

# Numerical Prediction of Natural Convection in Gas Turbines



Andrew James Pilkington

St Anne's College

University of Oxford

A thesis submitted for the degree of

*Doctor of Philosophy*

Trinity 2018

## **Abstract**

Engineers desire to be able to predict natural convection flow and heat transfer in high Rayleigh number flows, such as those found in gas and steam turbines during shutdown. Numerical simulations are used in order to predict these natural convection flows. The simulations must have a relatively low computational cost which is why the bulk of engineering simulations are Reynolds Averaged Navier Stokes (RANS) simulations.

In order to improve the predictions of natural convection heat transfer, new models for the turbulent heat flux have been implemented into a commercial solver. It was found that the newly implemented Generalised Gradient Diffusion Hypothesis (GGDH) and Algebraic Flux Model (AFM) were able to give improved predictions over the baseline model. The implemented models were then able to give predictions of wall heat transfer to within 20% of experimental values, making them suitable for engineering design calculations. A modification to the GGDH model was also developed, the GGDH+ model, in order to give improved predictions in regions with thermally stratified flow.

The models were validated using data from a rig representative of a large industrial gas turbine during shutdown. Experimental data from the casing rig was complemented by Large Eddy Simulations (LES) of the rig, giving a larger validation dataset. A fundamental investigation into vortex flow structures that occurred in the rig was also conducted. The RANS simulations were also validated using literature experimental data which was again complemented by Direct Numerical Simulations (DNS) of the literature cases.

An investigation was conducted into methods to control the natural convection flow and heat transfer inside of a gas turbine. The aim was to reduce the non-uniformity of heat transfer around the engine casing which would reduce casing distortion during shutdown. It was shown that flow extraction methods could improve the heat transfer distribution.



## **Acknowledgements**

Firstly I would like to thank Dr Budimir Rosic for his advice and supervision during this project.

I would like to thank Mitsubishi Heavy Industries Ltd. for sponsoring this work. I would also like to thank all of the engineers at MHI for their hospitality during my two visits to Japan. In particular I would like to thank Tanimoto-san and Horie-san for supporting this project.

I would also like to thank Dr Kemal Hanjalić for his advice, suggestions and patience when answering my many questions!

Finally I would like to thank Craig Parsons for all of his very prompt IT help, and to thank all of my friends at the Osney lab for making my DPhil so enjoyable.

# Contents

|          |                                                      |          |
|----------|------------------------------------------------------|----------|
| <b>1</b> | <b>Introduction</b>                                  | <b>1</b> |
| 1.1      | Background . . . . .                                 | 1        |
| 1.2      | Motivation . . . . .                                 | 3        |
| 1.3      | Project Aims . . . . .                               | 3        |
| 1.4      | Thesis Outline . . . . .                             | 5        |
| <b>2</b> | <b>Literature Review</b>                             | <b>7</b> |
| 2.1      | Natural Convection Flows . . . . .                   | 7        |
| 2.2      | Natural Convection Studies in Gas Turbines . . . . . | 8        |
| 2.3      | Numerical Methods for Natural Convection . . . . .   | 10       |
| 2.3.1    | Direct Numerical Simulation . . . . .                | 11       |
| 2.3.2    | Large Eddy Simulation . . . . .                      | 12       |
| 2.3.3    | Reynolds Averaged Navier Stokes . . . . .            | 14       |
| 2.4      | RANS Equations . . . . .                             | 14       |
| 2.5      | Turbulent Heat Flux Modelling . . . . .              | 15       |
| 2.5.1    | Simple Gradient Diffusion Hypothesis . . . . .       | 15       |
| 2.5.2    | Generalised Gradient Diffusion Hypothesis . . . . .  | 17       |
| 2.5.3    | Algebraic Flux Model . . . . .                       | 17       |
| 2.5.4    | Differential Flux Model . . . . .                    | 18       |
| 2.6      | Natural Convection Boundary Layer . . . . .          | 18       |
| 2.6.1    | Velocity Profile . . . . .                           | 18       |
| 2.6.2    | Temperature Profile . . . . .                        | 19       |
| 2.7      | Validation Test Cases . . . . .                      | 21       |

|          |                                          |           |
|----------|------------------------------------------|-----------|
| 2.7.1    | Measurement Challenges . . . . .         | 22        |
| 2.7.2    | Heated Vertical Plate . . . . .          | 23        |
| 2.7.3    | Tall Cavity . . . . .                    | 25        |
| 2.7.4    | Heated Cylinder . . . . .                | 26        |
| 2.7.5    | Square Cavity . . . . .                  | 26        |
| 2.8      | Conclusions . . . . .                    | 27        |
| <b>3</b> | <b>Gas Turbine Casing Rig</b>            | <b>29</b> |
| 3.1      | Introduction . . . . .                   | 29        |
| 3.2      | Casing Rig Design . . . . .              | 31        |
| 3.2.1    | Geometry . . . . .                       | 31        |
| 3.2.2    | Measurement . . . . .                    | 34        |
| 3.3      | LES methodology . . . . .                | 35        |
| 3.4      | Casing Rig Flowfield . . . . .           | 38        |
| 3.4.1    | Comparison to Vertical Wall . . . . .    | 38        |
| 3.4.2    | Temperature and Velocity field . . . . . | 38        |
| 3.4.3    | Outer Wall Heat Transfer . . . . .       | 42        |
| 3.4.4    | Streamwise Vortices . . . . .            | 42        |
| 3.5      | Conclusions . . . . .                    | 46        |
| <b>4</b> | <b>Tall Cavity Study</b>                 | <b>49</b> |
| 4.1      | Background . . . . .                     | 49        |
| 4.2      | Numerical Method . . . . .               | 51        |
| 4.3      | Simulation Validation . . . . .          | 53        |
| 4.3.1    | Numerical Mesh Validation . . . . .      | 53        |
| 4.3.2    | Experimental Data Validation . . . . .   | 55        |
| 4.4      | Tall Cavity Flowfield . . . . .          | 59        |
| 4.4.1    | Mean Flowfield . . . . .                 | 59        |
| 4.4.2    | Instantaneous Flowfield . . . . .        | 61        |
| 4.4.3    | Wall Heat Transfer . . . . .             | 62        |

|          |                                          |            |
|----------|------------------------------------------|------------|
| 4.5      | Turbulent Heat Flux . . . . .            | 63         |
| 4.6      | Conclusions . . . . .                    | 66         |
| <b>5</b> | <b>Turbulent Heat Flux Modelling</b>     | <b>67</b>  |
| 5.1      | Implementation . . . . .                 | 68         |
| 5.2      | Vertical plate validation case . . . . . | 70         |
| 5.2.1    | Numerical method . . . . .               | 70         |
| 5.2.2    | Results & Discussion . . . . .           | 72         |
| 5.2.3    | Vertical plate conclusions . . . . .     | 75         |
| 5.3      | Tall cavity validation case . . . . .    | 77         |
| 5.3.1    | Numerical method . . . . .               | 77         |
| 5.3.2    | Results & Discussion . . . . .           | 78         |
| 5.3.3    | Tall cavity conclusions . . . . .        | 87         |
| 5.4      | Casing Rig validation case . . . . .     | 88         |
| 5.4.1    | Numerical method . . . . .               | 88         |
| 5.4.2    | Results & Discussion . . . . .           | 90         |
| 5.4.3    | Buoyancy Modification . . . . .          | 96         |
| 5.4.4    | Computational Cost . . . . .             | 100        |
| 5.4.5    | Reduced Mesh study . . . . .             | 103        |
| 5.4.6    | Casing Rig conclusions . . . . .         | 105        |
| 5.5      | Conclusions . . . . .                    | 108        |
| <b>6</b> | <b>Streamwise Vortex Study</b>           | <b>111</b> |
| 6.1      | Background . . . . .                     | 112        |
| 6.2      | Numerical method . . . . .               | 114        |
| 6.3      | Results & Discussion . . . . .           | 114        |
| 6.3.1    | Vortex Formation . . . . .               | 114        |
| 6.3.2    | Heat Transfer Enhancement . . . . .      | 119        |
| 6.3.3    | Wall Temperature Study . . . . .         | 121        |
| 6.3.4    | Prandtl Number study . . . . .           | 124        |

|          |                                               |            |
|----------|-----------------------------------------------|------------|
| 6.3.5    | Heated Inner Wall . . . . .                   | 126        |
| 6.4      | Comparison with Hairpin Vortices . . . . .    | 131        |
| 6.5      | Conclusions . . . . .                         | 133        |
| <b>7</b> | <b>Shutdown Flow Control in Gas Turbines</b>  | <b>135</b> |
| 7.1      | Introduction . . . . .                        | 135        |
| 7.2      | Investigation . . . . .                       | 137        |
| 7.3      | Test Case . . . . .                           | 139        |
| 7.4      | Numerical Method . . . . .                    | 140        |
| 7.5      | Baseline Investigation . . . . .              | 142        |
| 7.6      | Turbulent Heat Flux Model . . . . .           | 146        |
| 7.7      | Heat Transfer Control Investigation . . . . . | 149        |
| 7.7.1    | Purge Case . . . . .                          | 150        |
| 7.7.2    | Extraction Case . . . . .                     | 153        |
| 7.7.3    | Two Hole Extraction . . . . .                 | 157        |
| 7.8      | Comparison . . . . .                          | 158        |
| 7.9      | High Mass Flow Case . . . . .                 | 160        |
| 7.10     | Post Shutdown Case . . . . .                  | 163        |
| 7.10.1   | Results and Discussion . . . . .              | 164        |
| 7.11     | Conclusions . . . . .                         | 166        |
| <b>8</b> | <b>Conclusions</b>                            | <b>169</b> |
| 8.1      | Summary . . . . .                             | 169        |
| 8.2      | Conclusions . . . . .                         | 171        |
| 8.3      | Future Work . . . . .                         | 173        |
| 8.4      | Final Remarks . . . . .                       | 174        |
|          | <b>Bibliography</b>                           | <b>175</b> |
| <b>A</b> | <b>Turbulence Modelling Study</b>             | <b>187</b> |
| A.1      | Turbulence Model . . . . .                    | 187        |

|          |                                                 |            |
|----------|-------------------------------------------------|------------|
| A.2      | Eddy-viscosity model . . . . .                  | 187        |
| A.2.1    | Spalart-Allmaras Model . . . . .                | 187        |
| A.2.2    | $k - \epsilon$ Model . . . . .                  | 188        |
| A.2.3    | $k - \omega$ Model . . . . .                    | 188        |
| A.2.4    | $k - \omega$ <i>SST</i> Model . . . . .         | 188        |
| A.3      | Reynolds Stress Model . . . . .                 | 189        |
| A.4      | Buoyancy Effects in Turbulence Models . . . . . | 189        |
| A.5      | Turbulence Model Study . . . . .                | 190        |
| A.6      | Buoyancy Effect Study . . . . .                 | 191        |
| A.7      | Conclusions . . . . .                           | 193        |
| <b>B</b> | <b>Casing Rig Inlet Design</b>                  | <b>196</b> |
| B.1      | Initial Inlet Design . . . . .                  | 196        |
| B.2      | Contraction Design . . . . .                    | 196        |
| B.2.1    | Contraction . . . . .                           | 197        |
| B.2.2    | Flow Conditioning Features . . . . .            | 199        |
| B.3      | Inlet Flowfield Validation . . . . .            | 199        |
| B.4      | Conclusions . . . . .                           | 200        |
| <b>C</b> | <b>Instrumentation Design</b>                   | <b>201</b> |
| C.1      | Introduction . . . . .                          | 201        |
| C.2      | Thermocouples . . . . .                         | 201        |
| C.3      | Fibre Measurement . . . . .                     | 203        |
| C.4      | Velocity Traverse . . . . .                     | 203        |
| C.5      | Laser Based Velocity Measurement . . . . .      | 204        |

# List of Figures

|      |                                                                                                                                  |    |
|------|----------------------------------------------------------------------------------------------------------------------------------|----|
| 1.1  | Industrial Gas Turbine, modified from [56] . . . . .                                                                             | 4  |
| 2.1  | Experimental [98] and modelled turbulent heat fluxes on vertical wall case .                                                     | 16 |
| 2.2  | Forced and natural convection boundary layer velocity profiles, data from<br>Kim et al. [39] and Tsuji and Nagano [97] . . . . . | 19 |
| 2.3  | Schematic of turbulent natural convection boundary layer, modified from<br>[35] . . . . .                                        | 21 |
| 2.4  | Schematic of Vertical Wall test case . . . . .                                                                                   | 24 |
| 2.5  | Tall Cavity Apparatus (modified from [6]) . . . . .                                                                              | 25 |
| 2.6  | Schematic of square cavity, modified from [1] . . . . .                                                                          | 27 |
| 3.1  | Schematic of flow in vertical wall and casing rig . . . . .                                                                      | 31 |
| 3.2  | Gas turbine casing rig, without contraction inlet . . . . .                                                                      | 32 |
| 3.3  | Gas Turbine Casing Rig Schematic . . . . .                                                                                       | 32 |
| 3.4  | Cross section of computational mesh from LES simulation . . . . .                                                                | 36 |
| 3.5  | History monitor of outer wall heat transfer . . . . .                                                                            | 37 |
| 3.6  | Heat Transfer Profile of Casing Rig and Vertical Wall . . . . .                                                                  | 39 |
| 3.7  | Time average temperature and velocity contours from LES, on mid-plane<br>of casing rig . . . . .                                 | 40 |
| 3.8  | Thermocouple and fibre experimental and LES temperature profiles at six<br>measurement locations . . . . .                       | 41 |
| 3.9  | Contours of outer wall heat transfer in casing rig from time averaged LES .                                                      | 43 |
| 3.10 | Contours of Q-criterion showing streamwise vortices next to heated wall . .                                                      | 44 |
| 3.11 | Contours of helicity at section 6 of casing rig, looking into rig . . . . .                                                      | 45 |

