only become negligible by 20D downstream. Depth-averaging is commonly used to predict barrage flow effects, however may not be appropriate within 20D. The experimental and 3-D CFD results were depth averaged and compared with 2-D SW2D results in order to determine the limits of applicability of depth-averaged modelling for predicting barrage flows.

5.5.1 X-Y VELOCITY VECTORS IN HORIZONTAL PLANE

Figure 5.21 shows the streamwise and cross-stream velocity vectors produced by the 2-D SW2D model at 1D, 2D, 5D, 10D and 20D. The vectors show jets forming downstream from the duct exits, which merge and decelerate with distance from the barrage. The 2-D jets do not exhibit the Coandă effect, which leads to smaller eddies and no asymmetry further down the flume. The effect was not triggered by an artificial initial cross flow in the 2-D model.

The extent of the jet influence, the eddy sizes, the asymmetry and the distance downstream where depth-averaged modelling is more applicable, can be assessed by evaluating the velocity profiles across the tank width.
5.5.2 STREAMWISE VELOCITY PROFILES

The experimental and 3-D CFD velocities were depth-averaged for comparison with the 2-D model. The experimental velocity values at 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.195 m/0.185 m from the bed were averaged. The streamwise velocities in the experiments were averaged so that the velocity measurements at each depth represented the average velocity across a 4 cm band. The velocity at 0.04 m from the bed represented the average velocity between 0.02 m and 0.06 m; 0.08 m from the bed covered the band from 0.06 m to 0.10 m; 0.12 m covered from 0.10 m to 0.14 m; 0.16 m covered from 0.14 m to 0.18 m; and finally, the close to surface measurement covered from 0.18 m to the surface. The mean of these values at each transverse and lateral location were used to generate depth-averaged velocity profiles. In StarCCM+ the velocities at 20 points throughout the depth at each cross-stream location were averaged. These covered 20 bands of approximately 0.009 m, with the first velocity measurement covering a band from 0.0045 m to 0.0135 m, through to the close to
surface band from 0.1755 m to the surface. These depth-averaged streamwise velocities are shown in Figure 5.22, with comparison with the SW2D results.

![Depth-Averaged Streamwise Velocity Profiles](image)

**Figure 5.22: Depth-averaged streamwise velocity profiles**

At 1D downstream from the barrage, the seven jets downstream from the tube exits are evident, with reversed flow at the tank sides indicating the presence of eddies. The jets predicted are quite similar in size and velocity magnitude; SW2D has an 11% error in the maximum depth-averaged velocity from the experiments. This difference between the SW2D results and the experiments may be inaccurate due to the coarseness of the experimental measurements. There are only 5 velocity measurements across the depth at each cross-stream location, which cover bands of approximately 4 cm; within this amount of depth the velocity can vary greatly, which isn’t represented by a single point measurement. The velocity measurements also only cover the flow between 0.02 m from the bed to the surface, so the close to bed velocity is not accounted for. At this
depth, the velocity is near-zero, whereas the first measured velocity (at 0.04 m) is in the jet, so is very dissimilar. If this close to bed flow were represented the error between the SW2D results and the depth-averaged velocity would be approximately 8% and the results would be more similar to the SW2D results than would physically be true.

The experimental and CFD eddies predicted are also of similar magnitude, but the depth-averaged model predicts smaller, symmetric eddies. However, the depth-averaged results are very different from the depth varying results. At 1D the max velocity was approximately 0.55 m s⁻¹, but the maximum depth-averaged velocity is only 0.255 m s⁻¹, so the depth-averaged model under predicts the velocity magnitude, particularly at the duct depth. Also, the jets that are evident here do not represent the close to surface flow, where the main flow direction is cross-stream and jets are not evident.

At 2D downstream the results again show reasonable agreement for the central jets; however, the depth-averaged model under predicts the velocity magnitudes and the 3-D model over predicts, especially for the outer jets. The difference between the experimental maximum velocity and the SW2D result is -23% and the StarCCM+ is +19%. The depth-averaged model shows more spreading than experiment; this leads to different jet tracks, lower velocity magnitudes and smaller eddies. The spreading rate of the depth-averaged model is dependent on the horizontal mixing length ratio, defined as a multiple, \( \beta \), of the vertical mixing length; \( \beta \) was varied and a value of unity gave best correlation between the model and experiment. The maximum velocity is again under predicted when compared with the depth-varying results, similar to the results at 1D. The experimental results are very coarse throughout the depth, with the first cell within
the jet region, therefore losing the close to wall detail; this may account for the comparability of the depth-averaged results to the SW2D at these distances downstream, which may not be an entirely accurate representation of the experimental conditions. The depth-averaged model velocities at 5D downstream can be seen to be quite different from the 3-D CFD and experimental results due to the lack of asymmetry. The maximum velocity differs by -40% in SW2D and +18% in StarCCM+. The SW2D also shows no reversed flow at this location, whereas the maximum eddy strength occurs at this distance in the 3-D results. At 10D downstream the velocity across the flume is much more uniform and the depth-averaged model velocities are closer to experiment and 3-D CFD, with only a 10% error. At 20D downstream both computations and experiment show a fairly constant depth-averaged velocity value of 0.13 m s\(^{-1}\), similar to the depth varying result and 26% of the inlet velocity of 0.103 m s\(^{-1}\), across the whole width. The SW2D results are 8% from the experimental result; the small difference between the results at this distance is a residue of the Coandă effect further upstream.

The 2-D model gives reasonably accurate results compared to the depth-averaged experimental results; however, the variation throughout the depth, particularly close to the surface, evident in the 3-D results is not represented. The strong cross-stream flow close to the surface and the large asymmetric eddies are not shown by the 2-D model. The maximum velocities are also under predicted, by a magnitude of 50% in some cases. At 20D, however, the variation throughout the depth and width is negligible and the results predicted by the 3-D experiment, the 3-D computation and the 2-D computation are all comparable.
5.6 CONCLUSIONS

Three main issues were addressed in this chapter:

1. How does velocity vary with depth and distance from the barrage?
2. How well this is predicted by generally available 3-D CFD?
3. At what distance downstream is 2-D depth-averaged modelling justified?

1. The experiment showed that at the tube centre level there are high velocity jets downstream of the tubes and recirculating eddies at the flume sides. Close to the surface, however, jets are not apparent. As distance from the barrage increases the velocity variation across the width and depth is reduced. The transverse velocity profiles are asymmetric and dependent on vertical level up to 10D downstream of the barrage and have become almost uniform by 20D downstream.

2. The 3-D computational modelling using StarCCM+ provides reasonable comparison with the experiment. The velocity vectors and streamwise velocity profiles are highly comparable and the StarCCM+ model gives a good approximation of the flow.

3. Depth-averaged modelling was expected to be inadequate close to the barrage and this work suggests that it only becomes valid at least 20D downstream. 3-D effects also clearly affect the surface profile within 20D of the barrage; the depth variation is much greater than predicted by the depth-averaged model with a standard bed friction coefficient and close agreement is only achieved by increasing this by a factor of over 100, causing the actual bed friction to be grossly overestimated. 3-D CFD only slightly underestimates the water level variation.
6. RESULTS- BULB HOUSING WITH STATORS

The addition of swirl to the experiments was conducted in order to simulate working turbines more accurately. Initially bulb bodies with inclined stators were added to the ducts, as described in Chapter 3, Section 3.6.1. The bulb provided a central blockage to the flow causing acceleration at the duct edges, as occurs in barrage operation; the stators provided a swirl component to the flow, but there was no rotor.

Experiments were conducted with seven bulb bodies with inclined stators. The water heights downstream were measured, in the same way as the experiments with no bulb/stator representation. The three-component velocities were recorded throughout the length, breadth and depth of the flume using the 3-D ADV, thus adding a vertical velocity component that was not available with the 2-D NDV probe. At 1D and 2D the velocities were recorded at 0.01 m spacing across the flume, whereas at 5D, 10D and 20D the swirl had dissipated so a coarser spacing of 0.02 m was used. The three-component velocities were measured at 0.04 m, 0.08 m, 0.12 m and 0.16 m from the bed, which was the same as the previous experiment. The close-to-surface flow was recorded at 0.185 m, except at 20D where the velocities were recorded at 0.175 m, because although the immediate downstream surface height was maintained from the previous experiments, the water height dropped more downstream, due to swirl-induced losses. The StarCCM+ model was adjusted to incorporate the swirl experienced in the experiments; this was conducted in several different ways. Firstly, the swirl and drag imposed by the stators on the flow was calculated and added to the model using a body force (as described in Chapter 4, Section 4.1.3.1). Second, the body force was adjusted to match the resulting swirl and thirdly, the guide vane bodies were added to the model.
Body forces were used initially because they do not increase the mesh size required for modelling and, therefore, the computer run time. The applicability of the modelling methods was assessed using three criteria: the pressure difference across the barrage, the swirl calculated from the tangential and streamwise velocity at the duct outlets, and the streamwise velocity profiles downstream. The experimental results were analysed first, then the modelling methods compared.

### 6.1 FLOW CONDITIONS

#### 6.1.1 FLOW RATE AND INLET VELOCITY

The flow rate was set so that the tubes were fully submerged and the downstream depth was similar to that in the experiments with no stators or bulb bodies; the inlet discharge, \( Q \), was 0.0222 m\(^3\) s\(^{-1}\), the average upstream height close to the barrage, \( h_1 \), was 0.233 m, giving an average upstream inlet velocity, \( U_{in} \), of 0.0781 m s\(^{-1}\). The velocity was, therefore, reduced from the previous experiment, due to the increased resistance to the flow imposed by the bulb bodies. The average height immediately downstream, \( h_2 \), was 0.215 m, giving an average head difference across the barrage, \( H \), of 0.018 m (calculated on Page 91). This corresponds to a head difference of 2.57 m at full scale, which is also optimal for energy generation (Xia et al. 2010 and Shaw and Watson 2003).

#### 6.1.2 WATER LEVEL VARIATION

The experimental water depths downstream of the barrage at 1D, 2D, 3D, 5D, 10D, 15D and 20D from the barrage were recorded on one side (right side looking downstream) of the flume. Figure 6.1 shows the depths recorded in Chapter 5 without stators and the depths with bulb bodies with stators:
The depths in both the experiments decreased with distance downstream of the barrage. The water level at the barrage was 0.215 m and at 2.2 m from the barrage was 0.201 m, a drop of 0.014 m when bulbs/stators were included in the model, compared to a drop of 0.01 m when the ducts are empty. With the addition of stators and generation of swirl, the wakes become more complex, leading to an increased rate of energy loss and quicker drop in water depth than when there is no swirl generated. From 10D the rate of energy loss and, therefore, rate of decreasing depth becomes comparable with that when there is no swirl, indicating that the jets and swirl have dissipated by this distance downstream.

Figure 6.1: Water depth variation, Head = 0.018 m

(NB No turbine – duct only, Vanes - Bulbs with stators in ducts)
6.1.3 SWIRL

In order to model the swirl produced by the stators, the circulation at the outer limit of the ducts and the angular momentum flux across the duct areas were calculated, for comparison with the 3-D model. The tangential and streamwise velocities were measured 3 cm from duct exits across the depth at the centreline of each duct and throughout the width of the flume. The vertical profiles were recorded from 0.03 m from the bed to 0.12 m and the horizontal profiles were taken across the duct from 0.005 m to 0.05 m radius. The tangential velocities recorded are shown in Figure 6.2.

\[ U_{\theta 0} = U_y \quad U_{\theta 90} = -U_z \quad U_{\theta 180} = -U_y \quad U_{\theta 270} = U_z \]

Figure 6.2: Tangential velocity direction looking downstream

The circulation, \( \Gamma \), was measured at the outer limit of the duct using the tangential velocity, \( U_{\theta 0} \), and radius, \( r \):

\[ \Gamma = \int U_{\theta} r d\theta \quad (39) \]
The duct diameter is 0.11 m; therefore, the radius at which the velocities were recorded was 0.05 m. For better comparison with the computational results, the experimental results were non-dimensionalised \((\Gamma_{N-D})\) using the inlet velocity and the duct diameter, as described in Equation 40.

\[
\Gamma_{N-D} = \frac{\Gamma}{U_{in}d}.
\] (40)

The duct diameter, \(d\), was 0.11 m and the inlet velocity for the stators was 0.0783 m s\(^{-1}\). The non-dimensionalised circulations at 3 cm are shown in Table 6.1.

The circulation calculation does not include the streamwise velocity, which varies significantly across the duct due to the acceleration caused by the bulb body, and is only analysed at the outermost radius, not across the entire duct exit. In order to incorporate the impact of the variation of the streamwise velocity and the variation across the duct when assessing swirl, the angular momentum flux was also analysed.

The angular momentum flux, \(AMF\), was measured using the integral of the tangential velocity, radius and streamwise velocity, \(U_x\), (Equation 41), where \(\dot{m}\) is the mass flow rate and \(\rho\) is the water density.

\[
AMF = \dot{m}U_\theta r
= \rho \int U_x U_\theta r dA
\] (41)
The results were also non-dimensionalised ($AMF_{N-D}$) as described in Equation 42.

$$AMF_{N-D} = \frac{AMF}{\rho U_{in}^2 d^3}.$$ (42)

The non-dimensionalised circulation and angular momentum flux produced by the computational models were compared with the experimental $\Gamma_{N-D}$ and $AMF_{N-D}$ values, shown in Table 6.1, to determine whether the swirl in the jets was accurately modelled.

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Gamma_{N-D}$</td>
<td>74.1</td>
<td>66.2</td>
<td>73.6</td>
<td>86.6</td>
<td>82.5</td>
<td>76.2</td>
<td>68.7</td>
<td>75.4</td>
</tr>
<tr>
<td>$AMF_{N-D}$</td>
<td>0.80</td>
<td>1.01</td>
<td>0.79</td>
<td>1.12</td>
<td>0.93</td>
<td>1.24</td>
<td>0.87</td>
<td>0.96</td>
</tr>
</tbody>
</table>

*Table 6.1: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment*
6.2 DUCT MID-HEIGHT RESULTS

The experimental streamwise and cross-stream velocities were measured at the duct midheight, \( z = 0.075 \) m. The velocity vectors and the streamwise velocity profiles are shown below.

6.2.1 VELOCITY VECTORS

The experimental velocity vectors at 1D, 2D, 5D, 10D and 20D at the duct midheight are shown in Figure 6.3. The maximum velocity in the flume is 0.31 m\( \text{s}^{-1} \) and the minimum is -0.13 m\( \text{s}^{-1} \); the velocity magnitudes are shown on the colour bar, which has units of m\( \text{s}^{-1} \). The barrage wall/duct locations are shown at the upstream end of the downstream region of the flume, i.e. along the y-axis of the vector plot.

![Velocity Vectors at Duct Midheight](image)

*Figure 6.3: Velocity vectors at duct mid-height in downstream region*
Jets are again evident at 1D and 2D downstream, however the key difference with these results is that the maximum velocities are in line with the duct edges, not the duct centres. This therefore creates 7 annular jets. The lateral variation of streamwise velocity across the axis of these 3-D jets appears as 14 individual jets. By the 1D cross-section these individual profiles have partly merged between adjacent jets to appear as 6 central jets and two narrower edge jets. Seven jets are apparent at 2D, because the outermost jets on the left side of the flume (looking downstream) have merged and are forced towards the centreline by the eddy that forms; the general jet profile is still similar to that at 1D.

The low velocity at the duct centrelines is a result of the wake that forms downstream from the bulb bodies in the ducts; the body within the duct restricts the flow in the duct centre and forces the water to the duct edge. This creates a velocity profile downstream of the duct as shown in Figure 6.4. The jets at the duct edges merge with the adjacent duct jets, forming maximum flow between the duct outlets.

Reversed flow occurs at the flume edges, indicating eddy formation. These are also asymmetric, as shown by the width of the reversed flow. At 2D downstream the jets appear to spread towards the flume sides, causing quicker jet spreading. Due to this faster spreading there is no longer reversed flow at 5D, though the flow is still asymmetric.
The flow appears to become uniform across the flume closer to the barrage, i.e. at 10D rather than at 20D, than in the experiment with no turbines. This can be further assessed by analysing the streamwise velocity profiles.

![Velocity Profile at Duct Midheight](image)

*Figure 6.4: Streamwise velocity profile at duct mid-height*

### 6.2.2 STREAMWISE VELOCITY PROFILES

The streamwise velocities at 1D, 2D, 5D, 10D and 20D at duct midheight are shown in Figure 6.5. The barrage wall/duct locations are shown along the x-axis.

![Streamwise Velocity at Duct Midheight](image)

*Figure 6.5: Streamwise velocity at duct mid-height*
The seven wakes downstream from the bulbs and the eight jets are evident at 1D directly downstream from the duct exits, with maximum velocity approximately 0.31 m s\(^{-1}\), 400\% of the inlet velocity. As distance from the barrage increases and the flow tends towards the inlet conditions, velocity decreases until at 20D it is approximately 0.091 m s\(^{-1}\), which is 117\% of the inlet velocity. The jets also merge with distance, so that at 2D the jets are still evident but are not observed at 5D or beyond. An eddy is evident on the left side of the flume, with reversed flow occurring at 1D and 2D. The reversed flow on the left side of the flume reaches maximum width, 0.1 m, and magnitude, \(-0.13\) m s\(^{-1}\), at 1D and 2D; however, the flow on the right side is near zero at 2D, indicating the formation of a much smaller and weaker eddy. This asymmetry is due to the Coandă effect, but may also be due to the asymmetry in the flow caused by the swirl. The swirl leads to velocities in opposite cross-stream directions at different depths in the flume (assessed in Section 6.3.2), which are affected by the local boundaries in different ways. At the bed there is a rough boundary layer within close proximity of the jets, whereas the free surface is stress-less and further from the ducts, leading to a natural asymmetry in the swirl. Any variations in jet location throughout the depth may, therefore, be a result of the Coandă effect or the asymmetry in the swirl. As swirl increases the Coandă effect is expected to become less significant. At 5D downstream there is still some asymmetry though no jets or eddies are apparent. By 10D downstream the flow appears to be mostly uniform across the flume, with slightly higher velocity in the tank centre, which reduces further by 20D downstream. The flow appears to be uniform at 20D downstream; however, whether this applies across the depth can be assessed by analysing the vectors and profiles throughout the depth.
6.3 DEPTH-VARYING RESULTS

The velocity vectors were analysed in the \( z=0.04 \) m, 0.08 m, 0.12 m, 0.16 m and 0.185/0.175 m planes to determine the flow features present throughout the flume depth. The velocity vectors and streamwise velocity profiles in the 1D, 2D, 5D, 10D and 20D planes were also examined, in order to analyse the vertical velocity variation.