|      |                                                                                                                                  |    |
|------|----------------------------------------------------------------------------------------------------------------------------------|----|
| 3.12 | Schematic showing effect of streamwise vortices on wall heat transfer . . .                                                      | 46 |
| 3.13 | LES and experimental temperature contours at Section 6 of casing rig . . .                                                       | 47 |
| 4.1  | Schematic of tall cavity test case, modified from [6] . . . . .                                                                  | 50 |
| 4.2  | Mid-plane section of mesh used for DNS simulation . . . . .                                                                      | 52 |
| 4.3  | Ratio of cell size to Kolmogorov length-scale at cavity mid height . . . . .                                                     | 54 |
| 4.4  | Spectrum plot of DNS velocity at a point on the cavity mid plane at 50%<br>height and 8% of the cavity width . . . . .           | 55 |
| 4.5  | Non-dimensional mean temperature profile on cavity mid-plane at 50%<br>cavity height . . . . .                                   | 56 |
| 4.6  | Non-dimensional rms temperature profile on cavity mid-plane at 50% cav-<br>ity height . . . . .                                  | 57 |
| 4.7  | Spectrum plot of DNS temperature signal at a point on the cavity mid plane<br>at 50% height and 8% of the cavity width . . . . . | 58 |
| 4.8  | Non-dimensional mean vertical velocity profile on cavity mid-plane at 50%<br>cavity height . . . . .                             | 59 |
| 4.9  | Non-dimensional rms vertical velocity profile on cavity mid-plane at 50%<br>cavity height . . . . .                              | 60 |
| 4.10 | Mean and rms temperature and velocity at cavity mid-plane . . . . .                                                              | 61 |
| 4.11 | Instantaneous temperature and velocity at cavity mid-plane . . . . .                                                             | 62 |
| 4.12 | Iso-surface of Q-criterion in bottom half of the cavity . . . . .                                                                | 63 |
| 4.13 | Contours of Nusselt Number on cavity hot wall . . . . .                                                                          | 64 |
| 4.14 | Contours of non-dimensional turbulent heat flux on cavity mid-plane . . . .                                                      | 64 |
| 4.15 | Temperature gradients on cavity mid-plane . . . . .                                                                              | 65 |
| 5.1  | Vertical plate heat transfer using unmodified internal SGD model and<br>UDF model . . . . .                                      | 69 |
| 5.2  | Cross section of vertical plate computational mesh . . . . .                                                                     | 71 |
| 5.3  | Temperature and Velocity profiles at different heights (in m) from bottom<br>of the heated plate . . . . .                       | 73 |
| 5.4  | Inner region thermal boundary layer profiles . . . . .                                                                           | 73 |



|      |                                                                                                    |     |
|------|----------------------------------------------------------------------------------------------------|-----|
| 5.5  | Heat transfer from vertical plate case using different turbulent heat flux models . . . . .        | 75  |
| 5.6  | Vertical plate turbulent heat fluxes profiles at $y=2.54$ m . . . . .                              | 76  |
| 5.7  | Schematic cross section of tall-cavity . . . . .                                                   | 78  |
| 5.8  | Contours of temperature and vertical velocity on cavity mid-plane . . . . .                        | 79  |
| 5.9  | Temperature profiles on mid-plane of tall cavity at different vertical heights . . . . .           | 80  |
| 5.10 | Near wall temperature profiles on mid-plane of tall cavity at different vertical heights . . . . . | 82  |
| 5.11 | Vertical velocity profiles on mid-plane of tall cavity at different vertical heights . . . . .     | 84  |
| 5.12 | Profiles of $\overline{u'T'}$ turbulent heat flux in tall cavity . . . . .                         | 85  |
| 5.13 | Profiles of $\overline{v'T'}$ turbulent heat flux in tall cavity . . . . .                         | 86  |
| 5.14 | Casing rig cross section of computational mesh . . . . .                                           | 89  |
| 5.15 | Surface monitor of outer wall heat transfer . . . . .                                              | 89  |
| 5.16 | Temperature profiles at six measurement locations in casing rig . . . . .                          | 91  |
| 5.17 | Wall temperature gradient at section 6 . . . . .                                                   | 92  |
| 5.18 | Wall temperature gradient at section 5 . . . . .                                                   | 93  |
| 5.19 | Wall temperature gradient at section 4 . . . . .                                                   | 93  |
| 5.20 | Time averaged heat transfer contours on casing rig outer wall . . . . .                            | 95  |
| 5.21 | Circumferential turbulent heat flux term in casing rig . . . . .                                   | 97  |
| 5.22 | Wall normal turbulent heat flux term in casing rig . . . . .                                       | 98  |
| 5.23 | Time averaged heat transfer contours on casing rig outer wall . . . . .                            | 99  |
| 5.24 | Circumferential turbulent heat flux term in casing rig . . . . .                                   | 101 |
| 5.25 | Wall normal turbulent heat flux term in casing rig . . . . .                                       | 102 |
| 5.26 | Cross section of reduced casing rig computational mesh . . . . .                                   | 103 |
| 5.27 | Outer wall heat transfer contours, RANS simulations with reduced mesh . . . . .                    | 105 |
| 5.28 | Non-dimensional temperature profiles in casing rig, RANS simulations with reduced mesh . . . . .   | 106 |
| 6.1  | Schematic of thermal vortex wavelength . . . . .                                                   | 112 |

|      |                                                                                                                       |     |
|------|-----------------------------------------------------------------------------------------------------------------------|-----|
| 6.2  | Schematic of casing rig geometry . . . . .                                                                            | 114 |
| 6.3  | Non-dimensional $z$ -vorticity, streamwise velocity & temperature at three<br>centreline locations . . . . .          | 115 |
| 6.4  | Schematic showing formation of streamwise vortices . . . . .                                                          | 116 |
| 6.5  | Contours of non-dimensional $z$ vorticity, streamwise velocity & tempera-<br>ture on a plane at $7.5^\circ$ . . . . . | 117 |
| 6.6  | Stream-traces released 5 mm from outer wall, coloured by non-dimensional<br>circumferential velocity . . . . .        | 118 |
| 6.7  | Contours of non-dimensional helicity at section 6 . . . . .                                                           | 119 |
| 6.8  | Schematic of heat transfer enhancement caused by streamwise vortices . . .                                            | 120 |
| 6.9  | Heat transfer in first $45^\circ$ section of the outer wall . . . . .                                                 | 120 |
| 6.10 | Outer wall heat transfer across width of duct on a line $15^\circ$ from start of<br>heated wall . . . . .             | 121 |
| 6.11 | Contours of instantaneous non-dimensional outer wall heat transfer at dif-<br>ferent wall temperatures . . . . .      | 123 |
| 6.12 | Instantaneous contours of helicity at section 6 at different wall temperatures                                        | 123 |
| 6.13 | Vortex wavelength against critical Grashof Number for different wall tem-<br>peratures . . . . .                      | 124 |
| 6.14 | Vortex wavelength against critical Rayleigh Number varying fluid properties                                           | 126 |
| 6.15 | Contours of instantaneous non-dimensional outer wall heat transfer with<br>varying fluid properties . . . . .         | 127 |
| 6.16 | Cross section of mesh used for heated inner wall case . . . . .                                                       | 128 |
| 6.17 | History monitor of inner wall heat transfer . . . . .                                                                 | 129 |
| 6.18 | Cross section of non-dimensional temperature and velocity from heated<br>inner wall case . . . . .                    | 129 |
| 6.19 | Inner wall non-dimensional heat transfer . . . . .                                                                    | 130 |
| 6.20 | Iso-surfaces of helicity caused by secondary flows from heated inner wall<br>case . . . . .                           | 131 |
| 6.21 | Vortex wavelength in wall units against critical Rayleigh number . . . . .                                            | 132 |
| 6.22 | Schematic of hairpin vortex . . . . .                                                                                 | 133 |

|      |                                                                                                                        |     |
|------|------------------------------------------------------------------------------------------------------------------------|-----|
| 7.1  | Cross Section of Gas Turbine Casing, modified from [59]                                                                | 136 |
| 7.2  | Schematic Showing Axial and Radial Sections of a Gas Turbine Casing                                                    | 138 |
| 7.3  | Computational Domain                                                                                                   | 140 |
| 7.4  | Cross Section of Mesh                                                                                                  | 141 |
| 7.5  | Baseline Case, Time Averaged Stream-Traces, looking rearward into cavity                                               | 143 |
| 7.6  | Baseline Case, Velocity Vectors in Section Through Combustors 16 & 1                                                   | 144 |
| 7.7  | Baseline Case, Velocity Vectors in Axial Section C-C                                                                   | 144 |
| 7.8  | Baseline Case, Outer Wall Heat Flux With Surface Streamlines                                                           | 145 |
| 7.9  | Circumferential distribution of axially averaged outer wall heat transfer                                              | 145 |
| 7.10 | Baseline Case, Circumferential Distribution of Axially Averaged Outer<br>Wall Heat Flux                                | 146 |
| 7.11 | Outer wall Nusselt number contours using different turbulent heat flux<br>models                                       | 147 |
| 7.12 | Circumferential distribution of axially averaged outer wall heat transfer<br>with different turbulent heat flux models | 148 |
| 7.13 | GGDH+ case, time averaged stream-traces looking rearwards into cavity                                                  | 149 |
| 7.14 | Schematic of Purge & Extraction Cases                                                                                  | 150 |
| 7.15 | Purge Case, Time Averaged Stream-Traces, looking rearward into cavity                                                  | 151 |
| 7.16 | Purge Case, Outer Wall Heat Flux With Surface Streamlines                                                              | 152 |
| 7.17 | Velocity Vectors in Purge Case, Section Between Combustors 1 & 16                                                      | 152 |
| 7.18 | Circumferential Distribution of Velocity at Inlet                                                                      | 153 |
| 7.19 | Purge Case, Iso-Surface of Q-Criterion at Diffuser Exit                                                                | 154 |
| 7.20 | Extraction Case, Time Averaged Stream-Traces, looking rearward into cavity                                             | 155 |
| 7.21 | Extraction Case, Velocity Vectors in Axial Section C-C                                                                 | 155 |
| 7.22 | Extraction Case, Outer Wall Heat Flux With Surface Streamlines                                                         | 156 |
| 7.23 | Circumferential Distribution of Axially Averaged Outer Wall Heat Flux                                                  | 157 |
| 7.24 | Extraction case, circumferential outer wall heat flux with different extrac-<br>tion mass flows                        | 158 |
| 7.25 | Two hole extraction case, outer wall heat flux with surface streamlines                                                | 159 |

|      |                                                                                                                       |     |
|------|-----------------------------------------------------------------------------------------------------------------------|-----|
| 7.26 | Two hole extraction case, circumferential outer wall heat flux with different extraction mass flows . . . . .         | 159 |
| 7.27 | Comparison of heat flux on cylindrical section of outer casing . . . . .                                              | 160 |
| 7.28 | Circumferential distribution of axially averaged outer wall heat transfer for baseline and extraction cases . . . . . | 161 |
| 7.29 | Outer wall Nusselt number contours for baseline and high mass flow rate cases . . . . .                               | 162 |
| 7.30 | Velocity vectors in high mass flow case, Section between combustors 1 & 16                                            | 163 |
| 7.31 | Outer wall Nusselt number contours at different shutdown times . . . . .                                              | 165 |
| 7.32 | +2 hour case, time averaged stream-traces looking rearwards into cavity . .                                           | 166 |
| 7.33 | Circumferential Distribution of Axially Averaged Nusselt Number . . . . .                                             | 167 |
| A.1  | Non-dimensional temperature profiles in tall cavity with different turbulence models . . . . .                        | 192 |
| A.2  | Non-dimensional turbulent kinetic energy in tall cavity . . . . .                                                     | 194 |
| B.1  | Smoke flow visualisation in initial inlet . . . . .                                                                   | 197 |
| B.2  | Schematic of Contraction Inlet . . . . .                                                                              | 198 |
| B.3  | Inlet Contraction . . . . .                                                                                           | 198 |
| B.4  | Velocity profiles at rig inlet . . . . .                                                                              | 200 |
| C.1  | Thermocouple Rake . . . . .                                                                                           | 202 |
| C.2  | Optical fibre schematic . . . . .                                                                                     | 203 |

# List of Tables

|     |                                                                                                       |     |
|-----|-------------------------------------------------------------------------------------------------------|-----|
| 3.1 | Experiment and LES wall heat fluxes ( $W/m^2$ ) . . . . .                                             | 42  |
| 4.1 | Values used to non-dimensionalise Navier Stokes . . . . .                                             | 52  |
| 4.2 | Experimental and DNS cold wall heat flux at different cavity heights ( $W/m^2$ )                      | 56  |
| 5.1 | Differences in predicted wall heat transfer from DNS data . . . . .                                   | 83  |
| 5.2 | Fluent UDF increase in computational cost per time-step . . . . .                                     | 100 |
| 6.1 | Wavelength and critical distance for streamwise vortices at different wall<br>temperatures . . . . .  | 123 |
| 6.2 | Wavelength and critical distance for streamwise vortices with different fluid<br>properties . . . . . | 126 |
| 7.1 | Mesh independence study. . . . .                                                                      | 141 |
| A.1 | Difference in wall heat flux from experimental data with different turbu-<br>lence models . . . . .   | 191 |
| A.2 | Difference in wall heat flux from experimental data with buoyancy effects .                           | 193 |
| C.1 | Rake thermocouple locations . . . . .                                                                 | 202 |
| C.2 | Optical fibre row locations . . . . .                                                                 | 204 |

# Nomenclature

## Abbreviations

|      |                                           |
|------|-------------------------------------------|
| AFM  | Algebraic Flux Model                      |
| CFD  | Computational Fluid Dynamics              |
| CFL  | Courant Friedrichs Lewy                   |
| DNS  | Direct Numerical Simulation               |
| FEA  | Finite Element Analysis                   |
| GGDH | Generalised Gradient Diffusion Hypothesis |
| GT   | Gas Turbine                               |
| LDA  | Laser Doppler Anemometry                  |
| LES  | Large Eddy Simulation                     |
| MHI  | Mitsubishi Heavy Industries               |
| RANS | Reynolds Averaged Navier Stokes           |
| rms  | Root Mean Square                          |
| SGDH | Simple Gradient Diffusion Hypothesis      |
| SST  | Shear Stress Transport                    |
| UDF  | User Defined Function                     |

UDM User Defined Memory

### **Greek Symbols**

|               |                                                       |
|---------------|-------------------------------------------------------|
| $\alpha$      | Thermal Diffusivity = $\frac{\lambda}{\rho c_p}$      |
| $\beta$       | Coefficient of Thermal Expansion                      |
| $\delta_{ij}$ | Kronecker Delta                                       |
| $\epsilon$    | Dissipation of turbulent kinetic energy               |
| $\Lambda$     | Vortex wavelength                                     |
| $\lambda$     | Thermal Conductivity                                  |
| $\nu$         | Kinematic Viscosity                                   |
| $\nu_t$       | Turbulent Viscosity                                   |
| $\omega$      | Specific Dissipation Rate of turbulent kinetic energy |
| $\rho$        | Density                                               |
| $\sigma$      | Standard Deviation                                    |
| $\tau$        | Characteristic time                                   |
| $\tau_w$      | Wall shear stress                                     |
| $\theta$      | Non-dimensional temperature                           |