6.3.1 X-Y VELOCITY VECTORS IN HORIZONTAL PLANE

Figure 6.6 shows the \( x \)- and \( y \)-direction velocity vectors close to the bed, at 0.04 m. The experimental results show the streamwise and cross-stream velocities at 1D, 2D, 5D, 10D and 20D. (Note that the velocity vector colour scales vary between each plot, but the velocity magnitudes are assessed in the velocity profiles in Section 6.4.3).

Seven jets form downstream from the ducts exits, because at this depth there is no obstruction within the duct directly upstream which could result in a wake. The stators
create swirl in a clockwise motion when looking downstream, resulting in the cross flow towards the left at this depth. The flow is then pulled toward the left wall of the flume (at 5D and 10D), from a combination of the swirl and the Coandă effect. This also results in an eddy forming that extends up to 2D downstream. By 20D, the flow is almost uniform across the flume. Close to the duct midheight, at 0.08 m (Figure 6.7), the velocity vectors are similar to those at 0.075 m (Figure 6.3). Wakes form directly downstream from the bulbs, with jets at the duct sides. The horizontal velocities are near zero close to the barrage, because the swirl at this depth will only instigate vertical velocity. With distance downstream the jets spread towards the flume sides and the swirl and Coandă effect start to pull the jets to the right side of the flume, the opposite direction from that close to the bed. By 5D downstream jets are no longer evident and there is no reversed flow. By 20D downstream the flow appears almost uniform.

Figure 6.7: Velocity vectors at 0.08 m from the bed
At 0.12 m from the bed, close to the top of the ducts (0.13 m), the swirl forces the jets to the right with distance downstream (Figure 6.8). The jets merge a lot closer to the barrage at this depth and the cross-stream flow is stronger. There is only an area of accelerated flow at 5D downstream and the resulting flow downstream appears to reach uniformity by 20D.

![Velocity Vectors at 0.12 m from the Bed](image)

*Figure 6.8: Velocity vectors at 0.12 m from the bed*

The vectors at 0.16 m from the bed, shown in Figure 6.9 (overleaf), show strong cross flow, especially near the barrage. There is some reversed flow on both sides of the flume; on the left it forms the eddy apparent at other depths and on the right the flow is also reversed but at a lower velocity. With distance downstream, the higher velocity flow is forced to the right side of the flume by the swirl and the Coandă effect. The flow effects close to the surface (z=0.185 m) are similar to those immediately below, i.e. at 0.16 m and 0.12 m, with pronounced cross-stream flow close to the barrage and
streamwise flow at 5D downstream; these are shown in Figure 6.10 (also overleaf). There is a particularly strong reversed flow at 1D downstream across most of the flume, indicating the presence of vertical eddies. The direction of the Coandă effect at this depth is similar to that at 0.12 m and 0.16 m, because the swirl at the top of the ducts causes the flow to tend to the right side of the flume. As at all other depths the flow appears uniform at 20D downstream.

The variation of the flow across the depth, including the jet formation, asymmetry and swirl/Coandă effect direction can be seen in Figure 6.11, which shows the vectors close to the bed, centreline and surface.

The vertical and cross-stream eddies were assessed using the vertical and cross-stream velocities recorded with the 3-D probe. The velocity vectors in the 1D, 2D, 5D, 10D and 20D planes were calculated and they are analysed below. The velocity magnitudes, jet locations and distance to uniformity throughout the flume were assessed using the streamwise velocity profiles. The streamwise velocity profiles at 1D, 2D, 5D, 10D and 20D are included below.
Figure 6.9: Velocity vectors at 0.16 m from the bed

Figure 6.10: Velocity vectors at 0.185 m from the bed
Figure 6.11: Velocity vectors at 1D, 2D and 5D close to the bed (0.04 m), at the duct centreline (0.075 m) and close to the surface (at 0.185 m from the bed)
6.3.2 Y-Z VELOCITY VECTORS IN VERTICAL PLANE

The velocity vectors in the Y-Z plane at 1D are shown below in Figure 6.12:

Swirl is evident downstream from each duct exit, acting in a clockwise direction whilst looking downstream. This leads to flow across the flume towards the left side close to the bed and flow towards the right side of the flume close to the surface. An upwelling is present on the left side of the flume above the two outermost duct exits, due to this circulation throughout the flume cross-section. The upwelling was also observed in experiments. Vertical eddies are evident at the flume sides and the flow is seen to vary considerably throughout the depth. Maximum secondary (in y-/z-directions) velocity is 0.26 m s$^{-1}$ close to the surface, approximately 74% of the maximum streamwise velocity ($U_x$) at 1D, so the dominant flow direction is in the streamwise direction.
As distance downstream increases, to 2D (Figure 6.13), the upwelling moves across the flume due to the swirl effect. The swirl still forms directly downstream from each duct, though it has become less pronounced as the jets start to merge. Circulation throughout the full flume cross-section is present at this distance downstream and the vertical eddies have become more pronounced, shown by the strong down flow at the flume sides, with up flow inboard of this. The maximum velocity reduced to 0.16 m s\(^{-1}\), which is 62% of \(U_x\) at 2D and still occurs at the location of the upwelling.

![Figure 6.13: Velocity vectors at 2D downstream](image)

At 5D downstream, shown in Figure 6.14, the flow circulation is no longer directly downstream from each duct. The flow circulates throughout the entire section, creating strong cross-flow close to the bed and the surface, with minimal velocities in the flume centre. Close to the bed the higher speed flow moves towards the left side of the flume,
which was seen in the velocity vectors at 0.04 m (Figure 6.6). Similarly, from 0.12 m to 0.185 m the flow moves to the right of the flume, as seen in Figures 6.8-6.10.

The maximum velocity has reduced by 50 % by this distance downstream because the swirl has caused the jets to spread throughout the flume. The maximum velocities occur at the bed and the surface in the horizontal direction, causing large velocity variations throughout the flume. The maximum velocity is 42 % of the maximum streamwise velocity, so cross-stream and vertical velocities reduce with distance downstream.

![Velocity Vectors in Five D Plane](image)

*Figure 6.14: Velocity vectors at 5D downstream*

At 10D downstream (Figure 6.15), the cross-stream variation and full cross-section circulation are both still observed, however the velocities are significantly reduced. Close to the surface and bed the average cross-stream velocity ($U_y$) is 0.03 m s$^{-1}$, 20 % of maximum $U_x$, but close to the duct centres the mean $U_y$ is 0.015 m s$^{-1}$ (50 %
difference), so the variation throughout the flume is still significant. The horizontal $X$-$Y$ velocity vectors show that at 10D the velocity appears near uniform across the flume; however, the velocities in the $Y$-$Z$ plane show that there is large variation throughout the flume. If the velocities at this distance were depth-averaged, this cross-stream variation may be lost and the distance at which uniform flow occurs could be under predicted. This will be further examined using the velocity profiles in Section 6.4.3.

![Velocity Vectors at 10D downstream](image)

**Figure 6.15**: Velocity vectors at 10D downstream

There are several anomalies in the vertical velocities, especially close to the bed. These may be due to two factors: the probe configuration and the bubble dissipation. Firstly the 3-D ADV probe has four receivers which record the streamwise and cross-stream velocities using the Doppler shift, however there are only two receivers that detect the vertical velocities, so these may not be as accurate as the velocities in the other directions. Secondly, due to the addition of swirl in the flume and the increased
sampling frequency, micro-bubbles were added to the flume to improve the velocity detection; however, by 10D downstream the bubbles may have dissipated throughout the flume, leading to less accurate velocity recording at this distance. Another method of seeding could be used to reduce this effect, such as the addition of fine material to the water, but cannot be used with our equipment. These anomalies can also be seen at 20D downstream.

At 20D downstream (Figure 6.16) the cross-stream velocities are again reduced and are less significant; $U_y$ is approximately 10% of $U_x$ throughout the depth, except at the surface where the residual effects of the upwelling and swirl lead to slightly elevated cross-stream velocities. The reduced variation across the depth means that at this distance downstream depth-averaged modelling may be appropriate.

*Figure 6.16: Velocity vectors at 20D downstream*
The variation in the results as distance downstream increases, particularly the transition of circulation at the duct exits to circulation throughout the flume and flow deceleration, can be seen in Figure 6.17, which shows the Y-Z plane vectors at 1D, 2D and 5D.

![Velocity Vectors in One D, Two D and Five D Planes](image)

*Figure 6.17: Velocity vectors at 1D, 2D and 5D downstream*
6.3.3 STREAMWISE VELOCITY PROFILES

The variation of streamwise velocity throughout the depth was compared at 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.185 m/0.175 m from the bed, at 1D, 2D, 5D, 10D and 20D downstream. At 1D downstream from the barrage (Figure 6.18), the jets downstream from the ducts are evident, with seven jets close to the bed and eight jets where there is a wake from the bulb body close to duct midheight. The maximum velocity of 0.336 m s\(^{-1}\) occurs close to the bed. As distance from the bed increases the jets merge and the streamwise velocity becomes more uniform across the flume, with reversed flow over the majority of flume width close to the surface. The average reversed velocity is 0.05 m s\(^{-1}\) which is 15% of the maximum \(U_x\). The upwelling seen in the vectors causes slightly elevated streamwise velocity on the left side of the flume.

![Streamwise Velocity at One D Downstream](image)

Figure 6.18: Streamwise velocities at 1D downstream at different vertical, \(z\), locations
Asymmetry in the flow is seen, particularly in the eddies’ size and strength. On the left side of the flume (looking downstream) an eddy forms, with maximum reversed velocity of $0.12 \text{ m s}^{-1}$, $35\%$ of $U_x$, and maximum width of $0.125 \text{ m}$ at $0.16 \text{ m}$ from the bed. On the right side the eddy is only $0.03 \text{ m s}^{-1}$ strong, $35\%$ of $U_x$, and $0.09 \text{ m}$ width at $0.04 \text{ m}$; this shows that the flow profiles vary with cross-stream and vertical location.

The streamwise velocities further downstream at 2D are shown in Figure 6.19. The jets have started to merge at this distance downstream close to the bed, however closer to the surface the jets have become more defined, particularly at $0.16 \text{ m}$ from the bed. This is because of vertical spreading due to the swirl in the jet. Close to the surface the velocity starts to become positive. Also at this distance, the eddy on the left is still apparent and strong; however no eddy forms on the right, again showing this velocity variation and asymmetry.

![Streamwise Velocity at Two D Downstream](image)

*Figure 6.19: Streamwise velocities at 2D downstream at different vertical, $z$, locations*
The eddy on the left does not extend to 5D downstream, as shown in Figure 6.20, where velocity across the flume is in the positive streamwise direction:

*Figure 6.20: Streamwise velocities at 5D downstream at different vertical, z, locations*

The velocities at this distance do not exhibit jets, but slightly accelerated flow is evident on the right side of the flume; this moves further to the right with distance from the bed. The streamwise profiles appear to be becoming uniform across the flume, at approximately 0.1 m s\(^{-1}\) close to the bed and 0.06 m s\(^{-1}\) close to the surface, but the vectors in Figure 6.14 show that the cross-stream velocities vary significantly, so uniform flow is not achieved.

Further downstream the flow becomes more uniform, as seen in Figure 6.21, which shows the streamwise velocities at 10D.
Figure 6.21: Streamwise velocities at 10D downstream at different vertical, z, locations

There is still slight variation, with maximum velocity at the flume centre close to the bed and maximum velocity on the right close to the surface. The maximum velocity at 0.04 m is 0.155 m s\(^{-1}\) and at 0.185 m is 0.125 m s\(^{-1}\), so there is a 19 \% variation throughout depth. The mean velocity also varies by 43 \%, from 0.105 m s\(^{-1}\) at the bed to 0.06 m s\(^{-1}\) at the surface, so the flow has some depth variation (also exhibited in Y-Z vectors).

At 20D downstream, as seen in Figure 6.22, the variation has reduced. The maximum velocity only varies by 0.8 \% from 0.124 m s\(^{-1}\) to 0.125 m s\(^{-1}\) throughout the depth and the mean varies from 0.06 m s\(^{-1}\) to 0.081 m s\(^{-1}\), therefore by 26 \%. The low variation throughout the depth and the low cross-stream and vertical velocities, shown in Figure
6.16, mean that at this distance from the barrage depth averaged modelling may be appropriate.

Figure 6.22: Streamwise velocities at 20D downstream at different vertical, z, locations
6.4 COMPUTATIONAL SIMULATIONS – SWIRL AND DRAG

Several 3-D StarCCM+ models were created to simulate the experimental conditions as closely as possible in order to determine whether the flow produced by the bulb housing with stators can be accurately modelled. Firstly, the lift, or swirl, and drag imposed by the stators, calculated using flat plate theory, were used with a body force density (described in Chapter 4) to simulate the experimental conditions. Next, the constants in the body forces were fixed to give the correct swirl and head difference and were then fixed to give a comparable velocity profile close to the barrage. Lastly, the vane bodies themselves were added to the model. The bulb bodies were added to the ducts at the experimental bulb locations, as described in Chapter 4. Due to the increased complexity of the flow around the bulb, the mesh was refined from 1D upstream of the bulb to 1D into the downstream tank; the base cell size was 0.02 m and the refined cell size was 0.01 m, approximately 9% of the duct diameter. In order to incorporate swirl into the model, the momentum source within the ducts was altered using a field function. Firstly, the location that the swirl/lift, $C_{Fb,S}$, and drag, $C_{Fb,D}$, are added to the model was specified; the body force acts at all locations along the straight edge of the bulb, so that the momentum source covers several cells. The ‘if’ statement, Location, states that if the downstream position is between 0.245 m and 0.3137 m from the upstream end of the barrage, i.e. the stators location, then the momentum source is added; if not then there is zero swirl/lift and drag. The field function, Equation 43, is then used to specify the amount and direction of swirl within the ducts by applying a body force density, $F_b$. 

$$F_{b(x,y,z)} = \text{Location} \times [-C_{Fb,D}, -C_{Fb,S} \times (z - z_{\text{ref}}), C_{Fb,S} \times (y - y_{\text{ref}})] \quad (43)$$
The $C_{Fb,D}$ is imposed in the streamwise, $x$, direction and the $C_{Fb,S}$ is imposed in the cross-stream, $y$, and vertical, $z$, directions. These forces create a clockwise swirl when looking downstream. The lift and drag imposed on the flow (Figure 6.23) were calculated using the standard flat plate theory in fluid dynamics (Munson et al. 2009).

![Figure 6.23: Lift and drag imposed on fluid by inclined flat plate](image)

The force in each direction, $F$, is calculated using the following formula:

$$F = \frac{1}{2} \rho V^2 c A,$$  \hspace{1cm} (44)

where $\rho$ is the density, $V$ is the velocity over the plate, $c$ is the drag or lift coefficient and $A$ is the projected area of the plate. The density of the water is 1000 kg m$^{-3}$. The velocity over the plates was calculated using the discharge and the cross-sectional area across the duct; the velocity at the stators is 1.34 m s$^{-1}$. The drag coefficient of a laminar flat plate parallel to the flow is 0.001 and a 3-D flat plate perpendicular to the flow is 1.28 (Houghton and Carpenter 2013); using trigonometry and the incline angle of 30°, the drag coefficient $c_d$ of the stators was 0.64. The lift coefficient $c_L$ is equal to $2\pi\alpha$ where $\alpha$ is $\pi/6$ radians (Anderson 2010), which is 3.29. This angle is above that typically associated with stall and a low Aspect Ratio, but the value is idealised for an
example. The area of the stators is 0.015 m by 0.022 m, which is equal to a projected area of $1.65 \times 10^{-4}$ m$^2$. Using these values, the lift and drag over a single vane were calculated as 0.485 N and 0.095 N respectively. Twelve stators are included over each bulb in the model, which together impart 5.82 N and 1.14 N of lift and drag to the flow. The body force is applied over a unit volume, so the lift/swirl and drag per unit volume, where the volume is the region that the body force is specified over, are $C_{Fb,s}=35655$ N m$^{-4}$ (the swirl constant is multiplied by length from Equation 43) and $C_{Fb,D}=6985$ N m$^{-3}$. These values were added to Equation 43 to provide a momentum source at the bulb. The pressure drop across the barrage, swirl at the duct exits and the velocity profiles were calculated and compared with experimental results to determine whether the CFD model gives an accurate representation of the experimental conditions.

---

**6.4.1 FLOW CONDITIONS**

**6.4.1.1 HEAD DIFFERENCE**

Firstly, the head difference across the barrage was calculated using the surface pressure on the upstream and downstream lids. The drag value in the momentum source equation creates a pressure drop across the barrage; the average immediate upstream surface pressure is 545 Pa and the average immediate downstream surface pressure is -20 Pa, as seen in Figure 6.24a. The pressure drops by approximately 565 Pa across the barrage, which is equivalent to a head drop of 0.058 m; this is shown in Figure 6.24b (NB surface height calculation described in Chapter 5, Sections 5.1 and 5.2).

The experimental head difference was 0.018 m; the pressure drop, and therefore head difference, is over predicted by approximately 222%, so the drag constant must be reduced to give an accurate prediction of the surface pressures.
Secondly, the swirl at 3cm from the duct exits was calculated using both Equation 40, for non-dimensionalised circulation, and Equation 42 for non-dimensionalised angular momentum flux and compared with the experimental result. The results are shown in Table 6.2.