### **Roman Symbols**

|         |                                                        |
|---------|--------------------------------------------------------|
| $c_\mu$ | Emperical constant = 0.09                              |
| $c_p$   | Specific Heat Capacity                                 |
| $g$     | Gravitational Acceleration                             |
| $Gr$    | Grashof Number = $\frac{g\beta(T-T_\infty)x^3}{\nu^2}$ |

|             |                                                             |
|-------------|-------------------------------------------------------------|
| $H$         | Casing Rig Height                                           |
| $h$         | Heat Transfer Coefficient                                   |
| $k$         | Turbulent Kinetic Energy                                    |
| $L$         | Characteristic Lengthscale                                  |
| $Nu$        | Nusselt Number = $\frac{hx}{\lambda}$                       |
| $Pr$        | Prandtl Number = $\frac{c_p \mu}{\lambda}$                  |
| $Pr_t$      | Turbulent Prandtl Number                                    |
| $q$         | Heat Flux                                                   |
| $R$         | Casing Rig Outer Wall Radius                                |
| $Ra$        | Rayleigh Number = $\frac{g\beta(T-T_\infty)x^3}{\nu\alpha}$ |
| $Ri$        | Richardson Number = $\frac{g\beta(T-T_\infty)x}{V^2}$       |
| $S_T$       | Energy Source Term                                          |
| $T$         | Temperature                                                 |
| $T'$        | Temperature Fluctuation                                     |
| $\tilde{T}$ | Instantaneous Temperature                                   |
| $\bar{T}$   | Mean Temperature                                            |
| $u^+$       | Dimensionless wall velocity = $u/U_\tau$                    |
| $u'$        | Velocity Fluctuation                                        |
| $\tilde{u}$ | Instantaneous Velocity                                      |
| $\bar{U}$   | Mean Velocity                                               |
| $U_\tau$    | Friction velocity = $\sqrt{\frac{\tau_w}{\rho}}$            |



|       |                                             |
|-------|---------------------------------------------|
| $V$   | Characteristic Velocity                     |
| $W$   | Casing Rig Width                            |
| $x$   | Characteristic Length                       |
| $y^+$ | Dimensionless wall distance = $yU_\tau/\nu$ |

# Chapter 1

## Introduction

### 1.1 Background

#### Natural Convection Problem

Natural convection is known to be the cause of several issues affecting gas and steam turbine power plants. During shutdown natural convection flows occur inside of the engine casings. These flows cause the cooling of the engine to be non-uniform, leading to increased stresses and distortion of engine components. This distortion may ultimately lead to larger clearances being required, causing a performance penalty.

Difficulties occur when trying to understand the natural convection flows occurring in these situations. Measuring air temperatures inside of an engine is difficult and only a few temperatures close to a wall can be measured. Uncertainty can be high in measurements due to thermocouples getting damaged or dislodged during engine running. Velocity measurements of the flow inside an engine cavity are also not possible. Furthermore there are few test rigs available to help understand the flowfield. Gas and steam turbines during shutdown operate at high Rayleigh numbers where the flow is turbulent, whilst most test facilities operate at much lower Rayleigh numbers and do not give representative flow conditions.

The natural convection flowfield inside of an engine would ideally be predicted using numerical techniques. It has been found that the RANS simulations available to engineers do not accurately predict natural convection heat transfer. Therefore to predict natural convection heat transfer engineers have to rely on correlations, often from simple geometries.

These correlations also cannot predict changes in heat transfer due to changes in design.

The lack of measurements, experimental rig data and reliable numerical simulations, makes it hard to understand the natural convection flow and implement new designs to alleviate the problems it causes.

## **Gas Turbine Casing**

A large industrial gas turbine of the type used for power generation can be seen in Fig.1.1. An industrial gas turbine has a thick outer casing which surrounds multiple annular combustors. Whilst an engine is running the dominant form of heat transfer inside of it is forced convection, giving a circumferentially uniform distribution of heat transfer around the engine casing. During shutdown the thin combustors cool much faster than the thick outer casing leading to a significant temperature difference between the different components. This temperature difference sets up buoyancy driven flows inside the engine and natural convection heat transfer starts to dominate. These natural convection flows occur throughout the engine but are dominant in the large cavity surrounding the combustor cans.

The flow-field inside a gas turbine casing is shown schematically in Fig.1.1. At the top of the casing a region of thermally stratified flow occurs, giving a region of almost stationary hot fluid. In this low velocity stratified region the heat transfer from the hot casing to the fluid can drop to almost zero. In the cavity below the stratified region natural convection flows occur, with fluid heating and rising near the outer casing wall. At the bottom of the casing colder fluid drawn through the engine sinks and impinges on the outer casing, leading to high heat transfer at the bottom of the casing. This difference of low heat transfer at the top of the casing and higher heat transfer at the bottom of the casing means that the top of the engine will cool slower than the bottom. This difference in the rates of cooling will cause the bottom of the casing to contract faster than the top of the casing, leading to “cat back” distortion of the casing.

This “cat back” distortion can cause clearances between rotating and static components to decrease. In order to mitigate the effect of the distortion, clearances have to be increased to avoid components rubbing or binding. These increased clearances can cause a performance loss when running, due to increased leakage flows and decreased surge margin.

The closure of components during shutdown also limits the operational flexibility of an engine. It would also be desirable to be able to switch on and off gas turbines in order to respond to changes in demanded electrical load, but it may not be possible to restart an engine that has partially cooled due to the reduced clearances.

## **1.2 Motivation**

In order to alleviate the problems occurring during shutdown engineers must understand the different rates of cooling and contraction of the components inside of the engine. The thermal contraction of components inside of an engine are typically predicted using Finite Element Analysis (FEA). In order to predict the contractions the FEA solver requires heat transfer coefficients (htc) to be provided as boundary conditions.

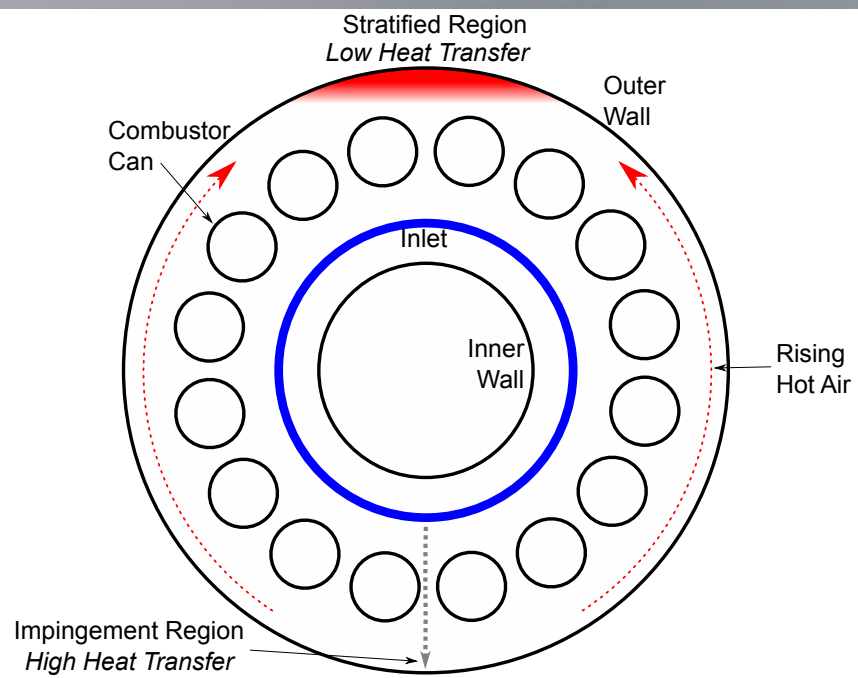
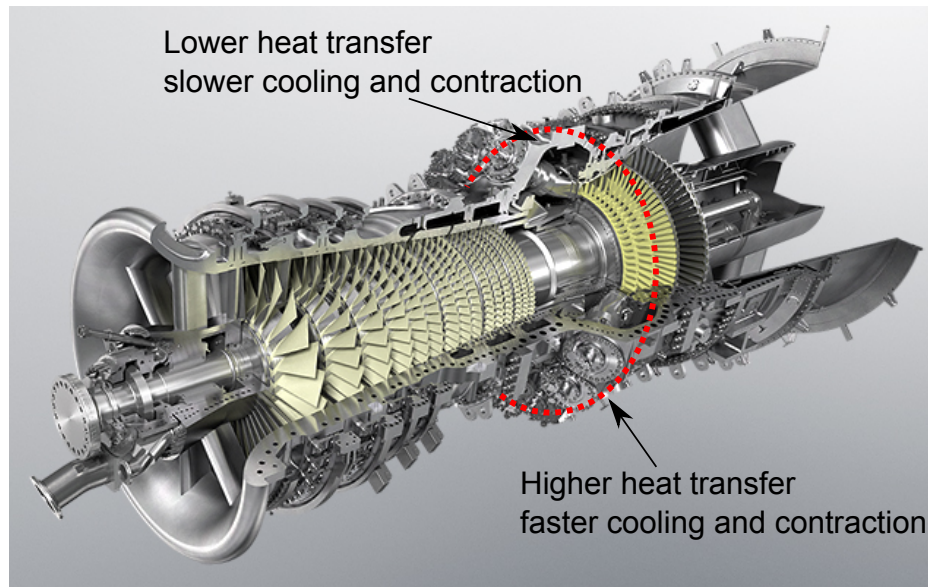
The htc boundary conditions could be provided by experimental data, but data from a real engine is limited and a way of predicting the effects on htc from new designs is also needed. The htc boundary conditions could therefore be predicted using Computational Fluid Dynamics (CFD) simulations. It has been found by engineers however that the current CFD methods are not able to accurately predict the natural convective flow and htc inside of an engine during shutdown. Engineers desire a CFD method that will give heat transfer predictions with an error of less than 20% and a reasonable computational cost.

The first step in alleviating the problems occurring in gas turbines during shutdown is therefore obtaining a reliable prediction of the natural convection flow inside of the engine.

## **1.3 Project Aims**

The overall aim of this project is to advance the state of the art in numerical predictions of natural convection flows in conventional power generation systems. The key project aims are:

- Improve the prediction of high Rayleigh number natural convective flows and heat transfer. The method should give heat transfer predictions with an error of less than 20% and have a computational cost low enough to be used by engineers.



**Figure 1.1:** Industrial Gas Turbine, modified from [56]

- Understand the natural convection flowfield and temperature profiles in geometry representative of a large gas turbine casing
- Investigate flow structures occurring during the transition of natural convection flows.
- Investigate modifications to the gas turbine design to alleviate the problems occurring during shutdown.

## 1.4 Thesis Outline

Chapter 2 Is a review of the literature. This chapter is to give an understanding of natural convection flows, what numerical models are available and what are the available validation cases.

Chapter 3 Looks at the gas turbine casing test rig, why a new rig is needed and the features of this rig. Followed by details of the rig geometry and measurements obtained. Finally an LES study of the casing rig is given, with details of the numerical set-up. The LES results are compared with the experimental data and together will be used as a validation dataset.

Chapter 4 Is a DNS study into a differentially heated tall cavity test case. This gives details of the numerical set-up and a comparison with the available experimental data. The key result is the prediction of turbulent heat flux terms that are not available in the experimental data, this will then form a high fidelity dataset for comparison with other simulations.

Chapter 5 Is an investigation into the use of different turbulent heat flux models in RANS simulations. Models from the literature are compared to determine if more advanced models can give improvements in wall heat transfer predictions. A modification is then developed to capture the effect of buoyancy on the turbulent heat flux. Finally the suitability of the models for use by engineers for natural convection simulations is discussed.

Chapter 6 Is a fundamental investigation into the formation of streamwise vortices that were found to occur in the casing rig. The investigation looks at how the vortices form as well as factors that can impact the vortex wavelength.

Chapter 7 Details a numerical investigation into modifications to the gas turbine design to try and alleviate the problems of casing distortion occurring during shutdown. The effectiveness of injecting and extracting flows from the casing are compared to a baseline to determine if the uniformity of the casing heat transfer has been improved.

Chapter 8 Gives a summary and the conclusions of the thesis, along with ideas for future research.

Appendix A Gives details of a study into the accuracy of different turbulence models and the effect of including buoyancy terms in a turbulence model.

Appendix B Gives details of the design of the inlet for the gas turbine casing rig.

Appendix C Gives details of the instrumentation used on the gas turbine casing rig.

# Chapter 2

## Literature Review

### 2.1 Natural Convection Flows

Natural convection flows occur in a wide range of engineering scenarios from small scales, such as electronic component cooling, to very large scales, such as in gas turbines or steam turbines during shutdown. In order to determine the state of a natural convection flow in each of these cases the Grashof number can be calculated. The Grashof number is an important non-dimensional parameter for natural convection flows and is the ratio of buoyancy forces to viscous forces, given by Equation.2.1. The Grashof number can be thought of as being analogous to the Reynolds number for forced convection flows, with the flow being laminar at low Grashof numbers before transitioning at a critical Grashof number.

$$Gr = \frac{g\beta(T - T_{\infty})x^3}{\nu^2} \quad (2.1)$$

For fluids such as air with a Prandtl number  $\sim 1$  the momentum and thermal boundary layers have similar thicknesses. For fluids with different Prandtl numbers such as oil  $Pr \sim 10^4$  and liquid metals  $Pr \sim 10^{-2}$  the thermal and momentum boundary layers will have different thicknesses, which in turn has an effect on the heat transfer. An important parameter for the heat transfer in natural convection is then given by the product of the Grashof number and Prandtl numbers, known as the Rayleigh number, given by Equation.2.2.



$$Ra = GrPr = \frac{g\beta(T - T_{\infty})x^3}{\nu\alpha} \quad (2.2)$$

To determine whether the heat transfer in a particular case is dominated by natural convection, forced convection or mixed convection the Richardson number  $Ri$  is used. This non-dimensional parameter is the ratio of natural convection to forced convection and can be given by Equation.2.3.

$$Ri = \frac{Gr}{Re^2} = \frac{g\beta(T - T_{\infty})x}{V^2} \quad (2.3)$$

Using criterion from Sparrow et al. [85] the heat transfer can then be classified by:

|                 |                    |
|-----------------|--------------------|
| $0 < Ri < 0.3$  | Forced Convection  |
| $0.3 < Ri < 16$ | Mixed Convection   |
| $16 < Ri$       | Natural Convection |

The convective heat flux from natural, mixed or forced convection can be determined using Newton's law of cooling,  $q = h\Delta T$ , where  $h$  is the heat transfer coefficient between the fluid and the surface. The convective heat transfer can be non-dimensionalised by the conductive heat transfer to give the Nusselt number, given by Equation.2.4.

$$Nu_x = \frac{hx}{\lambda} \quad (2.4)$$

## 2.2 Natural Convection Studies in Gas Turbines

There are only a limited number of studies in the literature looking at the natural convection flow and heat transfer in a gas turbine during shutdown. These studies tend to focus on smaller aero engines and look at rotor bow phenomenon, a distortion of the rotor due to non-uniform cooling. The rotor bow effect does not tend to occur in large industrial gas turbine as the rotor is barred, slowly rotated, during shutdown to ensure uniform cooling. Large industrial gas turbines also differ from aero engines during shutdown in that the casing of the industrial gas turbine is very thick and cools slowly, this provides the heat to drive natural convection flows. In an aero engine the casing is a relatively thin component

and cools relatively quickly, it is the thicker compressor and turbine discs inside of the casing that cool slowly and drive the natural convection flows.

An numerical investigation by Smith and Neely [82] and later an experimental investigation by Smith et al. [81] looked at the rotor bow phenomenon in geometry representative of a helicopter engine. The rotor used was 220 mm in diameter and was heated to 600 K. Despite the relatively high temperature the small diameter gives a Rayleigh number at the start of the experiment of  $\sim 10^7$ , putting the flow in the laminar regime. Another experimental investigation into rotor bow was conducted by Fahy et al. [20]. This rig used a heated cylinder of 326 mm diameter and operated at Grashof numbers  $\sim 10^8$ . This is likely still in the laminar regime and below the Grashof numbers of  $10^{10}$  found in large industrial gas turbines. The flowfield in these rotor bow cases is also quite different to that found in a large industrial gas turbine. The experiments featured a heated cylinder to represent the compressor and turbine discs, this creates a thermal boundary layer and a buoyant plume rising from the top of the cylinder. In contrast an industrial gas turbine features a hot outer casing with natural convection flows occurring inside of the casing where stratified flow can occur.

An investigation into the shutdown of a micro gas turbine was conducted by Tisarev et al [93], performing both a conjugate CFD simulation and an experiment. The engine was very small, with an outer casing diameter of 113 mm, which would likely give a low Rayleigh number laminar flowfield. This would therefore give a flowfield and heat transfer that are very different from that occurring in an industrial gas turbine.

A transient thermal analysis of an industrial gas turbine was investigated by Selvam et al. [76] and Reddy et al. [69]. The model used was a 2D axi-symmetric model of a large industrial gas turbine. The authors state that the natural convection flow inside of the engine is complex and plays an important role in the engine cooling, but difficult to capture with a 2D model and therefore used circumferentially averaged gas temperatures.

To the author's knowledge there are no investigations in the literature investigating the flowfield of a large industrial gas turbine during shutdown. A similar test case is that of a large steam turbine during shutdown. The construction of large steam turbines is similar to industrial gas turbines and feature a thick outer casing which cools slowly during shutdown.

An investigation into a large steam turbine during shutdown was conducted by Born et al. [8]. It was stated by the authors that it is not possible to conduct a transient conjugate simulation on the steam turbine due to the large difference in time-scales between the fluid and solid. An alternative approach was therefore taken where the fluid was modelled as a solid and the heat transfer caused by convection of the fluid is modelled as an increase in the conductivity of the fluid. The Nusselt numbers used to calculate the increase in conductivity came from correlations for natural convection flow in annular gaps and flat plates. This technique allowed the trends of the casing temperature during shutdown to be predicted but the use of correlations means that the effect of different engine designs on the shutdown temperatures cannot easily be predicted.

It has been found that there is very little data in the literature looking at gas turbines and specifically large industrial gas turbines during shutdown. There are no cases looking in detail at the flowfield inside of an engine and methods to accurately predict this flowfield.

## **2.3 Numerical Methods for Natural Convection**

To calculate natural convection heat transfer in gas turbines several options are available, the simplest of which is to use experimental correlations. The cost of a hand calculation or spreadsheet calculation using correlations is very low and can be used as a first analysis step. However, experimental correlations are usually only available for simple geometries and have limited validity when used outside of the simple geometries used to obtain them. An alternative method for calculating heat transfer is the use of Computational Fluid Dynamics (CFD). CFD simulations can cope with the more complicated geometries found in engineering situations and can predict changes in heat transfer for different engineering designs. Several numerical methods are available to predict natural convection flows, ranging from the very computationally expensive Direct Numerical Simulation (DNS), through to Large Eddy Simulations (LES) and Reynolds Averaged Navier Stokes (RANS) simulations.

### 2.3.1 Direct Numerical Simulation

In a Direct Numerical Simulation all of the turbulent scales of motion are resolved, from the largest scales set by the flow geometry to the smallest scales set by viscous dissipation [68]. In order to resolve down to the smallest turbulent length scales a fine computational mesh is required, giving a large computational cost. The mesh requirement scales with  $Re^{9/4}$  [37], for natural convection if the Reynolds number is defined using the buoyant velocity then  $Re = \sqrt{Gr}$  and the mesh scales as  $Gr^{9/8}$ .

Studies using DNS have been conducted for several natural convection test cases. The heated vertical plate was numerically investigated by Phillips [66] for Grashof numbers up to  $1.8 \times 10^5$ . The vertical slot was later investigated by Boudjemadi et al. [9] for Rayleigh numbers up to  $5.4 \times 10^5$  and by Versteegh & Nieuwstadt [104] for Rayleigh numbers up to  $5 \times 10^6$ . It can be seen that these Rayleigh numbers are all significantly lower than the Rayleigh number found in a gas turbine during shutdown. Simulations at a higher Rayleigh number would now be possible due to increased computational power, as the Versteegh and Nieuwstadt work was published in 1998 and used 9 million grid points.

A second geometry used for DNS of natural convection flows is that of a differentially heated cavity. A cavity with height:width aspect ratio of 4:1 was investigated by Trias et al. [95, 96]. Rayleigh numbers from  $6.4 \times 10^8$  up to  $10^{11}$  were investigated. At these high Rayleigh numbers the flow achieved transition to turbulence in regions close to the top and bottom of the cavity, with the laminar central region shrinking with increasing Rayleigh number. The cavity was also investigated by Pitz & Chew [67], for a Rayleigh number of  $2.0 \times 10^9$ . The DNS solver used spectral element code and compared the results to a simulation using ANSYS Fluent and the Trias et al. data. Both simulations predicted similar values for the mid-height Nusselt number, but the Fluent simulation was found to predict lower values for the turbulent statistics. A similar 5:1 cavity was used in an investigation by Trias et al. [94], up to Rayleigh numbers of  $4.5 \times 10^{10}$ . Later the 5:1 cavity was used by Kizildag et al. [40] for an investigation using water as the working fluid up to a Rayleigh number of  $3 \times 10^{11}$ .

These simulations show that DNS at the Rayleigh numbers found in industrial gas tur-

bines during shutdown is currently possible, at least for very simple geometries. In industry the cost of DNS simulations makes them too expensive for engineering analysis and are currently only used as a research tool.

### 2.3.2 Large Eddy Simulation

In a turbulent flow there is a large range of turbulent scales and resolving all of these scales has a significant computational cost. Large Eddy Simulations work on the assumption that it is the large turbulent scales that determine the flow behaviour, these large eddies are anisotropic with their behaviour dictated by the geometry. The smaller eddies are more isotropic and follow a more universal behaviour, allowing them to be modelled rather than resolved [103]. The mesh requirement for a simulation is reduced as only  $\sim 80\%$  of the turbulent kinetic energy needs to be resolved [68]. The computational expense of LES is therefore reduced in comparison with DNS, the computational cost now only rising as  $Re^{9/5}$  [10] or  $Gr^{9/10}$ . LES are currently used by engineers for analysis of some specific parts of a gas turbine, such as the combustor [57], but generally they are not used due to the computational cost.

Large eddy simulations have been performed for several natural convection cases. The simple case of the heated vertical plate was investigated recently by Ojofeitimi & Hattori [61], up to Rayleigh number of  $3 \times 10^{11}$ , where good agreement was found with experimental data. A similar case was investigated by Kogawa et al. [42] using water as the working fluid for Rayleigh numbers up to  $4.8 \times 10^{11}$ .

High Rayleigh number investigations using LES have also been conducted for cavity flows. A 5:1 aspect ratio cavity was investigated by Lau et al. [46] for a Rayleigh number of  $4.5 \times 10^{10}$ . The differentially heated square cavity was also investigated at a Rayleigh number of  $1.58 \times 10^9$  by Peng & Davidson [64] and also by Salinas-Vazquez et al. [73].

The vertical wall and cavity cases are both simple geometries, but there also test cases using geometries more representative of gas turbine cavities. An investigation using LES was performed on a single disc cavity by Sun et al. [88], simulations up to a Grashof number of  $7.41 \times 10^8$  were conducted. The authors found much better agreement with experimental data compared to RANS simulations but noted that the LES were compu-

tationally demanding. It is likely for this reason that investigations using more complex geometries have tended to use a hybrid RANS-LES method. Such as the 14-stage compressor cavity by Ansari et al. [2], the high pressure turbine cavity by Smirnov et al. [80] and the 10 stage compressor cavity by Tateishi et al. [89].

Whilst a large eddy simulation will resolve the bulk of turbulent eddies, a model is required to capture the effects of the smallest eddies that are not resolved. A sub-grid scale model is required in the momentum equation and similarly a sub-grid heat flux model is required in the energy equation. An investigation was conducted by Ghaisas et al. [23] evaluating the effect of different sub-grid scale models on the prediction of turbulent natural convection in the 4:1 heated cavity. Four different models were tested, dynamic Smagorinsky, dynamic Vreman, Sigma and a Stretched Vortex model. The dynamic Smagorinsky, Stretched Vortex and Sigma models were found to be in good agreement with DNS data with the Vreman model being less accurate at predicting turbulent statistics. All four models seemed to perform well when predicting mean flow quantities. A similar investigation was conducted by Lau et al. [45] looking at the turbulent flow in an asymmetrically-heated vertical parallel-plate. The Smagorinsky model with constant coefficients was compared with two different formulations of a dynamic Smagorinsky model along with the Vreman model with constant coefficients. It was found that the constant Smagorinsky model was overly dissipative and delayed transition, the dynamic Smagorinsky models tended to over predict fluctuations whilst the Vreman model gave good agreement for the temperature and velocity fields. A third study was conducted by Kumar & Dewan [44] investigating the effect sub-grid scale models on the flow in a square buoyant cavity. Simulations were run using the models, dynamic Smagorinsky, Vreman, one equation eddy viscosity and buoyancy modified one equation eddy viscosity. It was found that the dynamic Smagorinsky model outperformed the Vreman model in predicting the mean flow and statistics but the one equation model outperformed both of these models. The effect of the buoyancy modification to the one equation model was found to not give a significant improvement in predictions. This would fit with the theory of Bolgiano [7] that buoyancy only affects the large eddies and not the smaller eddies in the inertial subrange, which are those being modelled by the sub-grid scale model.

The effect of different sub-grid scale heat flux models on the predicted flowfield between two heated vertical plates was investigated by Peng & Davidson [65], with a linear model being compared to a proposed non-linear model. The results from the two models were found to both be similar, the authors claimed that the differences may be more significant at Rayleigh numbers higher than the  $10^5$  tested. A new sub-grid heat flux model was proposed by Morar [58] which included a buoyant production term. The results from this model were again found to be similar to models without the buoyancy term which also used dynamically calculated coefficients.

These results show that current sub-grid scale modelling is adequate for LES of natural convection flows. The small effect of different sub-grid scales on the solution can be understood by looking at a hierarchy of importance for LES, as described by Tyacke & Tucker [101]. The problem definition and boundary conditions dominate in terms of importance in the simulation with the sub-grid modelling having low importance.

### **2.3.3 Reynolds Averaged Navier Stokes**

As engineers desire simulations with a relatively low computational cost, the bulk of calculations performed in the design and analysis of gas turbines are RANS simulations. Simulations can be performed on an engineering workstation in a few hours as opposed to a several days as may be the case with LES simulations. It is also believed that the trend of engineers using RANS simulations over more expensive large eddy simulations will continue [79, 17].

There are a large number of cases in the literature of RANS simulations being used for natural convection flows. A review of natural convection simulations in enclosures was conducted by Choi & Kim [13]. They showed that it was possible in certain cases for RANS simulations to give results close to those from a large eddy simulation.

## **2.4 RANS Equations**

To obtain the equations used in RANS simulations the Reynolds averaging process is performed on the Navier Stokes equations. The Reynolds averaging process involves decom-

posing instantaneous flow variables into a mean and fluctuating components which are then time averaged [70]. The temperature is decomposed as  $\tilde{T} = \bar{T} + T'$ , and the velocity as  $\tilde{u}_i = \bar{U}_i + u'_i$ . The Navier Stokes energy equation is given by Equation.2.5, after the Reynolds averaging process has been performed the Reynolds averaged energy equation then given by Equation.2.6.

$$\frac{\partial \tilde{T}}{\partial t} + \frac{\partial(\tilde{u}_i \tilde{T})}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \alpha \frac{\partial \tilde{T}}{\partial x_i} \right) + S_T \quad (2.5)$$

$$\frac{\partial \bar{T}}{\partial t} + \frac{\partial(\bar{U}_i \bar{T} + \overline{u'_i T'})}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \alpha \frac{\partial \bar{T}}{\partial x_i} \right) + \bar{S}_T \quad (2.6)$$

The Reynolds Averaged Navier Stokes energy equation now contains the turbulent heat flux term  $(\overline{u'_i T'})$ , a product of the temperature and velocity fluctuations. There are no other equations with which to close the turbulent heat flux term, therefore a model must be created to close it.