The average $\Gamma_{N\cdot D}$ in the computational model is 118, which is 156% of the experimental value. The average $AMF_{N\cdot D}$ value was 1.48, which has a 54% error when compared to the experiments, so the swirl calculated from the flat plate theory over predicts the swirl.
Table 6.2: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and swirl and drag computational model

<table>
<thead>
<tr>
<th></th>
<th>Duct</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>( I_{N-D} )</td>
<td>Expt</td>
<td>74.1</td>
<td>66.2</td>
<td>73.6</td>
<td>86.6</td>
<td>82.5</td>
<td>76.2</td>
<td>68.7</td>
<td>75.4</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>110.9</td>
<td>132.6</td>
<td>118.4</td>
<td>134.2</td>
<td>113.9</td>
<td>120.2</td>
<td>95.7</td>
<td>118.0</td>
</tr>
<tr>
<td>( AMF_{N-D} )</td>
<td>Expt</td>
<td>0.80</td>
<td>1.01</td>
<td>0.79</td>
<td>1.12</td>
<td>0.93</td>
<td>1.24</td>
<td>0.87</td>
<td>0.96</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>1.48</td>
<td>1.85</td>
<td>1.49</td>
<td>1.81</td>
<td>1.43</td>
<td>1.64</td>
<td>0.68</td>
<td>1.48</td>
</tr>
</tbody>
</table>

6.4.2 VELOCITY VECTORS

6.4.2.1 MIDHEIGHT X-Y PLANE

The duct midheight velocity vectors produced by the StarCCM+ model have some similarities with the experimental results (Figure 6.25): the vectors show jets forming at the sides of the duct exits, with a wake formed by the bulb in the centre of the duct and the imposed swirl, and as distance from the barrage increases the jets merge and spread, creating uniform flow throughout the flume. The wakes formed by the bulb bodies and swirl are, however, under predicted by the computational model; this was assessed using the streamwise velocity profiles.

The flow around the bulb body (only streamwise and cross-stream velocities) and the resulting jet profile (3-D flow) are shown in Figure 6.26. This shows that the wake at the right hand duct (bottom-most duct in Figure 6.25) is qualitatively similar to that exhibited in the experiments. The vectors show low, and in some areas reversed, velocities in the centre of the duct area. This variation across the ducts shows that
asymmetry forms very close to the barrage, before the Coandă effect starts; therefore, the swirl imparted by the stators creates asymmetrical effects.

Figure 6.25: StarCCM+ midheight velocity vectors with swirl and drag from flat plate theory

Figure 6.26: StarCCM+ midheight velocity vectors of right-hand duct wake, 

a) 2-D velocities around duct b) 3-D wake velocities
The jets spread very quickly and the combined effect of the swirl and Coandă effect occurs relatively close to the barrage, causing accelerated flow on both the left and the right side of the flume; the jets from the left-hand duct are pulled to the left of the flume, whilst the other jets are pulled to the right. This suggests that the Coandă effect overrides the effect of the swirl when in close proximity of the side wall, but closer to the centreline of the flume, or further inboard, where distance from the wall is higher the direction of the jet is dominated by the swirl. This split in the flow direction also leads to asymmetric eddies forming at the flume edges. The comparative size of the eddies and the accuracy of the flow profiles predicted were assessed using streamwise velocities at 1D, 2D, 5D, 10D and 20D.

6.4.2.2 Y-Z PLANES

The swirl downstream from the ducts and the jet spreading was analysed using the Y-Z velocity vectors at 1D, 5D and 10D downstream (Figure 6.27).

The swirl downstream from each duct is shown at 1D, with eddies forming at each side, particularly close to the surface, with strong cross stream flow close to the bed. Strong up-flow occurs on the left-hand-side of the flume due to the flow forced along the bed by the swirl. No comparable down-flow is formed on the right-hand-side of the flume, because the near-surface transverse velocity is lower than the near-bed transverse velocity. This flow asymmetry is due to the asymmetry in the boundaries; the free surface is stress-less and further from the ducts so the flow has dissipated and the bed has a boundary layer and is close to the swirling jets. As distance downstream increases, the swirling jets merge and spread throughout the duct, leading to cross-stream flow close to the bed and surface at 5D, as seen in the experimental vectors (Figure 6.16).
The flow is comparable on the left of the flume, but the flow on the right does not show this flow feature. At 10D the cross-stream flow has significantly reduced. The velocities predicted by the model and the flow differences at 5D will be compared with the experiments using the streamwise velocity profiles at the duct midheights.

![Figure 6.27: StarCCM+ velocity vectors at 1D, 5D and 10D downstream from duct exits](image)

6.4.3 STREAMWISE VELOCITY PROFILES

The midheight streamwise velocities at 1D, 2D, 5D, 10D and 20D from both the computational model and the experiments are shown in Figure 6.28.

At 1D the profiles are as expected from the velocity vectors: jets form downstream from the duct edges, where the flow is accelerated by the bulb bodies, and wakes form at the duct centres. The difference between the results is caused by the differing velocity magnitudes of the central wake; the experiments show near-zero or reversed velocities
at the duct centres, but the experimental velocities, especially at the flume centre, only drop to 0.1 m s\(^{-1}\). At the right side of the flume the profiles are comparable, as stated previously. The jets and wakes that form downstream from the two outermost ducts are of similar velocity magnitudes and locations. The maximum velocities in the two methods are, however, extremely similar, with only a 1 % error.

![Streamwise Velocity at Duct Midheight](image)

**Figure 6.28: Midheight streamwise velocities with swirl and drag from flat plate theory, StarCCM+ and Experimental**

As distance from the barrage increases, the jets spread and by 2D the jets are still evident but the profiles are very different: the maximum velocities are both approximately 0.25 m s\(^{-1}\), with a 2 % difference, but their locations are different. Also, the predicted eddies are dissimilar due to the increased jet spreading in StarCCM+ and the Coandă effect reducing the left-hand eddy size.
Further downstream, at 5D, 10D and 20D the flow profiles are comparable; the error in the maximum velocities are 5%, 2% and 13% respectively, though the locations of the maximum velocities differ except at 20D. However, the StarCCM+ model predicts a stronger Coandă effect on the left side of the flume, which leads to quicker jet dissipation and less variation across the width.

These results show that using the lift and drag values calculated from flat plate theory in a momentum source leads to over prediction of the pressure drop across the barrage, the swirl at the duct exits and the spreading rate of the flow in the downstream tank. The wakes directly downstream from the bulb bodies within the annulus of the jets are under predicted for the majority of the flume. The flow is reasonably well modelled after 10D downstream; however, the flow features close to the barrage may be modelled better by altering the swirl, $C_{Fb,S}$, and drag, $C_{Fb,D}$, constants in Equation 43. Alternative methods of representing the stators within the CFD model are considered in Sections 6.5 to 6.7.
6.5 COMPUTATIONAL SIMULATIONS – FIXED BODY FORCES

The swirl and drag constants in the momentum source body force were altered so that the swirl and head difference in the model were the same as in the experimental model. The drag constant was specified so that the pressure drop across the barrage was consistent with the experimental head difference and the downstream lid pressure was approximately 0, i.e. hydrostatic pressure. The swirl constant was altered so that the average swirl within the ducts is equal to the experimental value, using circulation and angular momentum flux calculations, and the swirl rotated in the clockwise direction when looking downstream. The swirl and drag constants per unit volume were $C_{Fb,S} = 18100 \text{ N m}^{-4}$ and $C_{Fb,D} = 1600 \text{ N m}^{-3}$ respectively. These were approximately 51 % and 23 % of the original lift and drag values from flat plate theory.

6.5.1 FLOW CONDITIONS

6.5.1.1 HEAD DIFFERENCE

The accuracy of the drag constant was assessed by examining whether the head difference across the barrage was the same as found in the experiments. Figure 6.29 shows the surface pressures and surface heights upstream and downstream of the barrage produced by a drag constant of 1600 N m$^{-3}$. The area-averaged upstream pressure was 173 Pa and the area-averaged downstream pressure was -15 Pa; the drag value of 1600 N m$^{-3}$ gives a pressure drop of approximately 188 Pa across the barrage, which is consistent with the head drop of 0.019 m. This head difference is evident in Figure 6.29b. The experimental head
difference was 0.018 m, so the computational result is within 6% of this value, therefore, the drag constant is reasonably accurate for this case.

Figure 6.29: StarCCM+ upstream and downstream lids, a) Pressure b) Surface Heights

6.5.1.2 SWIRL

The non-dimensionalised circulation and angular momentum flux at 3 cm from the duct exits were calculated and compared with the experimental results (Figure 6.3). The average non-dimensionalised circulation is 64.7. This value is within 14% of the experimental circulation at the outer edge of the duct, however, the circulation only analyses the flow at the duct edge and does not incorporate the streamwise velocity. The angular momentum flux gives a more accurate representation of the swirl across the whole duct area (Table 6.3). The average \( AMF_{N-D} \) is 0.95. The average experimental
AMF\(_{N-D}\) is 0.96, so the computational value is within 1 % of the experimental and is accurate. However, the integral is taken over the projected area of the duct and the radial variation is quite different from the experimental result, so the wake profiles may also differ from the experiments. The velocity vectors at duct midheight, 1D, 5D and 10D and the streamwise velocity profiles were compared with the experimental results.

<table>
<thead>
<tr>
<th>Duct</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\Gamma_{N-D}) Expt</td>
<td>74.1</td>
<td>66.2</td>
<td>73.6</td>
<td>86.6</td>
<td>82.5</td>
<td>76.2</td>
<td>68.7</td>
<td>75.4</td>
</tr>
<tr>
<td>(\Gamma_{N-D}) Comp</td>
<td>61.7</td>
<td>69.1</td>
<td>57.9</td>
<td>67.6</td>
<td>57.8</td>
<td>69.2</td>
<td>69.8</td>
<td>64.7</td>
</tr>
<tr>
<td>(AMF_{N-D}) Expt</td>
<td>0.80</td>
<td>1.01</td>
<td>0.79</td>
<td>1.12</td>
<td>0.93</td>
<td>1.24</td>
<td>0.87</td>
<td>0.96</td>
</tr>
<tr>
<td>(AMF_{N-D}) Comp</td>
<td>0.85</td>
<td>1.06</td>
<td>0.84</td>
<td>1.08</td>
<td>0.84</td>
<td>1.08</td>
<td>0.92</td>
<td>0.95</td>
</tr>
</tbody>
</table>

*Table 6.3: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and fixed body force computational model*

### 6.5.2 VELOCITY VECTORS

#### 6.5.2.1 MIDHEIGHT X-Y PLANE

The duct midheight velocity vectors produced by the StarCCM+ model are shown in Figure 6.30. The vectors show jets forming at the sides of the duct exits; however, unlike the model with flat plate theory lift and drag, in this model the jets cover the entire duct diameter, with only a slight decrease in velocity as a result of the bulb body.

As distance from the barrage increases the jets merge and spread, as before, but the Coandă effect causes the jets to all bend to the left hand side of the flume, suggesting that in this instance the Coandă effect is stronger than the swirl, which has been reduced with a lower swirl coefficient. This also causes the eddies that form close to the barrage
on either side of the flume to be larger and stronger, which is closer to the experimental conditions.

![Image of velocity vectors](image)

*Figure 6.30: Midheight velocity vectors with swirl and drag from flat plate theory, StarCCM+*

6.5.2.2 Y-Z PLANES

The swirl downstream from the ducts and the jet spreading was analysed using the Y-Z velocity vectors at 1D, 5D and 10D downstream (Figure 6.31).

The swirl downstream from each duct is reduced in this model, because the swirl and drag parameters are reduced. As a result of this there is no full cross-section circulation at either 5D or 10D, which is dissimilar to the experiments. This shows that the cross-stream and vertical velocities are not well predicted by the model. The accuracy of the streamwise velocities was analysed using the streamwise velocity profiles at the duct midheights.
Figure 6.32 shows the duct midheight streamwise velocities at 1D, 2D, 5D, 10D and 20D from both the computational model and the experiments. Close to the barrage the velocity profiles are very different, as shown in the velocity vectors. The locations of the jets are exactly the opposite of those in the experiment, with peak velocities forming directly downstream from the ducts rather than downstream from the gaps between the ducts. The flow around the bulbs converges towards the pipe centreline as distance from the bulb increases, meaning that the flow is not forced to the outside of the duct, as it is in the experiments where the annular wake is sustained. Even though the swirl is similar to the experiments, shown by the angular momentum flux values, the distribution of the flow is very different. The higher streamwise velocities close to the duct centres result in similar angular momentum flux values to the high rotational velocities at the duct edges, but result in recovery of the streamwise flow immediately downstream from the bulb.
The spreading rate of the jets is also over-predicted, as it was in the flat plate model; however, the eddies at the tank edges are better predicted by this model. As distance from the barrage increases the streamwise velocity profile becomes more comparable to the experimental profile, although, the predicted eddies extend further than the experimental results, so the 5D profile is less accurate than the flat plate theory model. The velocities at the flume centre and left side are well represented at 10D and 20D, but the velocity on the right is under predicted, due to the low velocity region downstream from the eddy and the strong Coandă effect to the left side of the flume.

There are several problems with both the flat plate theory model and the fixed swirl and drag model. The model with lift and drag calculated from flat plate theory over predicts both the head difference over the barrage and the swirl in the ducts, which is not likely...
to alter with mesh refinement or different turbulence models. The erroneous jet profiles at 1D and 2D in the fixed swirl and drag model may be affected, so a parametric study of the model was conducted.

6.5.4 PARAMETER STUDY

Two problems with the model should be improved: firstly, convergence is not guaranteed, because the residuals in the model only drop to 5×10⁻³ and secondly, the velocity profiles predicted by the model are inaccurate. In order to remedy this, different turbulence models were used and the results compared: the standard $k-%e$ previously analysed, the realisable $k-%e$ and the $k-%o$ SST model (described in Chapter 4).

6.5.4.1 REALISABLE K-E MODEL

The realisable model improves the convergence, with residuals dropping below 1×10⁻⁴ for the $x$, $y$ and $z$-momentum and 5×10⁻⁴ for the continuity residuals. The resulting velocity profiles are not, however, improved; Figure 6.33 shows that on the right side and centre of the flume the realisable profile is the same as the standard $k-%e$ profile and still very different from that shown in the experiments. On the left side of the flume the profile varies from the standard result, particularly close to the barrage, with a reduced outermost jet and larger eddy that extends to 5D downstream. Though the convergence is improved, the results are less comparable to the experiment than the standard $k-%e$ model.
6.5.4.2 K-Ω SST MODEL

The $k-\omega$ SST model was also used and the results compared. The convergence was poor for this model and only dropped to $1 \times 10^{-3}$ for the $x$, $y$ and $z$-momentum and $5 \times 10^{-3}$ for the continuity residuals. The velocity profiles are also as inaccurate as those in the realisable model (Figure 6.34). The right hand eddy size and strength is particularly over predicted, from 1D to 5D, though further from the barrage the results show better correlation with the experiments than the standard $k-\varepsilon$ model.

The different turbulence models are found to affect the results, though the model that produces results most similar to the experiment is the standard $k-\varepsilon$ model.

Figure 6.33: Midheight streamwise velocities with fixed body force, Experimental, Standard $k-\varepsilon$ and Realisable $k-\varepsilon$
Figure 6.34: Midheight streamwise velocities with fixed body force,

Experimental, Standard k-ε and k-ω SST
6.6 COMPUTATIONAL SIMULATIONS – HIGH SWIRL CONSTANT, $C_{FB_S}$

In the previous models, reducing the swirl constant caused the jets to merge behind the bulb body, the eddies to be larger and the Coandă effect to be stronger. In order to determine how the results were affected by increasing the swirl coefficient, the value calculated from the swirl and drag calculation in Section 6.4 was doubled and the results analysed. The swirl from Section 6.4 was doubled to $C_{FB_S} = 71310 \text{ N m}^{-4}$, but the drag was kept as $C_{FB_D} = 1600 \text{ N m}^{-3}$.

6.6.1 FLOW CONDITIONS

6.6.1.1 HEAD DIFFERENCE

The drag constant that produced the correct head difference previously, 1600 N m$^{-3}$, was used in the model. Figures 6.35 and 6.36 show the surface pressures and surface heights upstream and downstream of the barrage. The immediate average upstream pressure was 227 Pa and the average immediate downstream pressure was -4 Pa; the drag value of 1600 N m$^{-3}$ gives a pressure drop of approximately 231 Pa across the barrage, which is equivalent to a head drop of 0.0235 m. This head difference is evident in Figure 6.36.

The experimental head difference was 0.018 m, so the computational result is within 31% of this value. Increasing the swirl by 50% increases the drag error by 25%, which leads to greater inaccuracy.
The swirl from Section 6.4 was doubled to 71310 N m$^{-4}$. The circulation, $\Gamma_{N-D}$, across each of the ducts is reasonably constant (Table 6.4) and the average circulation is 147.6. This value has an error of 96% from the experimental circulation; however, the angular momentum flux may give a more accurate representation of the swirl across the whole duct area. The average $AMF_{N-D}$ is 1.05, which is within 9% of the experimental and is fairly accurate. The angular momentum flux is proportional to both the streamwise and the tangential velocity; the experimental $U_x$ is double $U_\theta$ (approximately 0.3 m s$^{-1}$ and 0.15 m s$^{-1}$ respectively), but the computational $U_x$ is approximately equal to $U_\theta$ (0.2 m s$^{-1}$), which leads to equivalent angular momentum flux results for both the
experiment and the computation, though the tangential velocity is over predicted. The head difference and tangential velocity are both over predicted using this swirl constant, but the velocity vectors and streamwise velocity profiles may be more accurate with this higher value of swirl.

<table>
<thead>
<tr>
<th>Duct</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Gamma_{N,D}$</td>
<td>Expt</td>
<td>74.1</td>
<td>66.2</td>
<td>73.6</td>
<td>86.6</td>
<td>82.5</td>
<td>76.2</td>
<td>68.7</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>129.0</td>
<td>142.1</td>
<td>153.5</td>
<td>167.2</td>
<td>158.4</td>
<td>152.4</td>
<td>130.6</td>
</tr>
<tr>
<td>AMF</td>
<td>Expt</td>
<td>0.80</td>
<td>1.01</td>
<td>0.79</td>
<td>1.12</td>
<td>0.93</td>
<td>1.24</td>
<td>0.87</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>0.71</td>
<td>0.82</td>
<td>1.21</td>
<td>1.26</td>
<td>1.08</td>
<td>1.49</td>
<td>0.75</td>
</tr>
</tbody>
</table>

*Table 6.4: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and high swirl constant computational model*

6.6.2 VELOCITY VECTORS

6.6.2.1 MIDHEIGHT X-Y PLANE

The computational velocity vectors close to the barrage are much closer to those measured in the experiments, as shown in Figure 6.37. The high amount of swirl causes the flow to be forced to the duct edges much more than in previous models, leading to jets at the duct edges and wakes forming downstream from the bulb bodies; these are particularly evident in the close up of the left outermost ducts (Figure 6.37b). The jets spread very quickly, so the eddies at the flume edges do not appear to form, and the combined effect of the swirl and Coandă effect occurs relatively close to the barrage, causing accelerated flow on the right side of the flume. The flow also appears to become uniform across the flume much closer to the barrage than in the previous
computational models. These flow features were also assessed using the $Y$-$Z$ velocity vectors and whether eddies form and at what distance the flow becomes uniform across the tank was assessed using the streamwise velocity profiles at 1D, 2D, 5D, 10D and 20D.