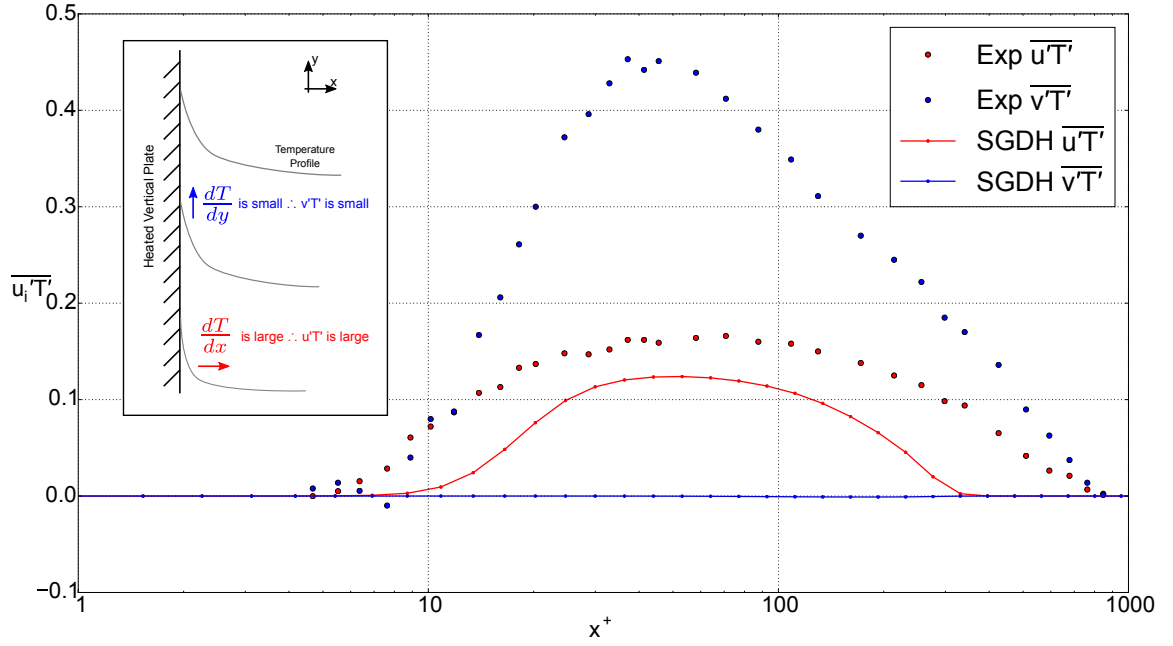
The turbulent heat flux term  $(\overline{u'_i T'})$  is analogous to the Reynolds stress term  $(\overline{u'_i u'_j})$  that appears in the Navier Stokes momentum equation after the Reynolds averaging process has been performed. Similarly the Reynolds stress term needs to be closed using a turbulence model, the choice of which can have a significant impact on the the solution [101]. An investigation into the use of different turbulence models for natural convection can be found in Appendix.A.

## 2.5 Turbulent Heat Flux Modelling

### 2.5.1 Simple Gradient Diffusion Hypothesis

There are a range of models in the literature that have been developed in order to close the turbulent heat flux term. The simplest of which is the Simple Gradient Diffusion Hypothesis (SGDH). The SGDH model, given in Equation.2.7, assumes that the turbulent heat flux is proportional to the mean temperature gradient. This is then scaled by the turbulent viscosity  $(\nu_t)$  and the turbulent Prandtl number  $(Pr_t)$ . The turbulent Prandtl number is often a constant in the region 0.7-0.9 [51] and the turbulent viscosity is obtained from the





**Figure 2.1:** Experimental [98] and modelled turbulent heat fluxes on vertical wall case

turbulence model. The SGDH is the model that is currently used in the commercial solver ANSYS Fluent.

$$\overline{u_i' T'} = -\frac{\nu_t}{Pr_t} \frac{\partial \overline{T}}{\partial x_i} \quad (2.7)$$

It can be shown that the SGDH model is not adequate for natural convection flows by considering the case of natural convection flow next to a heated vertical wall. Turbulent heat fluxes measured in a vertical wall case [98] are compared to values from a RANS simulation using the SGDH model in Fig.2.1. Whilst the SGDH model predicts well the turbulent heat fluxes normal to the heated wall, it under-predicts the turbulent heat flux parallel to the wall. This is due to the SGDH model assuming that the turbulent heat flux is proportional to the mean temperature gradient. Normal to the wall there is a significant temperature gradient, giving good predictions of turbulent heat flux. Parallel to the wall there is a very low temperature gradient, hence under-predicting the turbulent heat flux. This shows that for natural convection flows the assumption that the turbulent heat flux is proportional to the mean temperature gradient is not valid. Therefore the SGDH turbulent heat flux model will likely not give accurate predictions for natural convection cases.

## 2.5.2 Generalised Gradient Diffusion Hypothesis

An alternative method of modelling the turbulent heat flux is to use the Generalised Gradient Diffusion Hypothesis (GGDH) developed by Daly & Harlow [15], and given by Equation.2.8.

$$\overline{u'_i T'} = -C \frac{k}{\epsilon} \overline{u'_i u'_j} \frac{\partial \overline{T}}{\partial x_j} \quad (2.8)$$

The GGDH model has the advantage over the SGDH model in that it allows turbulent heat fluxes normal to the mean temperature gradient. The GGDH model is still only a two equation model, requiring a  $k - \epsilon$  or  $k - \omega$  turbulence model in order to obtain a turbulent time-scale  $k/\epsilon$ .

A value of 0.15 was used for the constant  $C$ , which is equal to  $3/2(c_\mu/Pr_t)$  [30], where the empirical constant  $c_\mu = 0.09$ . This constant was set so the GGDH model would give the same result as the SGDH model in the case of a horizontal shear flow, where the mean temperature and velocity gradients are vertical.

The GGDH was used successfully to investigate the flow in 2D and 3D heated cavities by Ince and Launder [30, 31]. It was noted however by Choi & Kim [13] that the GGDH does not perform well for flows with strong thermal stratification.

## 2.5.3 Algebraic Flux Model

A turbulent heat flux model with a further increase in complexity from the GGDH model is the Algebraic Flux Model (AFM), developed by Kenjeres & Hanjalic [38] and given by Equation.2.9.

$$\overline{u'_i T'} = -C \frac{k}{\epsilon} \left( \overline{u'_i u'_j} \frac{\partial \overline{T}}{\partial x_j} + \xi \overline{u'_j T'} \frac{\partial \overline{U}_i}{\partial x_j} + \eta \beta g_i \overline{T'^2} \right) \quad (2.9)$$

The model consists of three terms, the first  $\left( \overline{u'_i u'_j} \frac{\partial \overline{T}}{\partial x_j} \right)$  is the same term as is used in the GGDH model. The second term  $\left( \xi \overline{u'_j T'} \frac{\partial \overline{U}_i}{\partial x_j} \right)$  accounts for the interaction of the turbulent heat flux with the mean flow deformation. The third term  $\left( \eta \beta g_i \overline{T'^2} \right)$  accounts for the effect of the temperature variance interacting with the gravitational field. The constant  $C$  also used a value of 0.15 and the constants  $\xi$  and  $\eta$  were both set to 0.6.

The AFM model is the only one of the three to have an explicit buoyancy term. This buoyancy term does however require an equation is solved for the temperature variance, this makes the AFM model a three equation model. The AFM model uses the assumption that the dissipation of the temperature variance  $\epsilon_{T'}$  is proportional to the dissipation of turbulent kinetic energy  $\epsilon$ . This assumption avoids the need for a fourth equation for  $\epsilon_{T'}$  to be solved.

## 2.5.4 Differential Flux Model

Other more complex models for the turbulent heat flux are available, such as the Differential Flux Model (DFM) proposed by Shin et al. [77]. This model was tested by Choi and Kim [13] and was found to perform very similarly to the AFM. However the DFM requires the solution of extra equations so the AFM was recommended over the DFM.

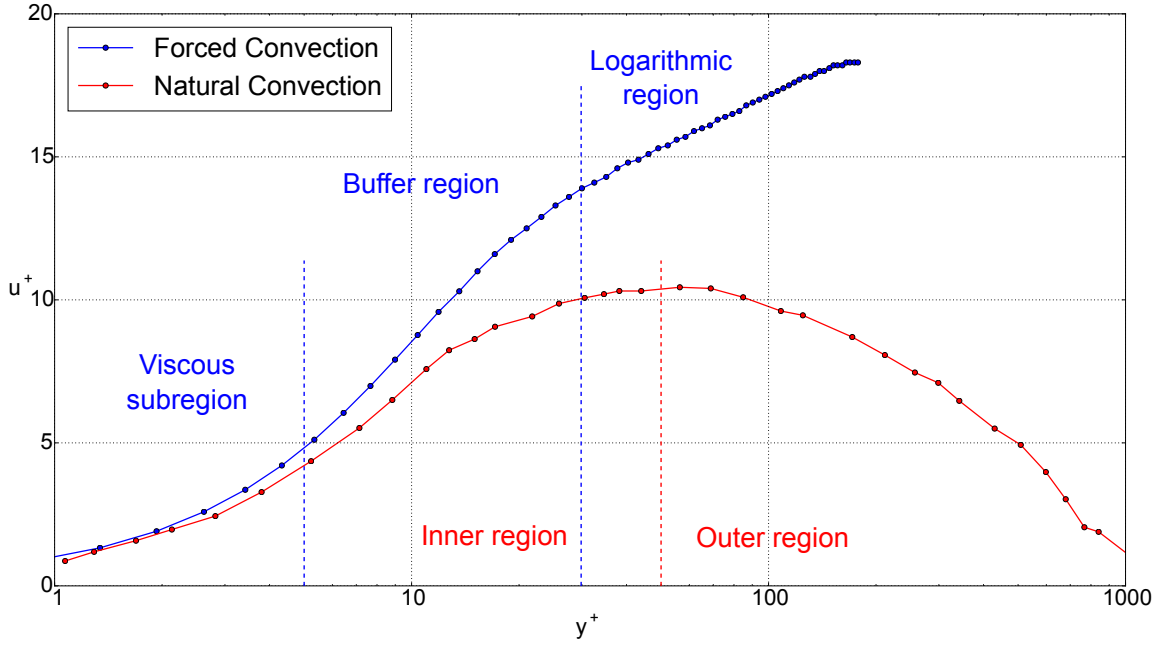
## 2.6 Natural Convection Boundary Layer

### 2.6.1 Velocity Profile

The velocity profile in a forced convection turbulent boundary layer starts with a viscous subregion close to the wall  $y^+ < 5$ , where  $u^+ = y^+$ , followed by a buffer layer  $5 < y^+ < 30$  then a logarithmic layer  $y^+ > 30$ , where  $y^+ = yU_\tau/\nu$  and  $u^+ = u/U_\tau$  [68].

This law of the wall does not hold for a natural convection boundary layer, with buoyancy forces near the wall having a significant effect on the velocity profile. The difference between the natural and forced convection velocity boundary layers can be seen in Fig.2.2. The forced convection profile is from Kim et al. [39] at  $Re_\tau = 180$ , the natural convection profile is from Tsuji & Nagano [97] at  $Gr = 1.8 \times 10^{11}$ . In a natural convection wall boundary layer the velocity starts at zero at the wall then, reaches a peak at  $y^+ \approx 30 - 50$  [97] before reducing down again. It was also found that the peak velocity increased and moved further from the wall with increasing Grashof number. It was found that the forced and natural convection velocity profiles differed even close to the wall, with  $u^+ \neq y^+$  in the natural convection case.

As the profile of a natural convection boundary layer is very different from that of



**Figure 2.2:** Forced and natural convection boundary layer velocity profiles, data from Kim et al. [39] and Tsuji and Nagano [97]

a forced convection boundary layer, wall functions that have been developed for forced convection are not valid for use in natural convection simulations. This means that unless specific natural convection wall functions are implemented, such as those by Yuan et al. [106], the simulations must be wall resolved.

## 2.6.2 Temperature Profile

Studies investigating the temperature profile of a natural convection turbulent boundary layer have been conducted by George & Capp [35] and Henkes & Hoogendoorn [27]. George & Capp split the natural convection boundary layer into an inner region, from the wall up to the velocity peak and an outer region, shown in Fig.2.3. The inner region was then further separated into a conductive sublayer, which is a constant heat flux region where the temperature profile is linear. Followed by the buoyant sublayer, where turbulent heat transfer and buoyancy are important and the temperature profile is inversely proportional to the cube-root of the distance from the wall. Length scales for the inner and outer regions were defined as  $\eta_t$  and  $\delta_T$  respectively.

$$\eta_t = \left( \frac{\alpha^2}{g\beta(T_{wall} - T_\infty)} \right)^{1/3} \quad (2.10)$$

$$\delta_T = \int_0^\infty \frac{(T - T_\infty)}{(T_{wall} - T_\infty)} dx \quad (2.11)$$

The inner region length scale was modified by Siebers et al. [78] to give  $\eta'_t$ , this modification was to account for large variations in temperate between a heated wall and the free stream.

$$\eta'_t = \eta_t \left( \frac{\lambda_{wall}}{\lambda_\infty} \right) \left( \frac{T_{wall}}{T_\infty} \right)^{0.14} \quad (2.12)$$

Correlations were then given by Siebers et al. [78] for the temperature profiles in the inner and outer regions.

Inner region – viscous sublayer

$$\frac{T_{wall} - T}{T_{wall} - T_\infty} = 0.127 \frac{x}{\eta'_t} \quad (2.13)$$

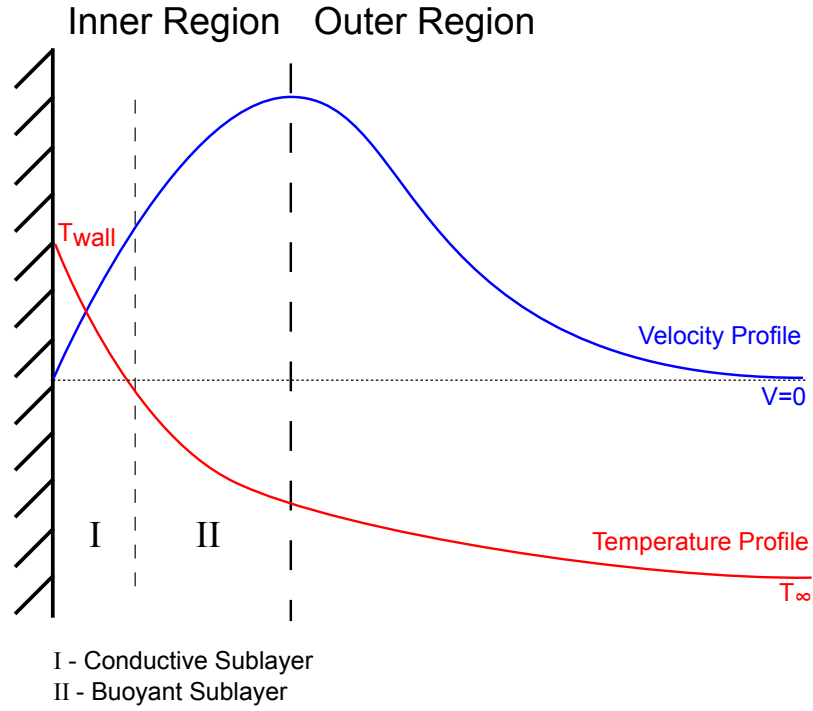
Inner region – buoyant sublayer

$$\frac{T_{wall} - T}{T_{wall} - T_\infty} = 1.36 - 1.42 \left( \frac{x}{\eta'_t} \right)^{-1/3} \quad (2.14)$$

Outer region

$$\frac{T_{wall} - T}{T_{wall} - T_\infty} = 0.08 \ln \left( \frac{x}{\delta_T} \right) + 0.81 \quad (2.15)$$

The temperature profile from a natural convention boundary layer is different from that of a forced convection flow. In a forced convection case Reynolds Analogy is used to assume that the temperature and velocity profile have similar forms. The temperature wall functions of Kader [36] give a temperature distribution of  $T^+ = Pr y^+$  close to a heated wall, analogous to the viscous sublayer. With the temperature having a logarithmic profile



**Figure 2.3:** Schematic of turbulent natural convection boundary layer, modified from [35]

from  $y^+ \approx 30$  for cases where  $Pr \sim 1$ . Where the non-dimensional temperature is given as  $T^+ = (T_{wall} - T) \rho c_p U_\tau / \dot{q}_{wall}$ .

## 2.7 Validation Test Cases

A number of natural convection test cases are available in the literature, a test case for validating simulations of a gas turbine during shutdown should ideally meet several requirements:

- The rig would operate at the same Rayleigh number ( $10^{10}$ ) as a real engine.
- The working fluid for the rig would be air to match the Prandtl number in a real engine.
- The rig would have a geometry that is representative of an industrial gas turbine.

It was found that there were no test cases that completely met all of the requirements for a validation data set. There was a gap in the literature for a test rig that had geometry

that was representative of a gas turbine and operated at the same Rayleigh number as a gas turbine during shutdown.

### 2.7.1 Measurement Challenges

The choice of validation test case also depends on the range of measurements available for the case. For validation of the general natural convection flowfield, measurements of temperature and velocity are required at several measurement locations. However natural convection flows pose challenges for obtaining these measurements.

Measurement of temperature profiles can be obtained by traversing a thermocouple or thermistor through an experimental rig. Depending on the flow it may take several minutes of measurement time to obtain an appropriate average value. If the wall heat flux is required then the temperature must be measured with a fine spatial resolution close to the wall, in order to capture the linear conductive sublayer of the temperature profile.

The measurement of velocity in a natural convection test rig can be difficult, due to the low velocities involved and raised temperatures. If a hot wire measurement system is used, the maximum fluid temperature that can be measured is  $\approx 150^\circ\text{C}$ , as the hot wire needs to be  $\approx 200^\circ\text{C}$  hotter than the flow to operate and the temperature limit of the tungsten hot wire is then reached [16]. The hot wire also needs to be temperature compensated as fluctuations in the temperature of the flow will affect the hot wire reading. The buoyancy forces generated by a hot wire also make low velocities difficult to resolve, this occurs when  $Re < 2Gr^{1/3}$  [16], or at a velocity  $\lesssim 0.1$  m/s for a  $200^\circ\text{C}$  overheat.

An alternative technique to using a hot wire is the use of laser based velocity measurement, such as Laser Doppler Anemometry (LDA). This removes the temperature limits but requires optical access to the rig and requires the flow to be seeded with particles. Seeding is made difficult due to the low velocities, meaning that particles may not follow the flow. Seeding can be achieved by vaporising oil or by burning sticks to create smoke, but care is needed to make sure the hot seeding parties don't create their own buoyancy forces. Due to these difficulties it is common to find that velocities were not measured in natural convection test cases.

If turbulent statistics are required for validation, then temperature and velocity fluctua-

tions are required as well as the mean values. Measuring fluctuations in the flow requires a relatively high sample rate compared to that needed for measuring averages. Sample rates in the  $\sim 100$  kHz can be achieved by modern data acquisition systems [60], but the temperature and velocity probes also need to have a sufficiently fast response time. This additional complexity means that turbulent statistics for validation are less common in experimental datasets than average values.

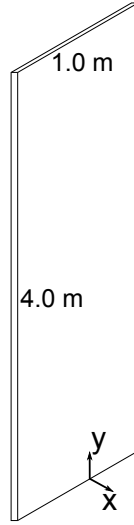
The turbulent heat flux term  $(\overline{u'_i T'})$  is the average of the product of a velocity and temperature fluctuation. For this term to be measured experimentally the velocity and temperature must be recorded simultaneously at the same location at a sufficiently fast sample rate. This creates challenges in the measurement probe design, as the sensors must have a sufficiently fast response time and be located physically close together. The probe design is made even more challenging if more than one component of the turbulent heat flux needs to be measured simultaneously. Consequently experimentally measured values of the turbulent heat fluxes are uncommon in the literature.

## 2.7.2 Heated Vertical Plate

The test case of natural convection over a heated vertical plate has been studied experimentally by several groups including Tsuji & Nagano [97] and Siebers et al. [78], giving a range of experimental measurements. The experimental rigs used by these authors were relatively large, which allowed for high Rayleigh number flows to develop. The Tsuji & Nagano [97] test rig is 4 m tall, shown schematically in Fig.2.4, allowing flows up to Grashof numbers of  $10^{12}$ . The test rig used by Siebers et al. [78] was 3.02 m tall and used high wall temperatures of up to  $520^\circ\text{C}$ , therefore also giving Grashof numbers of up to  $10^{12}$ . Both test rigs used air as the working fluid, giving Rayleigh numbers that could match those in an industrial gas turbine.

A key output of numerical simulations of natural convection cases is the wall heat transfer. Profiles of Nusselt number against Rayleigh number are available from both Tsuji & Nagano [97] and Siebers et al. [78]. The wall heat transfer results agreed well in the two test cases, with Tsuji and Nagano giving the correlations  $Nu_y = 0.387 (Gr_y Pr)^{1/4}$  in the laminar regime and  $Nu_y = 0.120 (Gr_y Pr)^{1/3}$  in the turbulent regime. The results from



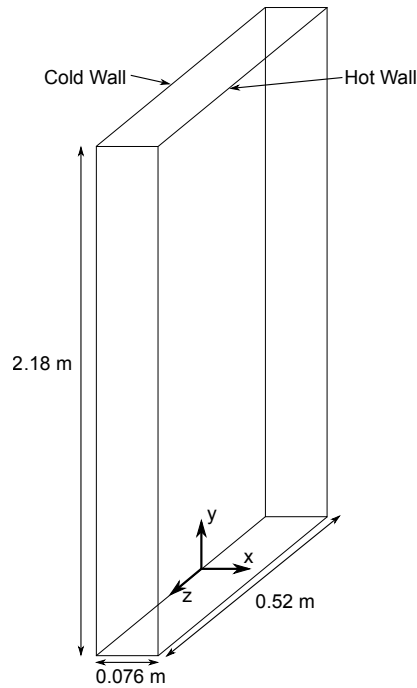


**Figure 2.4:** Schematic of Vertical Wall test case

Siebers et al. were  $Nu_y = 0.404 (Gr_y)^{1/4}$  and  $Nu_y = 0.096 (Gr_y)^{1/3}$  for the laminar and turbulent regimes respectively. The transition locations were also similar in both cases at  $Gr_y Pr \approx 10^9$  in the Tsuji & Nagano case and  $Gr_y \approx 10^9$  in the Siebers et al. case.

The Siebers results showed that the wall heat transfer for cases with a wide range of wall temperatures, from 60°C up to 520°C all collapsed well onto the two correlations. This shows that it is not necessary for a test rig to match wall temperatures as long as the Rayleigh number is matched.

The heated vertical plate as a test case also has experimentally measured turbulent heat fluxes, measured by Tsuji & Nagano [99]. Measurements were taken of both the wall normal  $(\overline{u'T'})$  and parallel  $(\overline{v'T'})$  turbulent heat fluxes. This was achieved using a probe containing two hot wires arranged in an “x” configuration to measure the two velocity components and a cold wire to measure the temperature. The cold wire was located 2.7 mm upstream of the hot wires and used for temperature compensation, with the signal being delayed to account for the spatial difference between the hot and cold wires. The data was sampled at 400 Hz for 1-5 minutes, allowing the turbulent heat fluxes as well as average velocities and temperatures to be obtained.



**Figure 2.5:** Tall Cavity Apparatus (modified from [6])

### 2.7.3 Tall Cavity

Another literature test case is that of the differentially heated tall cavity. The differentially heated tall cavity was investigated by Dafa'Alla & Betts [14] and later by Betts & Bokhari [6]. The test rig consisted of a rectangular cavity with dimensions 2.18 m high by 0.076 m wide by 0.52 m deep, shown schematically in Fig.2.5. One of the side-walls of the cavity was heated and the other side-wall was cooled to create a natural convection cavity flow. The authors paid close attention to making the rig boundary conditions simple to implement using CFD. The top and bottom walls were made of rubber to try and match the effective thermal conductivity in the cavity, this gave a very linear temperature variation along these walls.

The rig was run with two wall temperature differences of 19.6 °C and 39.9 °C giving Rayleigh numbers based on the cavity width of  $0.86 \times 10^6$  and  $1.43 \times 10^6$  respectively. If the cavity height of 2.18 m is used to calculate the Rayleigh number, this gives values  $10^{10}$ , the same order as a gas turbine during shutdown.

The tall cavity also has a large measurement dataset available. Profiles across the cavity

width of temperature and velocity are available at nine different cavity heights. Measurements were taken using a thermocouple and a Laser Doppler Anemometry (LDA) system. Both time averaged and rms temperature and velocity fluctuations were acquired. The availability of rms statistics allows for comparisons with turbulent fluctuations that may be modelled or resolved by numerical simulations. The correlation between the velocity and temperature fluctuations was not recorded so direct measurements of the turbulent heat fluxes are not available.

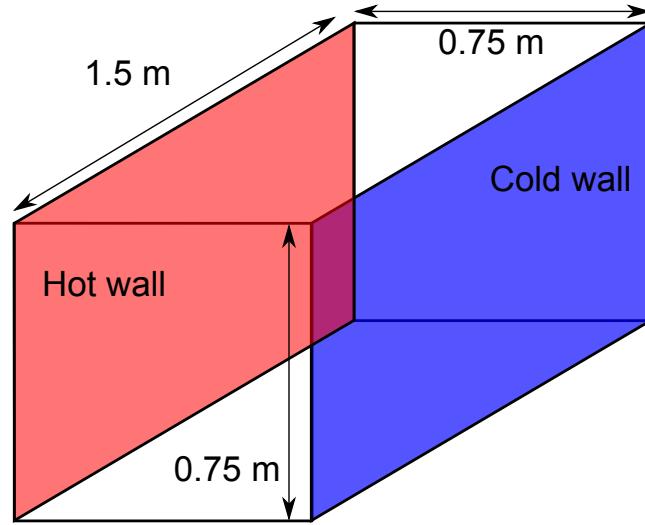
#### **2.7.4 Heated Cylinder**

A further literature test case is that of a heated concentric cylinder. The seminal investigation for this case is by Kuehn & Goldstein [43]. The experimental rig consisted of a 92.5 mm outer cylinder with a heated 35.6 mm inner cylinder and operated at Rayleigh numbers from  $2.11 \times 10^4$  to  $9.76 \times 10^5$ . A similar investigation was conducted by Sun & Zhang [87] using a cylinder with an outer diameter of 79.91 mm and different diameter inner cylinders, this gave Grashof numbers of  $4.6 \times 10^2$  to  $3.8 \times 10^4$ . A further test case was investigated by Van de Sande & Hamer [102] which operated at a higher Rayleigh numbers of  $10^8$  and used an outer cylinder of 299 mm diameter and inner cylinders of 50 mm or 129 mm diameter.

Whilst the case of a heated concentric cylinder may seem to be very similar to an industrial gas turbine annulus there are several key differences. The test cases use cylinders with small diameters and operate at Rayleigh numbers below those found in an industrial gas turbine. A concentric cylinder case that operated at Rayleigh numbers of up to  $5 \times 10^{11}$  was investigated by McLeod & Bishop [54], this case still used small diameter test rig with an outer cylinder diameter of 203 mm but operated at cryogenic temperatures. The other difference between the concentric cylinder case and an industrial gas turbine is that the inner cylinder is heated in the literature cases whereas it is the outer casing of a gas turbine that is hot relative to the inner components.

#### **2.7.5 Square Cavity**

Another natural convection cavity flow case is that of a differentially heated square cavity. This case was investigated by Tian & Karayiannis [91, 92] and later by Ampofo &



**Figure 2.6:** Schematic of square cavity, modified from [1]

Karayiannis [1]. The cavity was  $0.75 \times 0.75 \times 1.5$  m in size and had isothermal side-walls at  $50^\circ\text{C}$  and  $100^\circ\text{C}$ , shown schematically in Fig.2.6. This gave a Rayleigh number of  $1.58 \times 10^9$ . The cavity flowfield consisted of a thin boundary layer flow close to the hot and cold side-walls as well as along the insulated top and bottom walls, with the bulk of the cavity consisting of a low velocity stratified region.

Measurements of temperature and velocity were measured in the cavity, the temperature was measured using a  $25.4\ \mu\text{m}$  thermocouple and the velocity using a LDA system. Mean and rms values of temperature and velocity as well as turbulent heat flux measurements are available, therefore providing a large validation dataset.

## 2.8 Conclusions

Engineers desire to be able to predict natural convection heat transfer in high Rayleigh number cases, such as industrial gas turbines. Due to the limitations of using correlations a numerical method is required to predict the flow. As the numerical simulations need to be computationally low cost the simulations will likely be RANS simulations.

When performing RANS simulations a model is required to close the turbulent heat flux term  $(\overline{u'_i T'})$ . The SGDh turbulent heat flux model is commonly used by CFD solvers, but it was found that the assumptions used in this model are not valid for natural convection.

Alternative turbulent heat flux models were found in the literature, including the GGDH and AFM models, which could give improved predictions of natural convection heat transfer.

Any new models will need to be validated against experimental data. Test cases using simple geometries were found in the literature that operated at high Rayleigh numbers, including the heated vertical plate as well as natural convection cavity flows. It was found that there is a gap in the literature for a case that operates at the same Rayleigh number as a gas turbine during shutdown and has engine representative geometry.