Figure 6.37: Midheight velocity vectors with high swirl constant, $C_{Fb,S}$.

a) Full flume b) Left ducts
6.6.2.2 Y-Z PLANES

The swirl downstream from the ducts and the jet spreading was analysed using the Y-Z velocity vectors at 1D, 5D and 10D downstream (Figure 6.38).

Swirl is present downstream from each duct, with strong cross-flow at the bed and close to the surface at 1D; these vectors are in close agreement with the experiments, though the location of the upwelling is slightly different. Further downstream, at 5D, the full flume circulation present in the experiments is evident here, though the flow on the right side of the flume is weaker. There is also some circulation at 10D downstream, but the cross-flow is minimal. These results suggest that the cross-stream and vertical velocities are more accurately predicted using a higher swirl constant. The accuracy of the streamwise velocities was assessed by comparison of the lateral and vertical profiles.
6.6.3 STREAMWISE VELOCITY PROFILES

The streamwise velocities are shown in Figure 6.39. At 1D the profiles are fairly similar, with wakes forming at the duct centres and jets forming between the duct exits, as shown in the velocity vectors. The StarCCM+ model shows more merging of the jets between the ducts and, therefore, a slightly reduced peak velocity with an 8% error. The profiles at the flume edges are very different. The experimental results show eddies forming, whereas the computational results have stronger cross-stream flow and flow in both the positive and negative streamwise direction. This is due to the increased spreading from the high amount of swirl.

As distance from the barrage increases, the jets spread and by 2D the computational streamwise flow appears near uniform across the flume, whereas the experiment still shows jets. The profiles at 2D are very different; the jets and eddies in the experiment are not shown in the computation because the flow spread too quickly and the maximum velocity has a 15% error. Further downstream, at 5D, 10D and 20D the flow profiles are comparable; however, the StarCCM+ model predicts a stronger combined swirl and Coandă effect to the right side of the flume and, therefore, greater asymmetry.

These results show that increasing the swirl coefficient leads to wakes forming behind the bulb bodies and overspreading of the jets, causing no eddies to form at the flume edges and a reduced Coandă effect. The flow within 1D is reasonably well modelled in all three velocity directions; however, the other flow features may be modelled better by modelling the real vane bodies.
Figure 6.39: Midheight streamwise velocities with fixed body force, Experimental and Standard k-ε with high swirl constant, $C_{Fb,S}$.
The results predicted using body forces did not produce accurate results for the velocity vectors and streamwise velocity profiles, so the vane geometries were added to the bulb bodies within the ducts, as shown in Figure 6.40; the left hand image shows the bulb/stators in plan view and the right hand image shows the bulb/stators when looking upstream within the duct:

![Figure 6.40: Bulb body with stators in StarCCM+](image)

The stator vanes were inclined at 30° to the streamwise direction, causing swirl in the clockwise direction when looking downstream. The mesh required further refinement from the conditions with only a bulb body, due to the complexity of the flow around the blades and the resulting swirl flow. The base cell size was 0.02 m; from 1D upstream of the bulb to 1D downstream of the barrage the cell size was 0.004 m, 20% of the base cell size; from 1D downstream to 2D the cell size was 0.008 m, 40% of base; and up to 5D downstream the cell size was 0.01 m, 50% of base.
6.7.1 FLOW CONDITIONS

6.7.1.1 HEAD DIFFERENCE

The average pressures immediately upstream and downstream of the barrage were 5 Pa and -252 Pa respectively (Figure 6.41a); therefore the pressure difference over the barrage was 257 Pa. This gave a head of 0.026 m (Figure 6.41b), which is within 44% of the experimental value of 0.018 m.

![Pressure Distribution](image1.png)

*Figure 6.41: StarCCM+ upstream and downstream lids, a) Pressure b) Surface Heights*

6.7.1.2 SWIRL

The $\Gamma_{N-D}$ and $AMF_{N-D}$ are shown in Table 6.5. The computational result has a 1% error when compared with the experimental result, so is highly accurate. The angular momentum flux has an average value of 1.03. This is within 7% of the experimental results, so the swirl predicted using angular momentum flux assessment is also accurate.
<table>
<thead>
<tr>
<th>Duct</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Gamma_{N-D}$</td>
<td>Expt</td>
<td>74.1</td>
<td>66.2</td>
<td>73.6</td>
<td>86.6</td>
<td>82.5</td>
<td>76.2</td>
<td>68.7</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>66.2</td>
<td>90.8</td>
<td>68.7</td>
<td>87.2</td>
<td>66.1</td>
<td>74.2</td>
<td>68.0</td>
</tr>
<tr>
<td>$AMF_{N-D}$</td>
<td>Expt</td>
<td>0.80</td>
<td>1.01</td>
<td>0.79</td>
<td>1.12</td>
<td>0.93</td>
<td>1.24</td>
<td>0.87</td>
</tr>
<tr>
<td></td>
<td>Comp</td>
<td>0.85</td>
<td>1.30</td>
<td>0.97</td>
<td>1.18</td>
<td>0.90</td>
<td>1.10</td>
<td>0.94</td>
</tr>
</tbody>
</table>

Table 6.5: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and vane bodies computational model

6.7.2 VELOCITY VECTORS

6.7.2.1 MIDHEIGHT X-Y PLANE

The velocity vectors at duct midheight (Figure 6.42) show weak flow forming directly downstream from the bulb bodies, indicating that the velocity profiles may be similar to the experimental results, with wakes at the duct centres. The eddies appear to be more inboard than those in the previous computational models, causing the outermost jets to be reduced and a larger area of low velocity to form downstream from the eddies. Uniform flow occurs approximately halfway down the flume, though this can be assessed using the streamwise velocity profiles.

Figure 6.42: Midheight velocity vectors with vane bodies
The velocity vectors produced by the model (Figure 6.43) are comparable to those in the experiment, with reasonable results at 1D, 5D and 10D. Close to the barrage the swirl downstream from the ducts is apparent, with an upwelling in a similar location to that observed in the experiments. The flow along the bed is less strong in this simulation, but the swirl still leads to circulation throughout the full flume cross-section at 5D. There are low velocity eddies throughout the vertical at 5D, as seen in the experimental results, and at 10D there is slight circulation throughout the flume, with near-zero velocities in the centre at duct midheight.

*Figure 6.43: StarCCM+ velocity vectors at 1D, 5D and 10D downstream from duct exits*
6.7.3 STREAMWISE VELOCITY PROFILES

The streamwise velocity profiles in Figure 6.44 show that the jet profiles directly downstream from the ducts are better predicted by this model than the model with fixed swirl and drag, with reduced streamwise velocity occurring at the duct centres. The velocity does not reduce to near-zero as it does in the experiments, but the profile is more comparable with maximum velocities, only 13% over predicted.

The results were not compared to the case with high swirl constant, because that model under predicts the streamwise velocity and over predicts the tangential velocity, so is less comparable with the experimental results. The eddies predicted at the flume edges are larger than in the experiments due to increased merging of the outermost jets and the inboard location of the eddies.

At 2D downstream the jet locations are closer than the momentum source model, but are still different from the experiments and the eddies are over predicted. As distance from the barrage increases the results become more comparable, until at 10D and 20D the profiles are very similar, with only 14% and 0.9% errors in the maximum velocities respectively.
Figure 6.44: Midheight streamwise velocities

Experimental, Standard k-ε with fixed swirl and drag momentum source and with vanes
6.8 CONCLUSIONS

Three main issues were addressed in this chapter:

1. How does velocity vary with depth and distance downstream?
2. How well this is predicted by generally available 3-D CFD?
3. At what distance downstream is 2-D depth-averaged modelling justified?

1. The experimental flow conditions described show that the head difference is optimal for energy generation and the water level drops with distance from the barrage, tending toward normal flow, which is the same as the open duct flow conditions. The water height drops 40% more in the stator experiments, due to swirl-induced losses.

The swirl created by the stators causes wakes to form downstream from the bulb bodies and jets to form at the duct edges; this is very different from the velocity profile of the open ducts, which had normal jet profile. The jets at the duct edges also merge quicker than those in the open duct experiment, leading to more uniform flow at 5D.

Asymmetry due to the Coandă effect and the swirl creates strong cross flow close to the bed in one direction and close to the surface in the opposite; this is particularly evident at 1D and 2D downstream. At this distance the swirl occurs downstream from each duct exit, whereas further downstream, at 5D and 10D, circulation throughout the full cross-section of the flume occurs. At 20D the cross-stream and vertical flow is near-zero, and the streamwise flow is nearly uniform across the flume, suggesting that at this distance depth-averaged modelling would be appropriate.
2. Six models were compared with the experimental results, with respect to the head difference, circulation, angular momentum flux, velocity vectors, velocity profiles and convergence; a summary of the results is shown in Table 6.6.

The simulation with the most accurate head difference and angular momentum flux, indicating the most accurately modelled drag and swirl, was the \( k-\varepsilon \) model with fixed body forces. The realisable \( k-\varepsilon \) model has the best convergence of the fixed models. The velocity vectors and velocity profiles are, however, poorly predicted. The duct profile does not have a wake at the duct centre downstream from the bulb body, so the close to barrage profile is inaccurate. The profiles only become comparable at 20D.

The simulation with the most accurate close to barrage velocity profile is the model with high swirl constant, though the tangential velocity is over predicted using this model, so the circulation is too high even though the angular momentum flux is comparable to the experiments. The high amount of swirl leads to over spreading of the jets and reduced Coandă effect at 2D. This causes inaccurate velocity profiles at 5D and 10D, but by 20D the model is accurate again.

StarCCM+ does not accurately predict the flow created by the bulb bodies and stators, because it cannot simulate the swirl downstream from the ducts exits accurately. The model tends to over predict the jet merging downstream from the bulb body when the swirl within the duct is accurately modelled and over predict the tangential velocity and spreading rate of the jets when the profile immediately downstream from the ducts is simulated.
<table>
<thead>
<tr>
<th>Flow Features</th>
<th>Expt</th>
<th>Flat plate theory</th>
<th>Fixed body force</th>
<th>Fixed body force</th>
<th>Fixed body force</th>
<th>Fixed body force</th>
<th>Vane bodies</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.018</td>
<td>0.058, + 322%</td>
<td>0.019, - 6%</td>
<td>0.017, - 6%</td>
<td>0.017, - 6%</td>
<td>0.024, + 31%</td>
<td>0.026, + 45%</td>
</tr>
<tr>
<td>Head (m), % error</td>
<td>75.4</td>
<td>118, + 56%</td>
<td>64.7, - 14%</td>
<td>67.7, - 10%</td>
<td>67.6, - 10%</td>
<td>147.6, + 96%</td>
<td>74.4, - 1%</td>
</tr>
<tr>
<td>Circulation, % error</td>
<td>0.96</td>
<td>1.48, + 54%</td>
<td>0.95, - 1%</td>
<td>0.96, 0%</td>
<td>0.95, - 1%</td>
<td>1.05, + 8%</td>
<td>1.03, + 7%</td>
</tr>
<tr>
<td>Angular Momentum Flux, % error</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Velocity Vectors</td>
<td></td>
<td>Swirl ducts at 1D and flume at 5D</td>
<td>Swirl at ducts at 1D</td>
<td>Swirl at ducts at 1D</td>
<td>Swirl at ducts at 1D</td>
<td>Swirl at ducts at 1D and across flume at 5D</td>
<td>Swirl at ducts at 1D and across flume at 5D</td>
</tr>
<tr>
<td>Profile Correlation</td>
<td>1D</td>
<td>Jets at edges, slight wake at centre</td>
<td>Poor jet location</td>
<td>Poor jet location</td>
<td>Poor jet location</td>
<td>Jets at edges, wake at centre</td>
<td>Poor jet location</td>
</tr>
<tr>
<td></td>
<td>2D</td>
<td>Poor jet location</td>
<td>Poor jet location</td>
<td>Poor jet location</td>
<td>Poor jet location</td>
<td>Over predicts spreading</td>
<td>Poor jet location</td>
</tr>
<tr>
<td></td>
<td>5D</td>
<td>Poor at edge</td>
<td>Over predicts eddy length</td>
<td>Over predicts eddy length</td>
<td>Over predicts eddy length</td>
<td>Poor at edge</td>
<td>Over predicts eddy length</td>
</tr>
<tr>
<td></td>
<td>10D</td>
<td>Good</td>
<td>Poor at edge</td>
<td>Poor at edge</td>
<td>Good</td>
<td>Poor at edge</td>
<td>Good</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
</tr>
<tr>
<td>Convergence</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x, y, z</td>
<td></td>
<td>1x10^-4</td>
<td>5x10^-3</td>
<td>5x10^-4</td>
<td>5x10^-3</td>
<td>1x10^-4</td>
<td>5x10^-3</td>
</tr>
<tr>
<td>Continuity</td>
<td></td>
<td>1x10^-4</td>
<td>5x10^-3</td>
<td>1x10^-4</td>
<td>1x10^-3</td>
<td>1x10^-4</td>
<td>5x10^-3</td>
</tr>
</tbody>
</table>

Table 6.6: Summary of computational results
3. Depth-averaged modelling is not applicable up to 20D, because the swirl in the ducts cannot be modelled, but depth-averaged modelling could be used after 20D, where there is no swirl. This has not been fully investigated, but there is little variation in the velocity profiles with depth at 20D and the asymmetry is negligible.
7. RESULTS- BULB HOUSING WITH STATORS AND ROTORS

The addition of bulb housings with inclined stators (Chapter 6) was made to assess the effect of a stationary component creating swirl in the flow; however, in order to model a tidal barrage turbine more accurately, a rotary component was also required. Rotors were added to the bulb bodies, downstream from the stators (described in Chapter 3). The bulb provided a central blockage to the flow causing acceleration at the duct edges, which occurs in barrage operation. The stators provide a swirl component to the flow, which directs the flow onto the rotors, which are free to turn with some friction resistance. These in turn provide a swirl component to the flow, imparted by the rotary motion of the rotors. The effect of the rotors on the flow field and how this compares to that caused by the stators alone was examined.

Experiments were conducted with seven bulb bodies with inclined stators and free-moving rotors. To achieve maximum rotor rotation speed, the discharge, and thus the inlet velocity, was increased; this caused the water heights upstream and downstream of the barrage to also increase. The water heights downstream were measured, as in the experiments with no bulb/stator representation and bulb/stators only. The three-component velocities were recorded throughout the length, breadth and depth of the flume using the 3-D ADV at the same locations as the stators only experiment. The flow rate, inlet velocity, water level variation and swirl were assessed for comparison with the previous experiments. The velocity vectors and streamwise velocity profiles throughout the flume were also analysed to determine the velocity field and this was compared with the bulb/stator results.
Three computational methods were tested for accuracy when compared with the rotor results: two body force models used in the stators only assessment and a fan momentum source model. The experimental methods are described in Chapter 4.

7.1 FLOW CONDITIONS

7.1.1 FLOW RATE AND INLET VELOCITY
The flow rate was increased so that the rotors were running at maximum rotation frequency (Chapter 3) without the water overtopping the barrage. The inlet discharge, \( Q \), was 0.0383 m\(^3\) s\(^{-1}\), the average upstream height close to the barrage, \( h_1 \), was 0.287 m, giving an average upstream inlet velocity, \( U_{in} \), of 0.1095 m s\(^{-1}\). The average height immediately downstream, \( h_2 \), was 0.233 m, giving an average head difference across the barrage, \( H \), of 0.054 m. This corresponds to a head difference of 7.71 m at full scale, which is within the range of operation for the La Rance barrage (de Laleu 2009). The flow rate and head difference are greater than in previous experiments, because higher velocities were required to overcome the higher than desirable mechanical friction in the rotors to achieve maximum rotation rate. The water depth was higher than in previous experiments, so the velocities were recorded at depths of 0.04 m, 0.08 m, 0.12 m, 0.16 m and close to the surface at 0.2 m. At 20D the close-to-surface velocities were recorded at 0.195 m, because the depth had decreased due to swirl-induced losses.
7.1.2 WATER LEVEL VARIATION

The experimental water depths downstream of the barrage at 1D, 2D, 3D, 5D, 10D, 15D and 20D were recorded on one half (right side looking downstream) of the flume and at the centre of the flume and were compared with the previous results (Figure 7.1).

The depths in all cases decreased with distance downstream of the barrage. The water depth for the bulb turbines with stators and rotors (Props Right and Props Centreline) are higher than the previous experiments, because the flow rate is higher. The rate that the water depth decreases, however, is similar to the previous experiments. The water level at the barrage, with stators and rotors was 0.233 m and at 2.2 m from the barrage was 0.22 m, a drop of 0.013 m; this is similar to the drop of 0.0147 m when only stators/vanes were included in the model and but 30% more than the 0.01 m drop observed for the open duct experiments.

The rotor results show a drop in the water depth immediately downstream from the barrage at the flume centreline due to the formation of high velocity jets and swirl that occur at the duct exit locations. The velocity at the flume edge, however, is reduced so the water is deeper. With distance downstream the water levels across the flume become more uniform.
7.1.3 SWIRL

The swirl produced in the experiments has two components: that produced by the stators and that produced by the rotors. The stators direct the flow into the rotors to drive them and the resulting flow is still in the clockwise direction when looking downstream. The blades are set at an average slope of 23°; the La Rance blades are at -5° to 35° pitch, so the rotor is within this range. The rotation speed of each rotor was measured as 141, 135, 150, 127, 141, 153, and 122 rpm, with an average of 138 rpm. The La Rance turbines operate at 93.75 rpm with a maximum overspeed of 260 rpm, which when scaled gives a range of 1121-3109 rpm; these speeds cannot be achieved in the scale model due to the friction in the system. Higher velocities are required to initiate turbine
movement and overcome the friction at the rotor shaft, so the rotational speeds that are produced may not be representative of the full-scale barrage turbines.