# Chapter 3

## Gas Turbine Casing Rig

*A natural convection test rig was constructed by MHI which represents a section of a large gas turbine casing. The rig operated at real engine Rayleigh number and was able to provide data that could not be obtained from a real engine. The rig would provide a unique validation dataset.*

*In order to better understand the flowfield inside of the casing rig, an LES study was performed. The simulation allowed temperature and velocity contours to be visualised that could be obtained from the experimental data.*

*The results from the LES study and the experimental data showed the formation of streamwise vortices in the rig. The streamwise vortices have a large effect on the outer wall heat transfer, causing streamwise streaks of high and low heat transfer. These vortices are also the transition mechanism for this flow. The results of the LES simulation were compared to the experimental temperature profiles. It was found that the LES under-predicted the vortices and effect on the temperature profile. Downstream in rig, where the effect of the vortices is reduced, the LES was found to give good predictions of wall heat transfer with errors less than 10%.*

### 3.1 Introduction

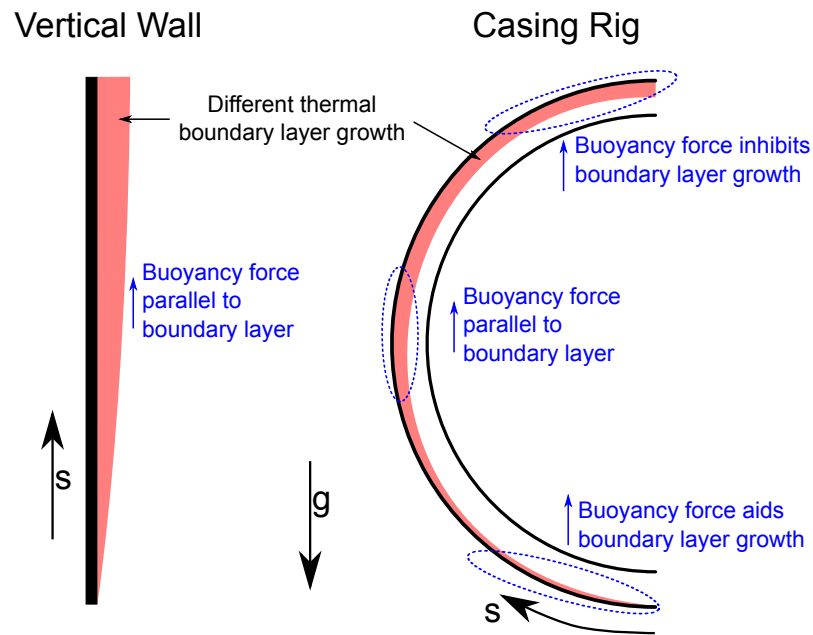
In order to validate any new CFD methods, experimental data is required. It would be desirable to obtain validation data directly from a real gas turbine during shutdown, but it is very difficult to obtain temperature data in a real engine. Validation data can however

be obtained from a test rig with the same flow conditions as a real engine. An important parameter for a natural convection test rig to match in order to match the flow conditions is the Rayleigh Number,  $Ra = \frac{g\beta\Delta TR^3}{\nu\alpha}$ . The Rayleigh Number in a large industrial gas turbine during shutdown is of the order  $10^{10}$ , at this high Rayleigh Number the natural convection flows will transition to turbulence. It is important for a test rig to also operate at this high Rayleigh Number and to have engine representative geometry to give an engine representative flowfield. The geometry would have a curved heated wall to represent the hot outer casing of a gas turbine. The fluid would be inside of this wall and would then represent the flow inside of the gas turbine casing.

Very few natural convection test rigs operate at the high Rayleigh number of  $10^{10}$  and use air as the working fluid. The rigs which do operate at the high Rayleigh numbers often use simple geometries such as the heated vertical wall, as used by Tsuji and Nagano [97] and Siebers et. al. [78].

The flowfield in the vertical wall case is that of a thermal boundary layer growing as it rises up the heated wall. Whilst this is similar to the desired flowfield of a natural convection boundary layer growing along a heated outer casing, there are key differences. The differences in the development of the thermal boundary layer in a gas turbine casing compared to that from a vertical wall are shown schematically in Fig.3.1. In the vertical wall case the buoyancy force is always parallel to the heated wall, this means that the buoyancy forces are not aiding or inhibiting the development of the boundary layer along the wall. In the example of a gas turbine casing the buoyancy force changes direction relative to the boundary layer flow. At the bottom of the casing the flow is on top of the heated outer wall, the buoyancy force is then pulling the boundary layer upwards and aiding its growth. This buoyancy aided boundary layer growth can cause earlier transition to turbulence. Similarly at the top of the casing the flow is underneath the heated wall, the buoyancy force is then pushing the boundary layer upwards and inhibiting its growth. This inhibited growth can reduce the wall heat transfer.

As existing test rigs are not giving a representative flowfield, a new test rig is required in order to provide validation data for a gas turbine during shutdown. The new rig should operate at the same Rayleigh Number as a real industrial gas turbine during shutdown and



**Figure 3.1:** Schematic of flow in vertical wall and casing rig

use a engine representative geometry.

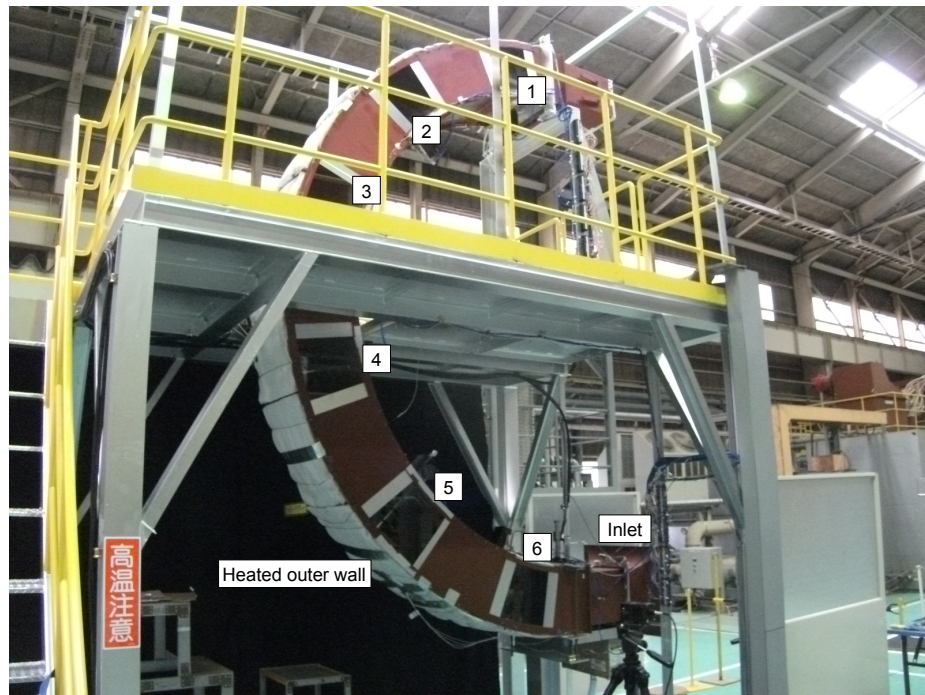
## 3.2 Casing Rig Design

### 3.2.1 Geometry

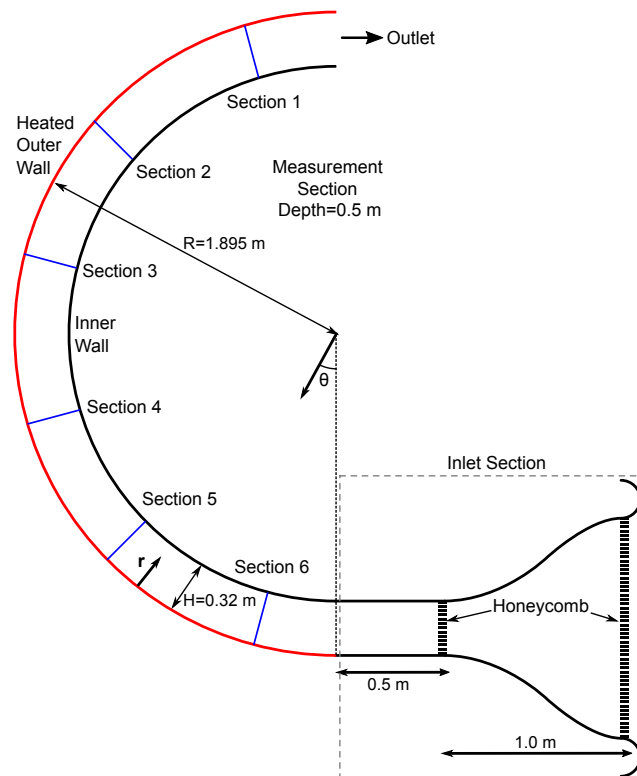
A new test rig was constructed by Mitsubishi Heavy Industries (MHI) at the research and development centre in Takasago, Japan. The aim of the rig was to provide CFD validation data for natural convection flows in an industrial gas turbine casing during shutdown. In order to match the high Rayleigh number found in gas turbines, the test rig was constructed to have the same diameter as a real engine and would also operate at real engine temperature differences. The casing rig, without the inlet contraction is shown in Fig.3.2.

The gas turbine casing rig is shown schematically in Fig.3.3. The rig can be split into two main parts, an inlet section containing the flow conditioning features and a measurement section, which contains the heated outer wall.





**Figure 3.2:** Gas turbine casing rig, without contraction inlet



**Figure 3.3:** Gas Turbine Casing Rig Schematic

## Measurement Section

The measurement section of the rig was a rectangular duct of height 0.32 m and depth 0.5 m which was curved into a “C” shape. The radius of the outer wall was 1.895 m in order to match the radius of an industrial gas turbine. The rig outer wall represents the hot outer casing of a gas turbine and was heated. The inner wall of radius 1.575 m would represent the combustors and inner part of the engine casing. The width of the rig was found to give a sufficiently two dimensional flowfield in the central section of the duct where the measurements were obtained.

Access was available at six locations in the rig measurement section to allow for traverses to be performed. These six locations were labelled section 1 to 6, with section 6 being the furthest upstream. Section 6 was located at an angle of  $15^\circ$  from the start of the heated wall with subsequent sections spaced apart by  $30^\circ$ . These six sections could be viewed through acrylic windows in the side of the rig.

The rig was constructed from rectangular steel sections mounted into a frame. The steel sections made up the side and inner wall of the rig, with holes in the side-walls for the acrylic viewing windows. The rig outer wall was made from a sheet of aluminium 3 mm thick, this was heated electrically using six silicone rubber heaters mounted on the reverse of the sheet. The six heaters could be adjusted independently to keep the outer wall at a uniform temperature. All other walls in the rig were unheated.

The flow was driven through the rig solely by the buoyancy forces created by the heating of the flow from the outer wall. The flow exited the measurement section into ambient surroundings of a building with a high ceiling. This meant that no thermally stratified region occurred at the rig exit, which could have impeded the flow through the rig.

## Inlet Section

The role of the inlet section was to provide a well conditioned uniform flowfield at the start of the measurement section. The first feature in the inlet section was a fine mesh screen across the inlet, designed to break up large flow structures. This was followed by a section of honeycomb to act as a flow straightener. The honeycomb was 40 mm in depth with

a core diameter was 6.3 mm and covered the full duct width and height. The main flow conditioning feature was a contraction of 9:1 area ratio and length 1 m. The contraction was to accelerate the flow coming into the rig and cause the boundary layers to be thinned. The contraction was followed by a second section of honeycomb then a 0.5 m straight section of duct to connect with the rig measurement section. The inlet section of the rig was designed by the author, details of the methodology behind the inlet design are given in Appendix.B.

### **3.2.2 Measurement**

The aim of the rig was to provide validation data for numerical simulations, the measurements taken were designed to reflect this requirement. Profiles of temperature were required at locations around the rig for comparison with CFD results. Detailed temperature data near to the heated wall is also needed to allow the calculation of wall temperature gradient and hence wall heat flux. The prediction of the wall heat flux being a key test of the accuracy of different numerical models.

Temperature measurements were taken at the six measurement sections around the rig. Temperature measurements were taken using rakes of 18 thermocouples which would measure the temperature profile across the duct height. The thermocouple spacing on the rake was not uniform with more thermocouples located closer together nearer to the heated wall. This is due to the largest temperature gradients occurring at the wall and small temperature gradients outside of the thermal boundary layer. The thermocouples used had a wire diameter of 0.2 mm and were K-type class 1 with an accuracy of  $\pm 0.9^{\circ}\text{C}$ . Details are given in Appendix.C.

The outer wall temperature was measured using thermocouples mounted between the wall and the heater. In total 78 thermocouples were used to allow the outer wall temperature distribution to be verified as uniform. On the inner wall 18 thermocouples were used to measure the temperature distribution.

Temperatures data from the thermocouples was recorded using a data acquisition system at a sample frequency of 1 Hz. Samples were taken over a period of 5 minutes giving 300 data points, averages were then computed using these 300 data points.

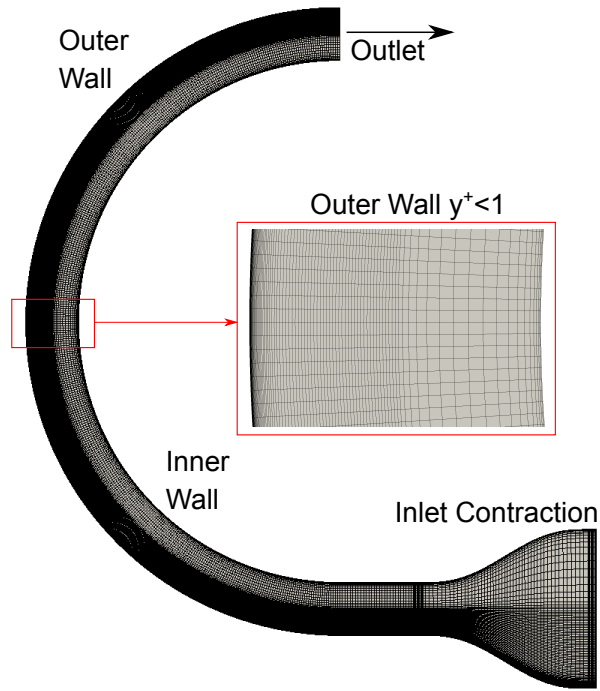
Temperatures data was also obtained using an optical fibre distributed temperature sensor. In an optical fibre system a laser is shone down an optical fibre and uses swept-wavelength interferometry to measure the Rayleigh scattered wavelength [50]. This wavelength is a function of the temperature allowing the temperature to be calculated. The measurements were taken using a LUNA ODiSI-B10 system. This was operated in high resolution mode allowing measurements to be taken with spacings of 0.64 mm along the fibre at a frequency of 11.9 Hz and an accuracy of  $\pm 1.6^\circ\text{C}$ . The optical fibre was arranged into 15 rows, each 106 mm in width. The closest row to the outer wall was at 1 mm distance. With the rows extending away up to 53 mm from the outer wall, details are given in Appendix.C. This technique allowed a temperature contour of area  $106 \times 52$  mm to be obtained simultaneously. The optical fibre was also traversed close to the heated outer wall and the first row of fibre data was used to obtain detailed wall temperature data. This allowed for the calculation of the wall temperature gradient and hence wall heat flux.