The swirl was assessed using the circulation, \( \Gamma_{N-D} \), and angular momentum flux, \( AMF_{N-D} \), described in Chapter 6 (Equations 40 and 42). The duct diameter, \( d \), was 0.11 m and the inlet velocity for the stators was 0.0783 m s\(^{-1}\) and for the rotors was 0.1095 m s\(^{-1}\). The circulation and angular momentum flux were calculated at 1D downstream from the barrage rather than at 3 cm previously evaluated in Chapter 6; this was due to an error in the data collection, so the 3-component velocities at 3 cm were not available for analysis. The swirl was analysed at 1D downstream and any comparisons with the swirl produced by the stators was also assessed at 1D downstream.

The circulations are shown in Table 7.1. The average circulation across the ducts at 1D with bulbs and stators only was 86.4; the rotor result is 120.2, 39 % greater than this, so the non-dimensionalised circulation increases with the addition of rotors and increased inlet velocity.

The non-dimensionalised \( AMF \) was 0.71 for the bulbs/stators and 1.00 for the bulbs/stators/rotors; the rotors result is within 41 % of the stators result, so the swirl at 1D downstream has increased with the addition of rotors. Whether this increased swirl affects the flow field was assessed from the velocity vectors and velocity profiles downstream in the flume.
Table 7.1: Table of non-dimensionalised circulation and angular momentum flux in bulb/stator experiment and bulb/stator/rotor experiment at 1D

<table>
<thead>
<tr>
<th>Bulbs/ Stators</th>
<th>Bulbs/ Stators</th>
<th>Bulbs/ Stators</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bulbs/Stators</td>
<td>89.2 110.8 111.7 125.0 135.7 152.9 116.2</td>
<td>Rotors</td>
</tr>
<tr>
<td>Bulbs/Stators</td>
<td>-     -     -     -     -     -     -     -</td>
<td>Rotors</td>
</tr>
<tr>
<td>Bulbs/Stators</td>
<td>0.67 1.04 0.85 0.76 1.24 1.69 0.78 1.00</td>
<td>Rotors</td>
</tr>
</tbody>
</table>

7.2 DUCT MID-HEIGHT RESULTS

The experimental streamwise and cross-stream velocities were measured at the duct mid height, \( z = 0.075 \) m. The velocity vectors and the streamwise velocity profiles are shown below.

7.2.1 VELOCITY VECTORS

The experimental velocity vectors at 1D, 2D, 5D, 10D and 20D at the duct mid height are shown in Figure 7.2. The maximum velocity in the flume is 0.60 m s\(^{-1}\) and the minimum is -0.13 m s\(^{-1}\); the colour bar shows the velocity magnitudes throughout the flume. The barrage wall/duct locations are shown at the upstream end of the downstream region of the flume, i.e. along the \( y \)-axis of the vector plot.
The velocity vectors agree closely with those shown in the stators only experiment (Figure 6.3), though the maximum velocity is approximately double that in the stators only experiment, due to the increased inlet velocity and swirl. There are wakes downstream from the bulb turbines and jets at the duct edges at 1D and 2D. As distance from the barrage increases the jets merge and become uniform across the flume at 20D downstream. The maximum reversed velocity, presumably at the eddy edge, is the same in both the experiments, though the eddy appears to extend further downstream when the velocity is increased. This will be assessed fully using the velocity profiles. The region of elevated velocity at 5D and 10D appears to be further over to the left side of the flume than that in the stator experiment. This is assessed by comparison of the velocity profiles.
7.2.2 STREAMWISE VELOCITY PROFILES

The streamwise velocities at 1D, 2D, 5D, 10D and 20D at duct mid height are shown in Figure 7.3. The barrage wall/duct locations are shown by thick lines along the x-axis.

![Streamwise Velocity Profiles](image)

*Figure 7.3: Streamwise velocity at duct mid-height*

The bulb wakes and adjacent jets are evident at 1D directly downstream from the duct exits, similar to the stator profiles, with maximum velocity of 0.6 m s\(^{-1}\) which is \(~5.5\) times the inlet velocity. As distance from the barrage increases the jets merge, so that at 2D the jets are still evident but the individual jets are merged by 5D downstream. The velocity decreases and becomes almost uniform across the flume at 20D, at approximately 0.13 m s\(^{-1}\), which is \(~120\%\) of the inlet velocity. The eddies that form either side of the flume extend further and have a stronger backflow velocity than those in the stator experiment. Reversed flow occurs at 1D and 2D, and at 5D the flow is near zero, whereas the stator experiment only had reversed flow at 1D and 2D, and at 5D the
velocity was in the positive streamwise direction at the flume edges. This is probably due to the increased jet velocity. The Coandă effect is evident because at 5D downstream the peak velocity occurs in the flume centre, whereas further downstream at 10D the peak is further to the left hand side, showing that asymmetry still occurs at this distance from the barrage. By 20D this is no longer apparent, suggesting that the flow is uniform across the flume.

There are some anomalies in the results (Figure 7.3), particularly close to the left side of the flume. These tend to occur at around 0.11 m from the side of the tank, possibly due to reflection of the signal from the flume wall. This may be improved by additional seeding material to the flow, but this is not possible with this equipment, due to potential pump damage. The anomalous results have been included but can, however, be disregarded.

The flow throughout the depth can be analysed by assessing the velocity vectors in both the horizontal X-Y and vertical Y-Z planes. The streamwise velocity profiles can also be assessed and compared with those in the stator only experiment.
7.3 DEPTH-VARYING RESULTS

The streamwise and cross-stream velocity vectors were analysed in the $z=0.04$ m, 0.08 m, 0.12 m, 0.16 m and 0.2/0.195 m planes to determine the flow features present throughout the flume depth. The velocity vectors in the 1D, 2D, 5D, 10D and 20D planes were also examined, in order to analyse the cross-stream and vertical velocities.

7.3.1 X-Y VELOCITY VECTORS IN HORIZONTAL PLANE

Figures 7.4-7.7 show the velocity vectors at 0.04 m, 0.08 m, 0.12 m, 0.16 m and 0.2 m/0.195 m; the experimental results show the streamwise and cross-stream velocities at 1D, 2D, 5D, 10D and 20D. (Note that the velocity vector colour scales vary between figures).

The vectors agree with those seen in the results with bulbs and stators only. Close to the barrage near the bed, the cross-stream flow is strong and directed towards the left side of the flume, with strong flow in the opposite direction near the surface. The cross-flow, combined with the Coandă effect, leads to a weak jet forming which varies across the depth; close to the bed the jet is pulled to the left and the jet moves to the right side of the flume at greater depths. This flow variation throughout the depth leads to full flume recirculation.

Close to the duct midheight, $z=0.08$ m, the profile is similar to that shown in Figure 7.2, with wakes forming behind the bulb bodies and jets between these that merge and spread with distance from the barrage. There is still some asymmetry in the flow, formed by the differences between the boundary conditions, the swirl and the Coandă effect, which pulls the central jet to the left of the flume at 5D and 10D. In the stator
only experiment the jet was pulled to the right; at this depth the cross-stream flow is weak, and the effect of the rotors may lead to a slightly different cross-stream flow. There is also stronger reversed flow close to the surface in the rotor experiment, suggesting that the vertical eddies in this flow are stronger than in the stator only experiment. As distance from the barrage increases the jets merge and spread, leading to near uniform flow at 20D downstream. There is still some variation across the flume at this distance, but it is minimal. Whether there is any cross-stream or vertical variation will be examined using the velocity vectors in the vertical Y-Z plane.

There appear to be anomalies in the results, particularly at 20D; this may occur because the ADV requires bubble seeding for accurate recording, but the swirl causes the bubbles to dissipate more rapidly and as distance increases the concentration of the bubbles decreases (described in Chapter 6, Page 183).

Figure 7.4: Velocity vectors at 0.04 m from the bed
Figure 7.5: Velocity vectors at 0.08 m from the bed

Figure 7.6: Velocity vectors at 0.12 m from the bed
Figure 7.7: Velocity vectors at 0.16 m from the bed

Figure 7.8: Velocity vectors at 0.20 m/0.195 m from the bed
7.3.2 Y-Z VELOCITY VECTORS IN VERTICAL PLANE

The cross-stream and lateral velocity vectors in the Y-Z plane at 1D, 2D, 5D, 10D and 20D are shown in Figures 7.9-7.13. The vectors are very similar to those in Chapter 6, where the bulb bodies had stators only. The clockwise swirl (when looking downstream) is apparent at 1D and 2D directly downstream from the ducts and throughout the entire flume at 5D. Close to the barrage the flow varies considerably throughout the depth. As distance from the barrage increases, the variation decreases, until at 20D there is little variation throughout the flume in the vertical and cross-stream direction, so depth-averaged modelling may be applicable at this distance.

![Velocity Vectors in One D Plane](image)

*Figure 7.9: Velocity vectors at 1D downstream*
Figure 7.10: Velocity vectors at 2D downstream

Figure 7.11: Velocity vectors at 5D downstream
Figure 7.12: Velocity vectors at 10D downstream

Figure 7.13: Velocity vectors at 20D downstream
There are three main differences between the stator results and the stator/rotor results: the rotor results show higher streamwise velocities, higher transverse velocities at 1 and 2D, reduced cross-stream variation at 5D and 10D and more anomalous results.

Firstly, the higher streamwise velocities are due to the increased inlet velocity. Secondly, there are reduced cross-stream velocities at 5D and 10D due to the higher swirl, and thus faster dissipation, of the jets. This leads to uniform flow occurring closer to the barrage. Lastly, there are several anomalies in the vertical velocities, especially at 5D, 10D and 20D close to the bed. These may be due to two factors, described previously in Chapter 6: the probe configuration and the bubble dissipation. The results are more anomalous than those shown in the results with stators only, because the rotors cause greater dissipation of the bubbles, so there are fewer bubbles present in the flow as distance from the barrage increases.

To determine the differences between the bulb/stator results and the bulb/stator/rotor results the streamwise velocities were extracted from both sets of experiments and compared using non-dimensionalised velocity profiles.
7.4 COMPARISON WITH STATORs

The streamwise velocity profiles were non-dimensionalised in order to compare them with the stator only results. The bulb/stator and bulb/stator/rotor streamwise velocity profiles were non-dimensionalised using the inlet velocities; these were 0.0781 m s\(^{-1}\) in the bulb/stator experiment and 0.1095 m s\(^{-1}\) in the bulb/stator/rotor experiment.

7.4.1 VELOCITY RATIO PROFILES

The ratio of streamwise velocity to inlet velocity at 1D downstream is shown below in Figure 7.14:

![Figure 7.14: Ratio of streamwise velocity to inlet velocity at 1D downstream](image)

The first thing that can be seen from this plot is the similarity between the two sets of results. Close to the bed (0.04 m) and the duct centreline (0.08 m) the jet locations and magnitudes are in close agreement. Close to the bed the jets are slightly different, with the rotor jets 32\% stronger than the stator only results. The wakes at the duct centres
are almost identical, except at the flume centre wake and in some of the jet magnitudes. The results are, however, highly comparable for most of the profile, with only a 20 % difference in the maximum and minimum velocity ratios. Further from the bed there are some anomalies between the results, with 30 % faster jets forming in the rotor experiment at 0.12 m from the bed, but as distance increases the profiles become more comparable. At all depths the eddy sizes and strengths are similar.

Further from the barrage, at 2D (Figure 7.15), the velocity magnitudes are less comparable, except at the flume centre close to the duct midheight. The difference between the maximum velocity ratios is, however, close to those at 1D; the difference is approximately 20 %. The eddies in the two sets of results are also less comparable at this distance downstream with up to 60 % difference in the eddy strength.

Figure 7.15: Ratio of streamwise velocity to inlet velocity at 2D downstream
As distance downstream increases the results become more comparable. By 5D downstream (Figure 7.16) the variation in the maximum velocities throughout the depth is approximately 15%, without the anomalous results included. The profiles show that the swirl and Coandă effect result in the jet pulling further to the right as distance from the bed increases. The slight difference in the jet location noted in the velocity vectors in the horizontal plane is also seen here, though the variation is minimal.

![Figure 7.16: Ratio of streamwise velocity to inlet velocity at 5D downstream](image)

This variation in the jet location due to the swirl and Coandă effect throughout the depth continues at 10D downstream (Figure 7.17), but at 20D downstream (Figure 7.18) this is no longer apparent. At 20D from the barrage, the streamwise velocity ratio profiles are approximately uniform across the depth, at 1.5, except at 4 cm from the bed where there is a velocity increase on the left side of the flume. This may be an error in the result, because at 10D from the barrage the velocity ratios had dropped to
approximately 1.5, so are unlikely to increase by 300% to 4.5. The results are highly comparable and the inclusion of rotors does not, therefore, greatly affect the non-dimensionalised velocity profiles, but does affect velocity magnitude.

Figure 7.17: Ratio of streamwise velocity to inlet velocity at 10D downstream

Figure 7.18: Ratio of streamwise velocity to inlet velocity at 20D downstream
7.5 COMPUTATIONAL SIMULATIONS – STATOR MODELS

The similarity between the stator and stator/rotor results suggest that any simulations that are applicable to the bulb with stators only may also be applicable for the bulbs with stators and rotors. Two simulations from Chapter 6 were used for comparison with the rotor results: the simulation with the most accurate angular momentum flux and the simulation with the most accurate close-to-barrage velocity profile. The swirl and drag constants were scaled proportional to the square of the inlet velocity, in accordance with Froude scaling. The amount of swirl and the mid-height non-dimensionalised velocity profiles were compared to determine the model accuracy. Another method of analysing the swirl using StarCCM+ is to apply a fan function, described in Chapter 4. This simulates a rotating fan or propeller/rotor in the model, using the blade pitch and the rotation frequency. The, water depths, swirl and non-dimensionalised profiles were again compared, as well as the water depths.

7.5.1 AMF$_{N-D}$ MODEL – REALISABLE K-E

There were several models in Chapter 6 with comparable swirl to the experiment, but the model that also had comparative head difference and lowest residuals (indicating convergence) was the realisable $k$-$\varepsilon$ model with fixed body force. The stator model results were non-dimensionalised for comparison with the rotor experiment and the model swirl constants were scaled in a StarCCM+ run with the rotor experiment dimensions and inlet velocity. The swirl and drag constants in the stator experiment were $C_{Fb,S}=18100$ N m$^{-4}$ and $C_{Fb,D}=1600$ N m$^{-3}$ respectively; these were scaled proportional to the square of the inlet velocity, Froude scaling, to values of $C_{Fb,S}=35580$ N m$^{-4}$ and $C_{Fb,D}=3145$ N m$^{-3}$. The non-dimensionalised angular momentum flux (swirl),
midheight velocity vectors and non-dimensionalised midheight velocity profiles were compared.

### 7.5.1.1 SWIRL

The non-dimensionalised angular momentum flux ($AMF_{N.D}$) from the rotor experiment was 1.00 at 1D downstream from the ducts. The average $AMF_{N.D}$ from the realisable $k$-$\varepsilon$ model simulating stators only was 0.96 at 3 cm from the ducts and 0.61 at 1D downstream. The swirl at 1D in the model has a 39% error when comparing with the experimental swirl, because the StarCCM+ model has rapid jet dissipation.

The applicability of the swirl and drag from Chapter 6 when simulating rotors was assessed by using the scaled body force constants in a StarCCM+ model with the inlet velocity and water heights from the rotor experiment. These gave an average $AMF_{N.D}$ of 0.92 at 3 cm and 0.77 at 1D. The swirl is more comparable in this simulation with a 23% error at 1D downstream. This suggests that the velocity vectors and profiles may be more comparable with the experiment because the swirl is more accurately modelled. There is also better convergence in this model, with residuals of the continuity and $x$-, $y$- and $z$-momentum dropping to $1\times10^{-4}$.

### 7.5.1.2 VELOCITY VECTORS

The velocity vectors at duct mid-height are shown in Figure 7.19. There is some reduced velocity at the duct centres due to the bulb bodies, but the flow profile is much more similar to that in the vane model (Figure 6.30, $k$-$\varepsilon$ model of stators only) than that in the rotor experiment (Figure 7.2) indicating that the velocity profiles may not be accurate. The asymmetry and the eddy sizes are also very different from those seen in
the experiment; these differences may be more obvious when analysing the velocity profiles.

![Figure 7.19: Midheight velocity vectors with fixed body force and realisable k-ε model](image)

### 7.5.1.3 VELOCITY RATIO PROFILES

The profiles of the ratios of streamwise velocity to inlet velocity for the rotor experiments, stator simulations and the rotor simulations are shown in Figure 7.20 to be very different. The rotor model has faster jets across the ducts, slower flow between the ducts and larger eddies. These differences continue throughout the flume until 20D, where the flow becomes more comparable to the experimental rotor result.

These results suggest that if the flow rate, and thus water depth, is increased then the swirl and drag in the model must also be increased to provide an accurate swirl. The more accurate swirl still leads to under prediction bulb wakes and over prediction of jets and eddies; however, it does also lead to better convergence. The CFD predictions of streamwise velocity are similar to the experiments at 20D.
7.5.2 JET PROFILE MODEL – HIGH SWIRL COEFFICIENT, $C_{FB_S}$

The model from Chapter 6 that had the most accurate velocity profile at the duct exits was the high swirl coefficient model. The body force constant values from this model were used to directly compare the non-dimensionalised rotor experiment results with the stator model results and were also scaled for use in a simulation with the rotor experiment dimensions. The swirl and drag constants in the stator experiment were $C_{Fb,S}=71310$ N m$^{-4}$ and $C_{Fb,D}=1600$ N m$^{-3}$ respectively; these scaled to values of $C_{Fb,S}=140180$ N m$^{-4}$ and $C_{Fb,D}=3145$ N m$^{-3}$.

7.5.2.1 SWIRL

The non-dimensionalised angular momentum flux ($AMF_{N,D}$) from the rotor experiment was 1.00. The average $AMF_{N,D}$ from the high swirl constant model using the standard $k-\varepsilon$ model simulating stators only was 1.05 at 3 cm and 0.03 at 1D; there is only a 97% error in the results when comparing the 1D experimental result. This high error further

Figure 7.20: Midheight velocity ratios, Experimental stators/rotors (Props), Realisable $k-\varepsilon$ model with stators (Vanes) and Realisable $k-\varepsilon$ model with stators/rotors (Props)
downstream is because there is fast dissipation of the jets due to the high swirl in the rotor experiments.

The average $AMF_{N-D}$ values from the StarCCM+ model with rotors included were 1.93 at 3 cm and 1.23 at 1D. The error at 1D for the model is 23%, so the swirl at this distance is better predicted in this model than in the vane model. This suggests that the velocity vectors and velocity profiles were more accurate for the rotor model than the stators only model.

### 7.5.2.2 VELOCITY VECTORS

The velocity vectors (Figure 7.21) show that the jets extend further downstream than previously (Figure 6.36). The wake at the duct centres is less prominent on the left hand side, but stronger on the right, showing strong asymmetry in the flow through the ducts. This leads to asymmetry further down the flume, which is enhanced by the Coandă effect. The distance that the flow is asymmetric and the comparability of the duct profiles was assessed using the velocity profiles.

![Figure 7.21: Midheight velocity vectors with high swirl constant, $C_{Fb,S}$](image)
7.5.2.3 VELOCITY RATIO PROFILES

The velocity ratio profiles (Figure 7.22) are obtained from the velocity vectors (Figure 7.21): there is strong asymmetry across the flume at 1D, with reduced wake effects at the duct centres on the left and strong wakes on the right. This has led to a velocity profile downstream from the central duct that is comparable to the experiment, but differing results at either flume side. The asymmetry and varying comparability of the results across the flume is still evident at 2D, but by 10D the velocity ratio magnitudes are closer to the experimental results than the stator only model. The profiles are very similar at 20D downstream, which is expected since 2-D modelling is reasonable at this distance.

![Velocity Ratios at Duct Midheight](image)

**Figure 7.22:** Midheight velocity ratios, Experimental stators/rotors (Props), Realisable k-e model with stators (Vanes) and Realisable k-e model with stators/rotors (Props)
7.6 COMPUTATIONAL SIMULATIONS – FAN FUNCTION

An alternative method of adding swirl to the model is to add a fan momentum source (as described in Chapter 4). The fan source uses actuator disc methodology to model the forces imposed by the blades and the detailed geometry of the blades is, therefore, not required. If a fan performance curve is available for the fan that is modelled then this can also be used, but is not a requirement; the rotors used in the experiments do not have a performance curve so the basic parameters of the model were used. The momentum source requires the input of the rotation rate and blade angle. The average rotor rotation rate was 138 rpm, or 14.5 rad s\(^{-1}\). The rotor blades have an average slope of 23°, so the blade angle in the model is 0.401 rad. The model also requires an upstream and downstream boundary; these were applied at the upstream and downstream end of the straight side of the bulb, giving a length of 0.0682 m that the fan source was applied over. Only the effect of the rotors was analysed, because the effect of the stators on the flow could not be accurately modelled using body forces, as shown in Section 7.5. The close agreement between the velocity profiles due to stators only and stators with rotors means that the differences in the computational model were assumed to be negligible. The flow conditions, velocity vectors and velocity profiles were compared with the both the stator only and the stator/rotor experiments.

7.6.1 FLOW CONDITIONS

7.6.1.1 HEAD DIFFERENCE

The average pressures on the upstream and downstream surfaces were 115 Pa and -45 Pa respectively; therefore the pressure difference over the barrage was 160 Pa. This gave a head difference of only 0.0163 m, which has an error of 70% from the experimental value of 0.054 m.
7.6.1.2 SWIRL

The circulation and angular momentum flux were calculated at 3 cm and 1D downstream from the barrage. The average circulation across the ducts close to the barrage was 75.3 and all of the duct results were within 8% of this, showing uniformity across the ducts. The average circulation across the ducts at 1D was 34.6 and all of the duct results were within 55% of this, showing larger variation in the swirl at this distance downstream. The circulation in the model is much lower than that exhibited in the experiments, $\Gamma_{N:D}=120.2$, but the $AMF_{N:D}$ may be more accurate because it incorporates the variation in the streamwise velocity. The average $AMF_{N:D}$ across the ducts at 3 cm from the barrage was 1.27. The average $AMF_{N:D}$ across the ducts at 1D from the barrage was 0.96, which is close to that in the experiments, 1.00, with only a 4% error. The $AMF_{N:D}$ and thus swirl, produced by the rotors are accurately modelled using this method. However, the swirl could be better modelled using the fan momentum source if a fan performance curve was available for the rotor.

7.6.2 MIDHEIGHT VECTORS

The velocity vectors produced by the experiment and the model at duct midheight are shown in Figure 7.23. The vectors show that asymmetric jets form at the duct edges, due to the asymmetry in the flow further downstream. The wake at the duct centre exhibited in the experiments is not apparent, though there are reduced velocities in the jet centres. Eddies form close to the barrage on both sides of the flume, slightly larger than those in the experiment. The computational Coandă effect is stronger than in the experiment, leading to a large area of low velocity forming on the right side of the flume and flow uniformity across the flume width is not achieved by 20D. The inaccuracy of these results was assessed using the velocity ratio profiles.
Figure 7.23: Midheight velocity vectors with fan momentum source, a) Experiment b) StarCCM+
7.6.3 VELOCITY RATIO PROFILES

The key difference between the StarCCM+ and experimental results is the location of the peaks in velocity close to the barrage (as shown in Figure 7.24): the computational model shows jets forming downstream from the ducts, with only a slight drop in velocity at the duct centre due to the wake forming behind the bulb bodies. This continues to 2D downstream. The flow profile is still very different at 5D; the maximum ratio in the fan model occurs on the left side of the flume, not the centre and has an error of 34 % from the rotor experiment. The eddies’ size and extent downstream are over predicted on the right side of the flume, but similar on the left. At 10D and 20D the region of low velocity on the right of the flume persists. The maximum velocity ratios at these distances are comparable to the rotor experiment, with a 17 % error at 10D and 25 % error 20D.

Figure 7.24: Midheight velocity ratios, Experimental stators only, Experimental stators/rotors and Standard k-ε model with fan momentum source
7.7 CONCLUSIONS

Three main issues were addressed in this chapter:

1. How does velocity vary with depth and distance downstream?
2. How well this is predicted by generally available 3-D CFD?
3. At what distance downstream is 2-D depth-averaged modelling justified?

1. The flow field and non-dimensionalised velocity profiles from the rotor experiment show very good agreement with the non-dimensionalised velocities from the stators experiment. The stator and stator/rotor water depth drops over the downstream flume length were equivalent. The jet profiles and depth variation were also extremely similar. The addition of rotors and increased inlet velocity did, however, increase the non-dimensionalised $AMF_{N,D}$ by 30% at 1D downstream, though the streamwise velocity profiles are not significantly altered by this. The cross-stream and vertical velocities are slightly elevated, but the velocity patterns are similar. When non-dimensionalised, using the inlet velocity, the addition of rotors does not appear to significantly change the flow field.

2. Three StarCCM+ models were compared with the non-dimensionalised stator and stator/rotor experimental results, with respect to the circulation, angular momentum flux, velocity profiles and model convergence; a summary of the results is shown in Table 7.2.
<table>
<thead>
<tr>
<th>Flow Features</th>
<th>Rotor Expt</th>
<th>Fixed body force, Realisable k-ε</th>
<th>Fixed body force, High $C_{Fb,S}$</th>
<th>Fan Function Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circulation, % error</td>
<td>120.2</td>
<td>30.14, -75%</td>
<td>50.98, -58</td>
<td>34.6, -71</td>
</tr>
<tr>
<td>Angular Momentum Flux, % error</td>
<td>1.00</td>
<td>0.77, -24%</td>
<td>1.23, 22</td>
<td>0.96, -5</td>
</tr>
<tr>
<td>Profile Correlation</td>
<td>1D</td>
<td>Poor jet locations</td>
<td>Good central jet, poor at sides</td>
<td>Poor jet locations</td>
</tr>
<tr>
<td></td>
<td>2D</td>
<td>Poor jet locations</td>
<td>Poor jet locations</td>
<td>Poor jet locations</td>
</tr>
<tr>
<td></td>
<td>5D</td>
<td>Over predicted eddy length and wake on right</td>
<td>Over predicted flow on left</td>
<td>Over predicted flow on left and wake on right</td>
</tr>
<tr>
<td></td>
<td>10D</td>
<td>Over predicted eddy length and wake on right</td>
<td>Good</td>
<td>Over predicted wake on right</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>Good</td>
<td>Good</td>
<td>Over predicted wake on right</td>
</tr>
<tr>
<td>Convergence</td>
<td>-</td>
<td>$1 \times 10^{-4}$</td>
<td>$1 \times 10^{-4}$</td>
<td>$1 \times 10^{-3}$</td>
</tr>
</tbody>
</table>

*Table 7.2: Summary of computational results*

The realisable $k-\varepsilon$ results show that the swirl and bulb body wakes at the duct exits are under-predicted, because jets, eddies, jet merging and dissipation are over-predicted. This also leads to poorly predicted velocity profiles. The jets are also over-predicted in the high swirl model, but the increased swirl leads to better predictions of $AMF_{N,D}$ at 1D downstream. The velocity profiles are more comparable, with better predicted duct profiles and good comparison with the experiments by 10D downstream. The profiles became comparable at 20D downstream.
The fan momentum model produced better swirl results. The circulation was under-predicted because it does not incorporate the entire duct or the streamwise velocity, but the $AMF_{N:D}$ was extremely similar. The bulb body and stator wakes were again under-predicted and the jets, eddies and Coandă effect were over-predicted leading to a less comparable profile at 20D downstream. The profile results may be improved if a performance curve was included for the propellers/rotors. The fan did, however, accurately predict the swirl produced.

3. Depth-averaged modelling is not applicable up to 20D, because the swirl in the ducts cannot be modelled, but depth-averaged modelling could be used after 20D, where there is no swirl. This has not been fully investigated, but there is little variation in the velocity profiles with depth at 20D and the asymmetry is negligible.
8. ANALYSIS OF BED SHEAR STRESS AND SEDIMENT TRANSPORT

Installing a tidal barrage in a river or estuary can affect the bed shear stress in the surrounding area. Bed shear is important because it controls sediment erosion, suspension and trapping, and affects water quality. Whether a barrage causes sediment transport and scour can be assessed by analysing where the bed shear stress exceeds the critical value, which is the limit for inducing sediment motion.

The bed shear stress was measured/predicted by:

- for the experiments: inferred (assuming a log law) from the close-to-bed (1 cm) velocity measurements;
- for the 3-D StarCCM+: directly from the shear stress reported by the code (which is based on near-wall cell values, using wall functions);
- for the 2-D SW2D: by relating depth-averaged velocity to shear stress using a friction factor.

StarCCM+ was also used to establish the validity of using velocities at 1 cm from the bed to infer shear stress.

The velocities, and thus the calculated or extracted bed shear stresses, were used to determine the bed shear stress patterns on the bed, the accuracy of bed shear stress in each model method and the regions where the critical bed stress would be exceeded and sediment transport would occur in a full scale barrage system.
Initially, the results for the experiment, StarCCM+ model and SW2D model with no turbines (Chapter 5) were compared. Then, the results for the experiment, the StarCCM+ model with the most accurately modelled swirl and the StarCCM+ model with the most accurately modelled velocity profiles with stators only (Chapter 6) were compared. Lastly, the experiments with stators/rotors (Chapter 7) were compared with the no turbine and bulb/stator experimental results.

8.1 METHODOLOGY

8.1.1 BED SHEAR STRESS

The bed shear stress, \( \tau_b \), is the drag per unit area of the flow; it is calculated using the friction, or shear, velocity \( u_\tau \) and the flow density, \( \rho \):

\[
\tau_b = \rho u_\tau^2 \quad \text{or} \quad u_\tau = \sqrt{\frac{\tau_b}{\rho}}. \tag{45}
\]

A rough boundary, such as the tank floor, with a fully-developed turbulent boundary layer, has a logarithmic mean-velocity profile:

\[
U(z) = \frac{u_\tau}{\kappa} \ln \left( 33 \frac{z}{k_s} \right), \tag{46}
\]

where \( \kappa \) is the von Karman’s constant (generally \( \sim 0.41 \)), \( k_s \) is the roughness height and \( z \) is the distance from the boundary.
The velocity at a given point from the bed can, therefore, be used to determine the friction velocity at the bed provided the value is within the log layer. The log law is generally considered to cover 20% of the flow depth, so in a flow of ~0.2 m the log law extends to 4 cm from the bed (provided that the presence of jets does not interfere with this profile). The experimental velocities were recorded at 1 cm from the bed and used to determine the friction velocity, with two key assumptions: firstly, that a fully-developed turbulent boundary layer is assumed to form and secondly, that the log layer extends to 1 cm from the bed. These assumptions may not be accurate due to the interaction of the jets with the bed and the relatively short flume and shallow depth. The accuracy of the results when using this method can be determined by comparing the results to those produced by the validated CFD model. The values that were used in Equation 45 and Equation 46 for the experimental bed stress were, therefore, \( U(z)=U(1 \text{ cm}), \kappa =0.41, k_s=0.001 \text{ m}, z=0.01 \text{ m} \) and \( \rho=1000 \text{ kg m}^{-3} \).

The bed shear stress can be extracted directly from the CFD simulation results and plotted at various locations. The bed stresses at the downstream distances matching the experiments were extracted for comparison and analysis of the experimental results.

The SW2D model produces depth-averaged velocities that can be used to calculate the bed stress using Equation 47 below:

\[
\tau_b = \frac{1}{2} \rho c_f U_{avg}^2.
\] (47)
The SW2D model assumes a constant coefficient of friction, $c_f$, calculated in Chapter 4 as 0.0138 and the velocities produced by the model are depth-averaged. The bed stress contours were plotted for each modelling method and the bed stress values at the experimental locations at 1D, 2D, 5D, 10D and 20D were directly compared.

### 8.1.2 COEFFICIENT OF FRICTION ASSESSMENT

The depth-averaged SW2D model assumes a constant coefficient of friction, $c_f$, along the bed and the velocities are depth-averaged; the accuracy of these assumptions can be assessed by comparing the bed stress previously calculated with the reference bed stress calculated using this friction coefficient and the depth-averaged velocity at 20D downstream. The reference $c_f$ was 0.0138 and the channel-averaged velocity at 20D downstream was 0.116 m s$^{-1}$. Using Equation 47, the reference bed stress, $\tau_0$, was calculated as 0.0928 N m$^{-2}$. The experimental and StarCCM+ computational ratios of the bed stresses to the reference stress were assessed to determine the accuracy of assuming a constant coefficient of friction.

### 8.1.3 THRESHOLD OF MOTION FOR SEDIMENT TRANSPORT

The threshold of motion, used to determine the locations of sediment deposition and scour, occurs where the bed stress exceeds the critical bed stress; this can be assessed using the Shields parameter (defined in Soulsby 1997) and determining where this exceeds its critical value ($\tau^* > \tau^*_{crit}$). An example of the effect of a full-scale barrage on bed movement can be assessed by analysing where sediment movement occurs when a bed of a certain grain size is added to the experiment or computational model. A reference quartz-like material, such as sand, with a grain diameter of 1 mm was added.
to the full-scale model and the Shields parameter and thus threshold-of-motion were
determined.

The Shields parameter is calculated using the bed shear stress, $\tau_b$, sediment density, $\rho_s$, water density, $\rho$, gravity, $g$, and sediment diameter/grain size, $d$:

$$\tau^* = \frac{|\tau_b|}{(\rho_s - \rho)gd}.$$  \hspace{1cm} (48)

The dimensionless critical stress, from Soulsby 1997, is calculated using the dimensionless diameter, $d^*$, which uses the specific gravity, $s$, and kinematic viscosity, $\nu$:

$$s = \frac{\rho_s}{\rho},$$ \hspace{1cm} (49)

$$\nu = \frac{\mu}{\rho},$$ \hspace{1cm} (50)

where $\mu$ is the dynamic viscosity. The dimensionless diameter and dimensionless critical Shields parameter are calculated using the following formulae:

$$d^* = d \left(\frac{(s-1)g}{\nu^2}\right)^{\frac{1}{3}}$$  \hspace{1cm} (51)
The threshold-of-motion occurs when \( \tau^* > \tau^*_{\text{crit}} \), i.e. when the Shields parameter exceeds the critical value. The ratio of Shields parameter to critical Shields parameter was plotted as contours, so that the area where sediment motion occurs was determined.

The values for the chosen parameters are listed below:

- \( \rho_s = 2650 \text{ kg m}^{-3} \) (sand)
- \( \rho = 1000 \text{ kg m}^{-3} \) (water)
- \( s = 2.65 \)
- \( d = 1 \times 10^{-3} \text{ m} \) (sand)
- \( \mu = 8.89 \times 10^{-4} \text{ kg m}^{-1} \text{s}^{-1} \)
- \( v = 8.89 \times 10^{-7} \text{ m}^{2} \text{s}^{-1} \)
- \( d^* = 2.74 \)
- \( \tau_f = \tau_b \times \text{Scale Factor} = \tau_b/0.007 \text{ Pa} \)
- \( \tau^*_{\text{crit}} = 0.032 \)
8.2 NO TURBINE REPRESENTATION

Several aspects of the bed shear stress and sediment transport predicted by the experiments, StarCCM+ and SW2D were assessed: the bed stresses, the ratio of bed stress to reference bed stress and full-scale threshold-of-motion.

8.2.1 BED SHEAR STRESS

The bed stress vectors and resultant bed stress magnitude contours from the experiments (a), StarCCM+ (b) and SW2D (c) are shown overleaf in Figure 8.1 and Figure 8.2. The experimental results show that the experimental bed stresses reach 0.25 Pa in areas approximately 5D downstream from the duct exits. At the outermost ducts the peak stress occurs close to the barrage, whereas at the flume centreline the higher stress occurs further downstream. This is because the jet exits are not touching the bed, so attach to the bed at some distance downstream. This leads to low velocities and bed stresses at 1D, shown in the bed stress vectors. There are also areas of high bed stress at the flume edges where there is high reversed flow due to the eddy formation (shown in Figure 5.10), with low bed stress inboard of this where the velocities at the eddies’ centres are low. As distance from the barrage increases the bed stress reduces, until at 20D downstream the bed stress is between 0 and 0.05 Pa. There is slightly elevated stress in the centre (10D) and towards the left side of the flume (20D) as the jets merge, forming a central jet that bends to the tank side due to the Coandă effect (also shown in Figure 5.10).

These flow features are similar to those exhibited by StarCCM+, although the computational bed stress magnitudes are higher than in the experimental results. The peak bed stress in StarCCM+ exceeds 0.25 Pa both at the jet locations and at the
reversed flow location on the left side of the flume; the maximum bed stress is approximately 0.95 Pa close to the barrage. This high stress is caused by large cross-stream velocities at less than 1D, which may also occur in the experiments but the profiles this close to the barrage were not measured. The spreading of the jet towards the bed is evident just beyond 1D, creating high bed stresses at 2D and 5D and reversed bed stress at less than 1D. This could lead to sediment deposition at the barrage and a scour hole. The bed stresses further from the barrage, towards the flume end, are also much higher, with the majority of the bed experiencing stresses between 0 and 0.1 Pa at 20D downstream.

The SW2D results are not, however, as comparable as the StarCCM+ and experimental results. The results show elevated bed stresses downstream from the ducts, but much closer to the barrage, because the results are 2-D which is equivalent to ducts located on the bed. Vertical jet spreading does not, therefore, occur. The bed stress contours show the same discrepancies in the results that occurred in the velocity vectors (Figure 5.21): smaller, weaker eddies and no asymmetry lead to lower bed stresses at the flume edges. Though the velocity vectors appear to show the jets merging, the bed stresses caused by alternate jets are still apparent at 20D downstream. The maximum and minimum bed stresses are comparable to those in StarCCM+, but the stress patterns are very different.
Figure 8.1: Bed shear stress vectors, a) Experiments b) StarCCM+ c) SW2D

(NB Colour scales vary between plots)
Figure 8.2: Bed shear stress, a) Experiments b) StarCCM+ c) SW2D

(NB Colour scales vary between plots)
The bed stress profiles at 1D, 2D, 5D, 10D and 20D (Figure 8.3) show these same features:

![Figure 8.3: Experimental, StarCCM+ and SW2D bed shear stresses at 1D, 2D, 5D, 10D and 20D](image)

The experimental and SW2D results show similar magnitudes of bed stress at the flume edges at 1D, with approximately 24% difference, but as the distance from the barrage increases the SW2D bed stresses decrease too quickly (as seen in Figure 8.2). After 1D the locations of the areas of elevated bed stress are also dissimilar, except at 20D where the profiles are much more comparable, with only a 16% difference in the maximum bed stress. The StarCCM+ results show much higher bed stresses than the experiments, with 64%, 65%, 32% and 49% differences in the maximum at 1D, 2D, 5D and 10D respectively. The difference drops to 6% at 20D. The bed stresses created by the eddies
are in closer agreement, though, with only an average 21% predicted difference of the left eddy and highly accurate eddy sizes (shown by the similar locations of zero bed stress).

The difference between the experimental and StarCCM+ results is a consequence of the different calculation methods: StarCCM+ uses the stress tensor at the wall to calculate bed stress, whereas the experimental stress is calculated using the velocities at 1 cm from the wall. The near-wall velocities are assumed to have a log profile defined by the velocities at 1 cm, but this may not by accurate for two reasons: firstly, the velocity recording may not be within the log layer, so the velocity close to the wall would be under predicted; and secondly, the interference of the jets with the velocity profile is not accounted for in the log layer profile. Whether a log law is a reasonable approximation for the close to wall flow can be determined by assessing the $z^+$ values in both the experiment and StarCCM+.

### 8.2.2 LOG-LAW LAYER ASSESSMENT

Figure 8.4 below shows the experimental $z^+$ contours at the bed calculated using Equation 53 below, where $z$ is the vertical distance from the wall, 1 cm, $u_t$ is the friction velocity and $v$ is the kinematic viscosity.

$$z^+ = \frac{zu_t}{v}$$  \hspace{1cm} (53)

Pope (2000) states that if $z/\delta<0.3$ then the log-law is applicable; for this experiment $z/\delta=0.09$, so the log-law is a reasonable assumption. Pope also states that when $z^+>30$
the log-law assumption is true, but if $5 < z^+ < 30$ the flow is in the buffer layer, the region between the viscous sub-layer and the log-law region, where viscous effects are significant compared to the shear stress. The results (Figure 8.4) show that the experimental $z^+$ values range between $\sim 30$ and 180, which shows that the log-law is a good assumption for the flow at this distance from the boundary. $z^+$ is directly proportional to the friction velocity, so high $z^+$ values occur where there are regions of high friction velocity and bed stress.

![Experimental $z^+$ values](image.png)

Figure 8.4: Experimental $z^+$ values

The streamwise and cross-stream velocities at 1cm from the bed were extracted from the StarCCM+ model and the friction velocities, and thus the $z+$ values, were calculated using the same method as for the experiments. The $z+$ values for these two methods are shown in Figure 8.5. Also shown are the $z+$ values directly extracted from the StarCCM+ model at the bed cell.
The $z^+$ values calculated using the velocities at 1 cm from the bed are similar between the experiments and the StarCCM+ model. The peaks in $z^+$, caused by high levels of friction velocity, occur in similar locations and the maximum $z^+$ values are within 21% of each other at 1D. This difference reduces to 2% at 20D. This also shows that friction velocities and bed stresses calculated from the velocities at 1cm should be reasonably accurate because the flow is within the log-law layer. The $z^+$ values directly computed by StarCCM+ in the near-wall cell are much lower, but still above the viscous sub-layer, so can be used to compute the bed stresses.

![Experimental and StarCCM+ $z^+$ Profiles](image)

*Figure 8.5: Experimental and StarCCM+ $z^+$ profiles at 1D, 2D, 5D, 10D and 20D (with dashed lines to depict $z^+$=30 and $z^+$=200)*

The bed stresses from these two methods were compared to determine the accuracy of assuming that the bed stress can be calculated from the velocity at 1 cm from the bed. The bed stresses for these two methods are shown in Figure 8.6. Also shown are the bed stresses.
stress values directly extracted from the StarCCM+ model at the bed cell. The results show that the bed stresses calculated from the StarCCM+ velocities at 1 cm from the wall are similar to those directly output from the model, particularly at 10D and 20D, indicating that this is a reasonable method for calculating the experimental shear stress. Close to the barrage there is a slight difference between the results - at 1D the maximum varies by 45% and at 2D the maximum varies by 24% – but the general profiles and stresses at the centre and edges are similar. The StarCCM+ model still predicts slightly higher bed stresses than the experiment, but the values extrapolated from 1 cm are much more comparable with the experiment, with only 28% and 25% differences in the maximum stress at 1D and 2D respectively. The difference in the maximum bed stress between the StarCCM+ results drops to 5% at 20D and these are within 5% of the experimental values. The method of calculating the bed stress from the velocity at 1 cm from the bed is therefore reasonably accurate, particularly after 10D.

Figure 8.6: Experimental and StarCCM+ bed stress profiles at 1D, 2D, 5D, 10D and 20D
8.2.3 BED SHEAR STRESS RATIO

The SW2D model assumes a constant coefficient of friction across the bed and 2-D depth-averaged velocities. The accuracy of these assumptions was assessed using the ratio of the magnitude of local bed shear stress, $\tau_b$, to a reference stress, $\tau_0$. The reference bed stress was 0.0926 N m$^{-2}$, calculated using the skin friction coefficient computed from Equation 35, 0.0138, and the average velocity at 20D, 0.116 m s$^{-1}$. Contour plots of these ratios are shown in Figure 8.7 and the ratio profiles at 1D, 2D, 5D, 10D and 20D are shown in Figure 8.8. The experimental and StarCCM+ contours show that downstream of the barrage the local bed stress can be up to 3 or 5 times the reference stress where the jets become attached to the bed.

When the ratio is close to one, the coefficient of friction can be assumed constant and the depth-averaged velocities simply determine the bed shear stress. Figure 8.8 shows that this can only be assumed at 20D downstream. Closer to the barrage the computational bed stress can reach up to 6 times the reference stress, where strong cross-stream flow creates strong cross-stream bed stresses, shown in Figure 8.1.

This shows that even if $U_{avg}$ is known the friction coefficient is highly influenced by 3-D effects, as jet attachment to the bed and strong turbulent mixing causes magnification of friction. 3-D effects remain significant at 10D and a reference stress calculated from $c_f$ and $U_{avg}$ should only be used at 20D.
Figure 8.7: Ratio of bed stress magnitude from to reference bed stress,

a) Experiment b) StarCCM+
Figure 8.8: Experimental and StarCCM+ ratios of bed stress to reference bed stress

(with duct locations shown by thick lines)
8.2.4 FULL-SCALE THRESHOLD OF MOTION

Figure 8.9 shows the contours of the full-scale ratios of Shields parameter to critical Shields parameter, with sediment size of 1 mm used for experimental stress coefficients, StarCCM+ model coefficients and SW2D model coefficients. If this ratio exceeds a value of 1 then sediment transport occurs, because the Shields parameter exceeds the critical Shields parameter. The contours show that the critical Shields parameter is exceeded at all locations throughout the flume region.

The full-scale barrage would, therefore, cause sand of this size to move at all regions within 20D of the barrage, though low ratios occur where the friction velocity is small, at the eddy centres and flume sides, so movement will be minimal. High ratios occur where the jets attach to the bed (experiment and StarCCM+) and close to the barrage (SW2D), potentially indicating the presence of scour. Again the StarCCM+ model predicts higher bed stresses, and thus bed stress ratios, than those in the experiments, but where the jets attach there is only a ~25% difference. The location of scour and extent of high stress ratios, and thus high bed-load transport, is under predicted by the SW2D model.

Local scour and deposition will, therefore, be considerably underestimated using depth-averaged assumptions in the near field. Sediment scour and deposition may affect barrage operation and alter the flow fields downstream, but the bed may reach a state of equilibrium, where scour and deposition no longer occur.
Figure 8.9: Full-scale ratios of sand Shields parameter to critical Shields parameter,

- a) Experiments
- b) StarCCM+
- c) SW2D

280
8.3 BULB HOUSING WITH STATORS

When stators were added to the experiment the bed velocities had an increased cross-stream element due to the swirl at the duct exits. This created higher close-to-bed velocities closer to the barrage, which reduced more quickly with distance downstream than the previous results with no turbine representation. This indicates that bed stress levels may be higher closer to the barrage, but decrease more quickly, creating higher levels of scour close to the barrage, but a smaller region of bed-load transport.

The close-to-bed velocities from the experiment were used to calculate the friction velocities and bed stresses and the bed stresses were directly extracted from the StarCCM+ models. Two StarCCM+ models were analysed: the model that best predicted the stator swirl, the realisable $k$-$\varepsilon$ turbulence model, and the model that best predicted the 1D profile, the high swirl constant model. The SW2D model was not analysed because it cannot model the rotational element of the flow.

8.3.1 BED SHEAR STRESS

The bed stress vectors and contours of the bed stress magnitude from the experiments, realisable $k$-$\varepsilon$ turbulence StarCCM+ model and high swirl constant StarCCM+ model are shown overleaf in Figure 8.10 and Figure 8.11. The experiments show higher bed stresses downstream from the ducts, as expected, with no areas of high bed stress at the edges, because the eddies are smaller and weaker than the previous experiment (as seen in the velocity vectors in Chapter 6, Figure 6.6). Jet spreading appears to occur closer to the barrage due to the additional swirl, resulting in higher bed stresses closer to the barrage. There is also a stronger cross-stream stress component, particularly at 1D downstream.
Figure 8.10: Bed shear stress vectors,

a) Experiment b) Realisable k-ε turbulence model c) High swirl constant model
Figure 8.11: Bed shear stresses with stators,

a) Experiment b) Realisable k-ε turbulence model c) High swirl constant model
The realisable $k$-$\varepsilon$ turbulence model results show comparable bed stress magnitudes, but the larger computational eddies create higher reversed bed stresses. The bed stress vectors and magnitudes may be similar because the velocity magnitudes in the experiment are slightly higher (Chapter 6, Section 6.5.4) and the results in Section 8.2 suggested that the experiments under predicted the friction velocity. The patterns of the bed stress are also highly comparable, with peaks in stress downstream from the ducts (closer than before, as predicted) that form into a central peak further downstream, but the StarCCM+ model shows higher bed stresses further downstream. Very low bed stress occurs on the right side of the realisable $k$-$\varepsilon$ turbulence model due to the low velocities, shown in Chapter 6, Figure 6.32.

The high swirl constant model produces a very different contour plot, with much higher bed stresses at the barrage, due to the cross-stream velocities that form (Chapter 6, Figure 6.37). This stronger swirl also leads to a pronounced asymmetry, which forces the flow to the right side of the flume, causing elevated bed stresses along the right side of the tank. The bed stresses in the realisable $k$-$\varepsilon$ turbulence model are more comparable to the experiment than the high swirl constant model, even though the velocity profiles (shown in Chapter 6) are dissimilar. The difference in bed stress magnitude and asymmetry is shown in the bed stress profiles (Figure 8.12).

The high bed stresses close to the barrage and along the right side of the flume in the high swirl model are shown, with maximum bed stresses varying from the experiment by ~280% at 1D. The maximum bed stresses predicted by the realisable $k$-$\varepsilon$ turbulence model are closer to the experimental results, with differences as low as 17%, at 2D. The
higher bed stresses downstream from the ducts exits, with the central (slightly asymmetric) peak further downstream, are also evident in the experimental and realisable $k$-$\varepsilon$ turbulence model profiles. At 20D downstream, the bed stresses are very low, between 0 and 0.1 N m$^{-2}$ (or Pa) and the models show good agreement, except for the high stresses on the right side of the flume in the high swirl constant model.

![Bed Shear Stress](image)

**Figure 8.12:** Experimental, StarCCM+ realisable $k$-$\varepsilon$ turbulence model and StarCCM+ high swirl constant model bed shear stresses with stators at 1D, 2D, 5D, 10D and 20D

8.3.2 LOG-LAW LAYER ASSESSMENT

The wall $z^+$ values were again analysed and Figure 8.13 shows the experimental $z^+$ values at the bed. Throughout the majority of the flume the $z^+$ values are above 30, indicating that the flow at 1cm from the bed is within the log-law, with high values of 200-220 at the location of the jet attachment.

285
Figure 8.14 shows the $z^+$ values calculated from the velocities at 1 cm from the bed in the experiments, realisable $k$-$\varepsilon$ turbulence model and the high swirl constant model. The experimental and realisable $k$-$\varepsilon$ turbulence model results are similar in both general profile and $z^+$ magnitude, with the majority of the flume showing $z^+$ values between 30 and 220. In areas where the $z^+$ is below 30 the viscous effects have more of an effect on the flow and in areas where the $z^+$ is high (>220) the viscous effects become more negligible. The near-wall bed stresses and bed stress calculated from the velocity at 1 cm from each of the StarCCM+ models are shown in Figure 8.15. These show that there is less difference between the bed stresses in the realisable $k$-$\varepsilon$ turbulence model, which better predicts the bed stress field (Figure 8.11) than the high swirl coefficient results, due to the high amount of swirl and therefore depth variation. These results show that the theory of predicting bed stress from the velocity at 1 cm from the bed is accurate when there is a lower amount of swirl.
Figure 8.14: Experimental and StarCCM+ $z^+$ profiles

Figure 8.15: StarCCM+ bed stress profiles
8.3.3 BED SHEAR STRESS RATIO

The 3-D effects and assumption of a constant coefficient of friction across the bed were analysed by comparing the magnitude of bed stress to a reference stress. The reference bed stress was 0.0444 N m$^{-2}$, calculated using the skin friction coefficient computed from Equation 35, 0.0138, and the average velocity at 20D, 0.0802 m s$^{-1}$. Contour plots of these ratios are shown in Figure 8.16 and the ratio profiles at 1D, 2D, 5D, 10D and 20D are shown in Figure 8.17.

The experimental and realisable $k$-$\varepsilon$ turbulence StarCCM+ model contours show that downstream of the barrage the local bed stress can be up to 9 times the reference stress where the jets become attached to the bed, though the StarCCM+ results are 19% higher than the experiments. The high swirl constant predicts magnitudes 250% higher than the experiments, but the swirl here is over predicted. As distance from the barrage increases the ratio reduces to 1, though even at 20D downstream the ratio is slightly above 1 in some locations.
Figure 8.16: Ratio of bed stress magnitude to reference bed stress,
a) Experiment b) Realisable $k$-$\epsilon$ turbulence model c) High swirl constant model
Similar to the result with no turbine representation, bed-load transport occurs in the entire flume (Figure 8.18 overleaf) when an example bed material is put in with the density of sand and grain size of 1 mm. Scour is predicted close to the barrage and lower levels of bed movement occur further down the flume. The previous differences between the models are apparent in these cases as well, including the difference in the maximum bed stress and the bed stress patterns. The lower ratios of bed stress to critical stress close to the barrage could result in sediment deposition at the barrage, which could affect the turbine performance and downstream flow field, but equilibrium will be reached where no more bed transport will occur.
Figure 8.18: Full-scale ratios of silt Shields parameter to critical Shields parameter,
a) Experiment b) Realisable $k$-$\varepsilon$ turbulence model c) High swirl constant model
8.4 BULB HOUSING WITH STATORS AND ROTORS

The addition of rotors to the bulbs with stators and a higher inlet velocity lead to increased bed velocities in the cross-stream direction, due to higher swirl and higher streamwise velocities. The effects on the bed stresses are assessed below and compared with the previous experimental results with stators only to determine the effect of adding a rotating component to the turbine. The bed stresses produced by a StarCCM+ model were not assessed because the models were not accurate in predicting the velocity vectors, so would not be accurate when predicting the bed stresses.

8.4.1 BED SHEAR STRESS

In order to compare the stator/rotor results with the stator only results, the velocities were normalised using the inlet velocities; the normalised bed stresses were then calculated using these normalised velocities. The normalised bed stress vectors and contours of bed stress magnitude from the experiments with stators and the experiments with stators and rotors are shown in Figure 8.19 and 8.20.

The increased bed velocities close to the barrage lead to higher bed stresses, particularly downstream from the right-hand ducts; this is because there are greater velocities on this side of the flume, shown in Chapter 7, Figures 7.4 and 7.9. The maximum bed stresses in the stator/rotor experiment occur in similar locations to those in the stators only experiment, though the bed stress magnitudes are different (note the different scales on the velocity vectors). The profiles appear to be similar, particularly downstream at 5D, 10D and 20D, though this was assessed using the normalised bed stress profiles.
The key differences between the normalised bed stresses occur close to the barrage and at the flume sides. The vectors show the comparative normalised bed stress magnitudes throughout the flume for each experiment, but the contours in Figure 8.20 show the difference in the normalised bed stress magnitude between each experiment. The contours have the anomalous results removed for clarity.
These show that the bed stresses close to the barrage in the stator/rotor experiment are higher than those in the stator only experiment, but become more comparable as distance from the barrage increases. Within 10D from the barrage the normalised bed stresses show similar patterns, but the magnitudes are very different. Over 10D from the barrage the bed stress magnitudes and patterns are in better agreement, but the location of the central region of higher bed stresses is slightly further to the left of the flume, due
to the additional swirl and Coandă effect created by the rotors. There are some anomalous results in the bed stress, particularly at 10D and 20D, which are a result of the errors in the velocity measurement described in Chapter 7, but these have been removed.

The differences between the normalised bed stress magnitudes were assessed using the normalised bed stress profiles. The normalised bed stresses (without anomalous results) at 1D, 2D, 5D, 10D and 20D in each set of experiments are shown in Figure 8.21:

*Figure 8.21: Experimental bed shear stresses with stators and stators/rotors at 1D, 2D, 5D, 10D and 20D*

The results show that there is a significant variation between the experiments, particularly noticeable at 1D and 2D downstream, where the maximum normalised bed stress varies by 100% at 1D and 81% at 2D. However, as distance from the barrage...
increases this variation reduces. At 5D there is a 32% difference in the maximum normalised bed stress and this reduces to 8% at 10D and 6% at 20D. The peaks in bed stress close to the barrage occur at similar locations across the flume, especially the stator only and stator/rotor results. At 5D, it can be seen that the peak bed stress location slightly changes as the amount of swirl increases: this causes the peak in the velocity to move from the right side of the flume towards the centre. This is true at 10D also, but the variation between the results is diminished. At 20D the results are in good agreement.

8.4.2 LOG-LAW LAYER ASSESSMENT

The accuracy of the log-law assumption was again assessed using the wall $z^+$ values, shown in Figure 8.22.

![Experimental $z^+$ values](image)

*Figure 8.22: Experimental $z^+$ values*
The $z^+$ values are generally between 30 and 200 at further than 5D downstream, but closer to the barrage the $z^+$ values are as high as 400. This is due to the high amount of swirl created by the rotors and the increased inlet velocity. The $z^+$ values are only high in limited regions of the flow and within these regions the bed friction would not be the dominant factor in the flow dynamics.

8.4.3 BED SHEAR STRESS RATIO

The accuracy of these depth-averaged assumptions of a constant coefficient of friction and depth-averaged velocities was again assessed using the ratio of the magnitude of local bed shear stress, $\tau_b$, to a reference stress, $\tau_0$. The reference bed stress was 0.115 N m$^{-2}$, calculated using the skin friction coefficient computed from Equation 35, 0.0136, and the average velocity at 20D, 0.13 m s$^{-1}$. Contours of the ratios in the stator and the stator/rotor experiments are shown in Figure 8.23.

The experimental stator and stator/rotor contours show that downstream of the barrage the local bed stress can be up to 9 or 12 times the reference stress where the jets become attached to the bed. At 20D downstream the ratio is approximately 1 across the full flume, indicating the region where these assumptions can be used, because the 3-D effects are reduced.
Figure 8.23: Ratio of bed stress magnitude to reference bed stress,

a) Stators only  b) Stators/rotors
The differences between the results are shown in the ratio profiles at 1D, 2D, 5D, 10D and 20D in Figure 8.24.

![Figure 8.24: Experimental ratios of bed stress to reference stress](image)

The variation in the maximum bed stress ratios are 52 %, 38 %, 1 %, 22 % and 6 % at 1D, 2D, 5D, 10D and 20D respectively, excluding the anomalous results. This is similar to the variation in the normalised bed stresses and the profiles are also similar.

### 8.4.4 FULL-SCALE THRESHOLD OF MOTION

The full-scale threshold of motion contours for all the experiments are shown in Figure 8.25. With the addition of swirl the bed stress patterns can be seen to alter, but there is little difference in the stress patterns between the stator and the stator/rotor patterns. The stator/rotor contours show very comparable results to those with stators only, though the bed stress magnitude is higher due to the increased swirl and inlet velocity. Sediment motion will occur at all locations on the bed in the full-scale model.
Figure 8.25: Full-scale ratio of silt Shields parameter to critical silt Shields parameter,

a) No turbine representation b) Stators only c) Stators/rotors
8.5 CONCLUSIONS

Three main issues were addressed in this chapter:

1. How does a barrage affect the bed stresses downstream?

2. How well this is predicted by 3-D CFD and 2-D modelling?

3. Does a full-scale barrage cause bed movement?

1. Marked bed stresses occur where the jets attach to the bed; at approximately 5D downstream when there is no swirl and closer to the barrage at 2D when swirl is present. The central jet that forms close to the bed leads to higher bed stresses at the centre of the duct, which moves to the left side of the flume with increased swirl and Coandă effect. When eddies are present there is strong reversed bed stresses at the flume edge, with areas of low bed stress at the eddy centres. 3-D effects are very marked, particularly at the region of jet attachment. This can cause multiples of the reference stress (calculated using a constant coefficient of friction and a depth-averaged velocity) of up to 3 (no turbines), 9 (stators only) and 12 (stators and rotors) times. Depth-averaged assumptions are, therefore, only valid at 20D downstream.

2. When there is no swirl, StarCCM+ accurately models the bed stress profiles, but the bed stresses extracted directly from the model are over predicted, due to the different calculation methods. When using the velocities at 1 cm from the bed and the same calculation method as the experiments, the $z^+$ and friction velocities are similar, indicating that the bed stresses would also be similar. When swirl is added, the models become less accurate, similar to the velocity vectors and profiles in Chapter 6, particularly when there was a high amount of swirl.
The SW2D model was not accurate in predicting the bed stresses, because the jets were located as if on the bed, so no jet spreading occurred. The asymmetry and coefficient of friction were all under predicted.

3. When sand was added to the full-scale barrage, with an example grain size of 1 mm, the critical Shields parameter was exceeded at all regions within 20D. This indicated the presence of a mobile bed, though the ratio of Shields parameter to critical Shields parameter close to the barrage and at the flume sides was low, indicating areas of potential sediment deposition. Sediment transport is a dynamic equilibrium, so only occurs where there is a divergence of stress. If the stress is decreasing, as it is at the flume sides and further downstream, then accretion occurs. Changes in the bed profile due to scour and deposition may affect turbine performance and downstream flow field, but equilibrium may be reached, where no more scour at the barrage occurs. Barrages will use angled walls so the pattern of erosion may differ from that shown here, but modelling of this erosion is still required.
9. CONCLUSIONS AND FURTHER WORK

9.1 SUMMARY OF RESEARCH

The research had six key objectives and the results of this research with regards to these objectives have been summarised below.

- Investigation of the near-field flow downstream of a barrage through velocity profiles

The experiments were based on a 1:70\textsuperscript{th} geometric scale model of a proposed design for the Severn Barrage. The head difference across the barrage imposed in each experiment is based on the head difference found to maximise power output of the full-scale barrage and the water level drops with distance from the barrage. Close to the barrage, particularly at 1D and 2D downstream, jets are apparent at the culvert exit depths. Close to the surface, however, no jets are apparent. Asymmetric eddies form close to the barrage causing jet merging. As distance from the barrage increases the velocity variation across the width and depth is reduced. The profiles of streamwise velocity are asymmetric at all depths due to the Coandă effect and dependent on vertical level up to 10D downstream of the barrage and have become almost uniform by 20D downstream.

- Investigation of the effect of different stator/rotor configurations on the flow field

The water depth profile is similar to the open duct flow conditions, but the water height drops 40\% more in the stator, or guide vane, experiments, due to swirl-induced losses and drag due to the bulb and stator bodies. The swirl created by the stators and the bulb bodies causes wakes to form downstream from the bulb bodies and jets to form at the
duct edges; this is very different from the velocity profile of the open ducts, which had normal jet profiles. The jets at the duct edges also merge quicker than those in the open duct experiment, leading to more uniform flow at 5D.

Strong cross flow occurs close to the bed in one direction and close to the surface in the opposite; particularly at 1D and 2D downstream. At this distance the swirl occurs downstream from each duct exit, whereas further downstream, at 5D and 10D, circulation throughout the full cross-section of the flume occurs. At 20D the cross-stream and vertical flow is near-zero, and the streamwise flow is nearly uniform across the flume, similarly to the open duct results. This dissipation of the circulation is related to the flume width and the duct geometries; for fewer ducts or a wider flume the channel circulation at 5D and 10D may not develop.

Rotors were incorporated in the model to investigate the effect of dynamic swirl on the jet circulation and angular momentum flux, particularly on jet merging. Due to relatively high mechanical friction the angular speed of the rotors was only approximately one-eighth of a Froude scale model of representative barrage turbines. The stator and stator/rotor water depth decrease over the downstream flume length were very similar. The addition of rotors and increased inlet velocity did, however, increase the non-dimensionalised circulation by 39% and the angular momentum flux by 30%. Negligible change to the streamwise velocity profiles, jet profiles and depth variation was observed. The cross-stream and vertical velocities are slightly magnified close to the barrage, but further from the barrage at 5D and 10D the channel circulation is less defined.
• Determination of velocity variation with depth in the flow

There is significant variation in the velocities up to 10D from the barrage, but after 20D this variation is minimal. Within 2D, the key difference throughout the depth is the jet profile at the duct height and the velocity profile close to the surface. Eddies are also present at all depths in the flow, but of different widths and magnitudes. With the addition of swirl by stators the direction of the cross-stream flow varies with depth at this distance downstream. Within 10D there is asymmetry in the velocity profiles that varies with depth when there are no turbines present. With the addition of swirl there is full flume circulation, so cross-stream velocity varies with depth; this feature would be lost with depth-averaging. At 20D downstream there is no significant velocity variation across the depth and vertical and cross-stream velocities are minimal justifying the depth-averaged assumption.

• Investigation of the effect of a barrage on the bed stresses and, thus, sediment transport;

The jet at the centre of each duct that forms close to the bed leads to higher bed stresses downstream from the duct midlines; the jet moves to the left side of the flume due to the Coandă effect and increased swirl. Eddies cause strong reversed bed stresses at the flume edge, with areas of low bed stress at the eddy centres. 3-D effects are very marked, particularly at the region of jet attachment. This can cause multiples of the reference stress to the magnitude of 3 (no turbines), 9 (guide vanes/stators only) and 12 (guide vanes/stators and rotors). Depth-averaged assumption is, therefore, only valid at 20D downstream.
When sand was included in the full-scale barrage model (in both the experiments and CFD), with an example grain size of 1mm, the critical Shields parameter was exceeded in most regions within 20D. This indicated that the bed would be mobile and scour/erosion would occur. However, the ratio of Shields parameter to critical Shields parameter close to the barrage and at the flume sides was low, indicating areas of potential sediment deposition. Changes in the bed profile due to scour and deposition may affect turbine performance as the downstream flow field would be modified; equilibrium may be reached, where no more scour at the barrage occurs.

- Determination of how well the flow field is predicted by a 2-D depth-averaged model

Depth-averaged modelling was expected to be inadequate close to the barrage and this work suggests that it only becomes valid at least 20D downstream. 3-D effects also clearly affect the surface profile within 20D of the barrage; the streamwise water depth variation is much greater than that predicted by the depth-averaged model with a standard bed friction coefficient. The jet magnitudes, jet sizes and eddies were underpredicted and no asymmetry was apparent in the model. The large variation across the depth was not modelled, so SW2D modelling only became accurate at 20D downstream. The SW2D model was not accurate in predicting the bed stresses, because the jets were located as if on the bed, so no vertical jet spreading occurred. The asymmetry of the streamwise velocity, coefficient of friction and bed stress patterns were all under predicted, though the model did still indicate that a mobile bed would occur.
• Determination of whether the flow field can be accurately modelled using 3-D CFD

The 3-D computational modelling using StarCCM+ provides reasonable prediction of experimental measurements when there was no turbine present and thus no swirl. The velocity vectors and streamwise velocity profiles are in close agreement. 3-D CFD only slightly underestimates the water level variation.

When bulb bodies and stators were added to the model causing swirl, StarCCM+ becomes less effective in predicting the velocity profiles. When the head difference and angular momentum flux were accurately predicted, using the realisable \( k-e \) turbulence model with fixed body forces, the velocity profiles did not exhibit the same jet profile as the experiments and the profiles only became comparable when swirl had decayed at 20D downstream. The simulation that produced the most accurate close-to-barrage velocity profile was the high swirl coefficient model, but the tangential velocity, and thus swirl, is over predicted leading to overestimated spreading of the jets and reduced Coandă effect. This causes inaccurate velocity profiles at 5D and 10D, but by 20D the jets have merged and the model is accurate again. StarCCM+ does not accurately predict the flow created by the bulb bodies and stators, because it cannot simulate the swirl downstream from the ducts exits accurately. The bulb body and body force method also did not produce accurate results for the rotor results. The fan momentum model produced better swirl results, but the bulb body wakes were again under-predicted and the jets, eddies and Coandă effect were over-predicted leading to a profile in weaker agreement at 20D downstream. The velocity profile results may be improved if a performance curve is included for the rotors. In conclusion, when there is swirl in
the model, the correct velocity profiles downstream cannot be modelled using StarCCM+ and the turbulence, field function or fan function modelling methods.

When there is no swirl, StarCCM+ accurately models the bed stress profiles, but the bed stresses extracted directly from the model are slightly over predicted; this may be due to the different calculation methods. When using the velocities at 1 cm from the bed and the same calculation method as the experiments, the $z^+$ values and bed stresses were in closer agreement. When swirl was included, the models became less accurate, similar to the inaccuracies in the velocity vectors and profiles in Chapter 6, particularly when there was a high amount of swirl.
9.2 FUTURE WORK

Some parts of this research have been highly successful and provided conclusive answers to the research objectives; however, there are areas that require further research. There are also areas that, due to the work already undertaken, have raised questions about other flow conditions. The proposed extensions to the work are outlined below:

- In the experiments the velocities were recorded at only 6 depths throughout the flume. This was partly associated with the relatively time consuming experimental method of manual repositioning of the probe head. A finer grid resolution, particularly in the vertical plane, would provide further detail of the downstream flow which would be particularly useful close to the bed.

- The first set of experiments, with no turbines, was only conducted with a 2-D probe, so the vertical velocities could not be assessed; to determine whether full flume circulation occurred would require further 3-D velocity recordings.

- When the bulb bodies were added to the model the stators imparted swirl to the flow, which couldn’t be adequately modelled using CFD; if the bulbs were added firstly without stators then the effect of the bulb wake and culvert jets would be separated from the swirl effect for further clarification.
• The rotors were not able to achieve the required rotational speeds, scaled from the La Rance barrage turbines, so an improved rotor configuration could be used. Better blade profiles or ball bearings could be used to increase the rotation rate, which would be more representative of a full-scale tidal barrage turbine.

• The swirl modelling in CFD did not produce accurate results, so further investigation of the CFD model would be required to create a better model with swirl added.

• As an extension to the experimental study the effects of asymmetry could be investigated; for example, the effect of counter-rotating turbines, asymmetry in the duct locations or a sloping bed could be studied.

• Three further experiments could be run to determine a variety of effects. Firstly, the scale effects could be investigated, by repeating the experiments at a larger scale to increase the Reynolds number and ensure the flow is fully turbulent.

• The larger scale experiment would need to be conducted in a larger flume, so additionally the influence of the flume wall proximity could be examined, particularly its effect on the Coandă effect and the formation of cross-channel circulation.
Lastly, the experiments could be repeated with different types of turbines, such as low head bi-directional turbines, to determine the different resulting flow effects. The effect of different turbines on the head difference across the barrage, water levels downstream of the barrage, downstream flow patterns and bed shear stresses could be analysed.

Within the CFD model, the effect of time varying the inlet conditions could be assessed. The upstream flow rate and water heights could be altered to simulate the tidal cycle and the resulting effect on the flow field and bed stresses could be studied.

A further study of the CFD model could be to input the bathymetry of a potential tidal barrage site, such as the Mersey, with the appropriate sediment structure to determine the full scale sediment transport at a specific site.

There are several possible improvements to the CFD model, but this thesis work provides a good basis for further CFD research into tidal barrage modelling. The work in this thesis, such as the limit of applicability of depth-averaged modelling and the 3-D experimental flow field, is hopefully of use to tidal engineers and researchers.
LIST OF REFERENCES


Source: EDF, Dossier de Presse (Nov 2009). L’usine marémotrice de la Rance: 40 ans d’exploitation au service d’une production d’électricité inépuisable sans CO2.
**Turbine caisson for bulb turbines 9m diameter**


**Elevation (Dimensions in m)**

**Plan (Dimensions in m)**
**Scaled turbine caisson**

**Scaling**

Caisson height = 41 m

Caisson width = 19

Original experimental tank dimensions = 0.31 m height x 0.3 m width

(NB this flume was not used for the experiments, because the flume sides were too short, but the model was scaled originally using these dimensions)

Model Scale Factor = 0.007

Model height = 287 mm

Model width = 133 mm x 2 = 266 mm

**Elevation (Dimensions in mm)**
Plan (Dimensions in mm)

End (Dimensions in mm)
APPENDIX 3: MODIFIED TANK DIMENSIONS FOR TUBE OF 0.11M DIAMETER
APPENDIX 4: BARRAGE ENGINEERING DRAWINGS
**APPENDIX 5: LA RANCE TURBINE**


NOTE
Machine blanks without
With
Slot details to be
confirmed