### 3.3 LES methodology

In order to give further insight into the flowfield inside of the casing rig, CFD simulations were conducted using the casing rig geometry. A Large Eddy Simulation (LES) was performed, this was chosen over a cheaper RANS simulation due to the desire to resolve streamwise vortices that were thought to be present inside of the rig.

The 3D computational domain consisted of the inlet and measurement sections of the rig, shown in Fig.3.4. The computational mesh used was a structured hexahedral type, which in total contained  $7.0 \times 10^6$  cells. The cells were clustered near to the outer wall to give an outer wall  $y^+ < 1$  and to give greater resolution to flow structures occurring near to the outer wall. It was found that the maximum ratio of cell size to Kolmogorov length-scale was around 10, making the cell size sufficient for well resolved LES.

The inlet and outlet of the domain had pressure inlet and outlet boundary conditions applied respectively. Both the inlet and outlet were set at ambient pressure, meaning that the flow was only being driven by buoyancy forces. The inlet temperature was set at  $28^\circ\text{C}$  with an inlet turbulence intensity of 5%. All of the other boundary conditions were wall bound-

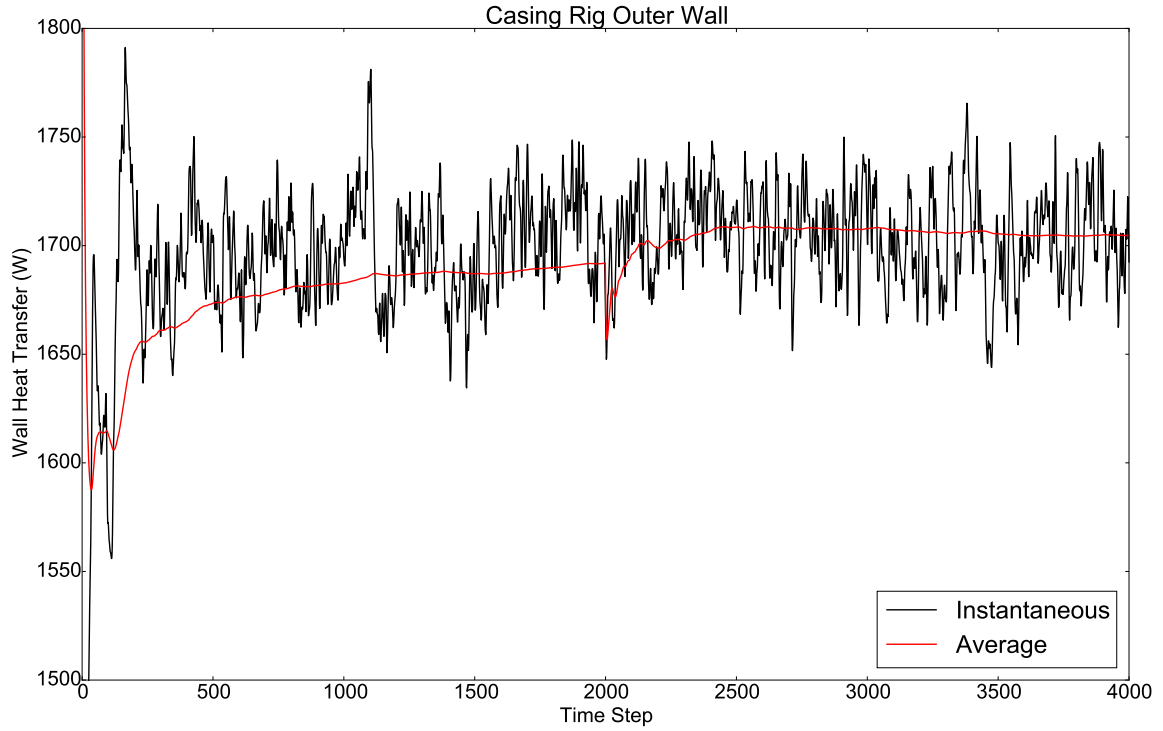


**Figure 3.4:** Cross section of computational mesh from LES simulation

aries. The outer wall and inner walls had a constant temperature boundary condition set to match the experiment. The bottom surface of the straight section of duct in the inlet section also had a temperature profile applied, this was to represent the heating it received through conduction from the outer wall. All of the other walls in the simulation were treated as adiabatic.

The honeycomb used in the rig inlet section acted as a flow straightener. It's affect was included in the simulation by modelling the honeycomb as a porous zone, with the flow exiting the porous zone axially. The porous zone was modelled as being non-restrictive, which may have led to the mass flow rate in the simulation being higher than in the experiment. Based on the experiments of Roach [72] there would be approximately one dynamic head of loss through the honeycomb.

The solver used was ANSYS Fluent v15.0, the numerical scheme was pressure based and used third-order MUSCL spatial discretization. A bounded second order implicit scheme was used for time-stepping. The subgrid-scale model used was the Smaoginsky-Lilly model with dynamically calculated Smaoginsky coefficient and turbulent Prandtl number [48].



**Figure 3.5:** History monitor of outer wall heat transfer

The simulation was run using a timestep of 0.1 s. The buoyant time-scale is given by  $\tau = \sqrt{R/(g\beta\Delta T)}$ , which is equal to 0.69 s for the casing rig case. A rule of thumb is that the simulation timestep  $\Delta t$  should be  $\approx 0.25\tau$  [4]. The simulation timestep of 0.1 s also kept the the maximum cell Courant number around 20. The simulation was initially run for 2000 timesteps, equivalent to  $\approx 10$  through-flow periods from inlet to outlet. This initial period was to allow for transients to decay before averages were taken over a further 2000 timesteps.

In order to check the simulation for convergence, several history monitors of the flow were used. Monitors of the instantaneous and time average outer wall heat transfer were used. When the average outer wall heat transfer stopped changing this showed the thermal boundary layer next to to outer wall had reached a steady state. The flow was also monitored at a spot point at section 6 where the instantaneous mean and rms values of temperature and velocity were monitored. Section 6 was chosen as a monitor location as it is in the transition region which could take longer to converge. These monitors then showed that the simulation had been run for a sufficient length of time, this can be seen in Fig.3.5.

## 3.4 Casing Rig Flowfield

### 3.4.1 Comparison to Vertical Wall

The reason for the requirement for a test rig to be representative of a gas turbine geometry can be seen by comparing the casing rig to the vertical wall test case. The differences between the casing rig and vertical wall flowfields is apparent when looking at the heat transfer from the heated surface, shown in Fig.3.6.

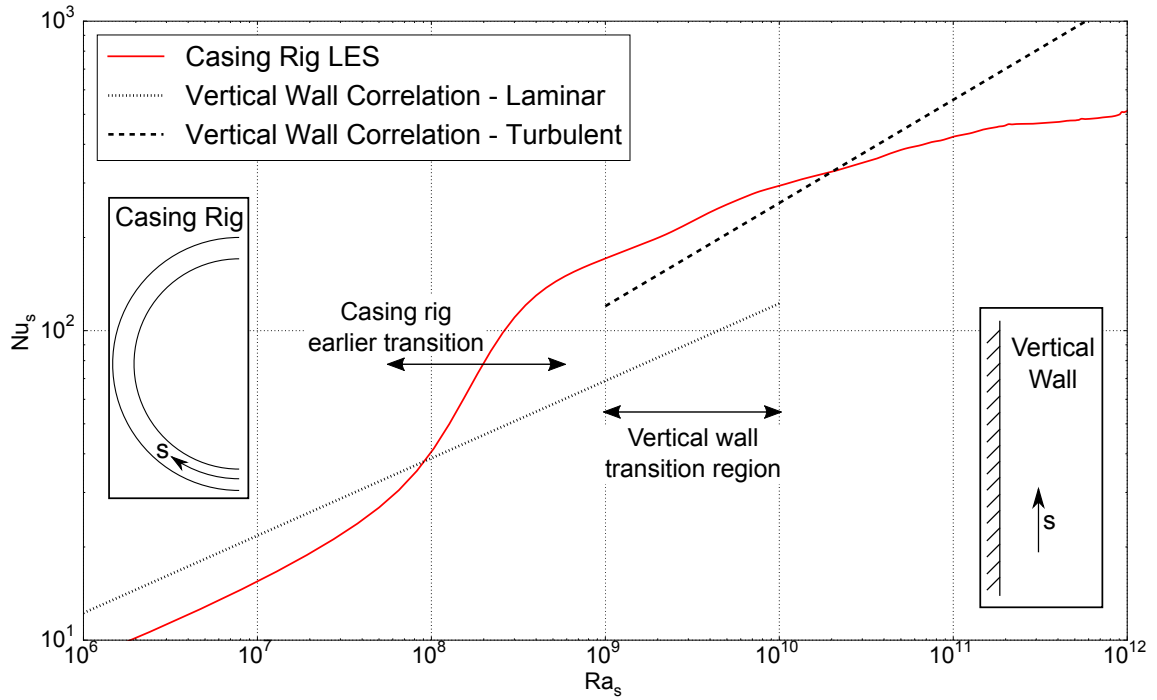
The flow in the vertical wall case is a boundary layer flow rising up the heated wall. The flow starts off as laminar before transitioning to turbulence, with the transition happening at a Rayleigh number between  $10^9 - 10^{10}$  [97]. The Nusselt number in the laminar regime follows the trend  $Nu_s \propto Ra_s^{1/4}$  and in the turbulent regime  $Nu_s \propto Ra_s^{1/3}$ .

The flow in the casing rig is also a boundary layer type of flow, which again starts off as laminar before transitioning to turbulent flow. The heat transfer in the laminar region follows the same  $Nu_s \propto Ra_s^{1/4}$  slope as the vertical wall but the transition now happens earlier at a Rayleigh number  $\approx 10^8$ . This earlier transition is likely due to the boundary layer growth being enhanced by the buoyancy force, as the flow is on top of the heated surface, Fig.3.1. At the top of the casing rig the flow is now underneath the heated surface where the buoyancy force is restricting boundary layer growth. This has the effect of reducing the heat transfer towards to top of the rig. It can be seen that the slope of Nusselt number is reduced from the  $Nu_s \propto Ra_s^{1/3}$  in the vertical wall case.

### 3.4.2 Temperature and Velocity field

The flowfield in the casing rig is shown in Fig.3.7. Looking at the temperature contour it can be seen that there is a thermal boundary layer growing as the air flows over the heated outer wall. It is the growth of this thermal boundary layer which is aided or restricted by the buoyancy forces.

The mean velocity profile of the rig is shown by stream-traces coloured by velocity magnitude. It can be seen that towards the inlet of the duct the velocity profile is very uniform across the duct height. As the flow develops the buoyancy forces cause a velocity peak to occur near to the heated outer wall. Near the the inner wall the buoyancy forces are



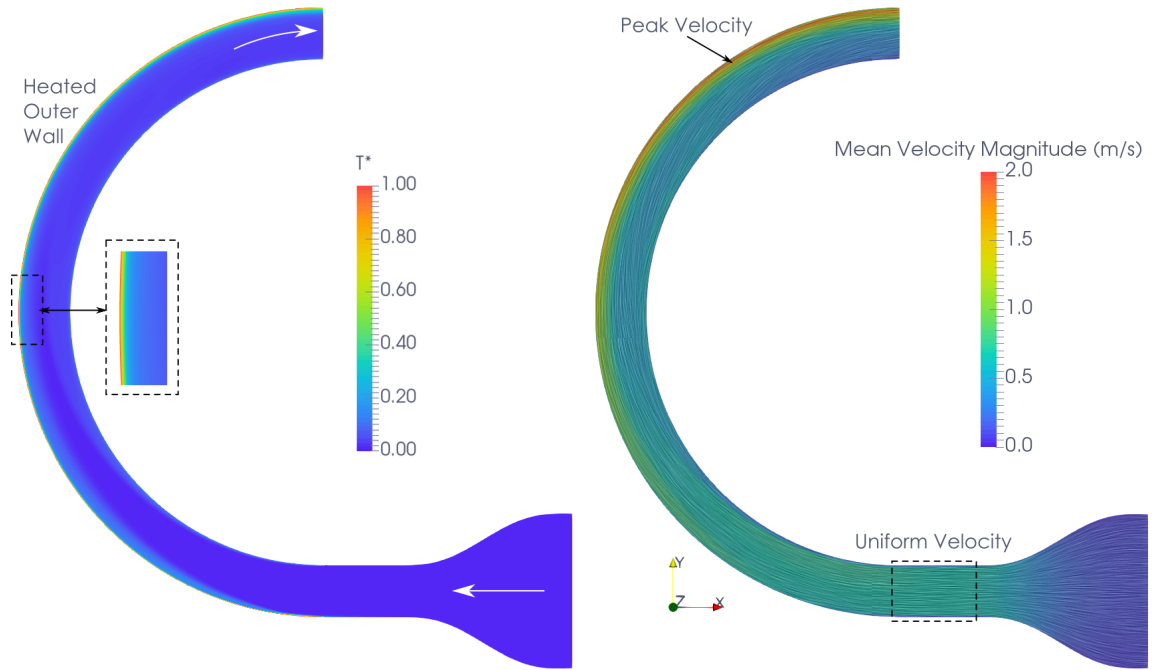
**Figure 3.6:** Heat Transfer Profile of Casing Rig and Vertical Wall

smaller and the velocity is reduced compared to the inlet velocity.

Temperature profiles measured experimentally using the thermocouple rakes and the fibre traverse are compared to the time averaged LES results at the six measurement locations in Fig.3.8. The temperature profiles have been non-dimensionalised as  $T^* = (T - T_{inlet}) / (T_{outer\ wall} - T_{inlet})$ .

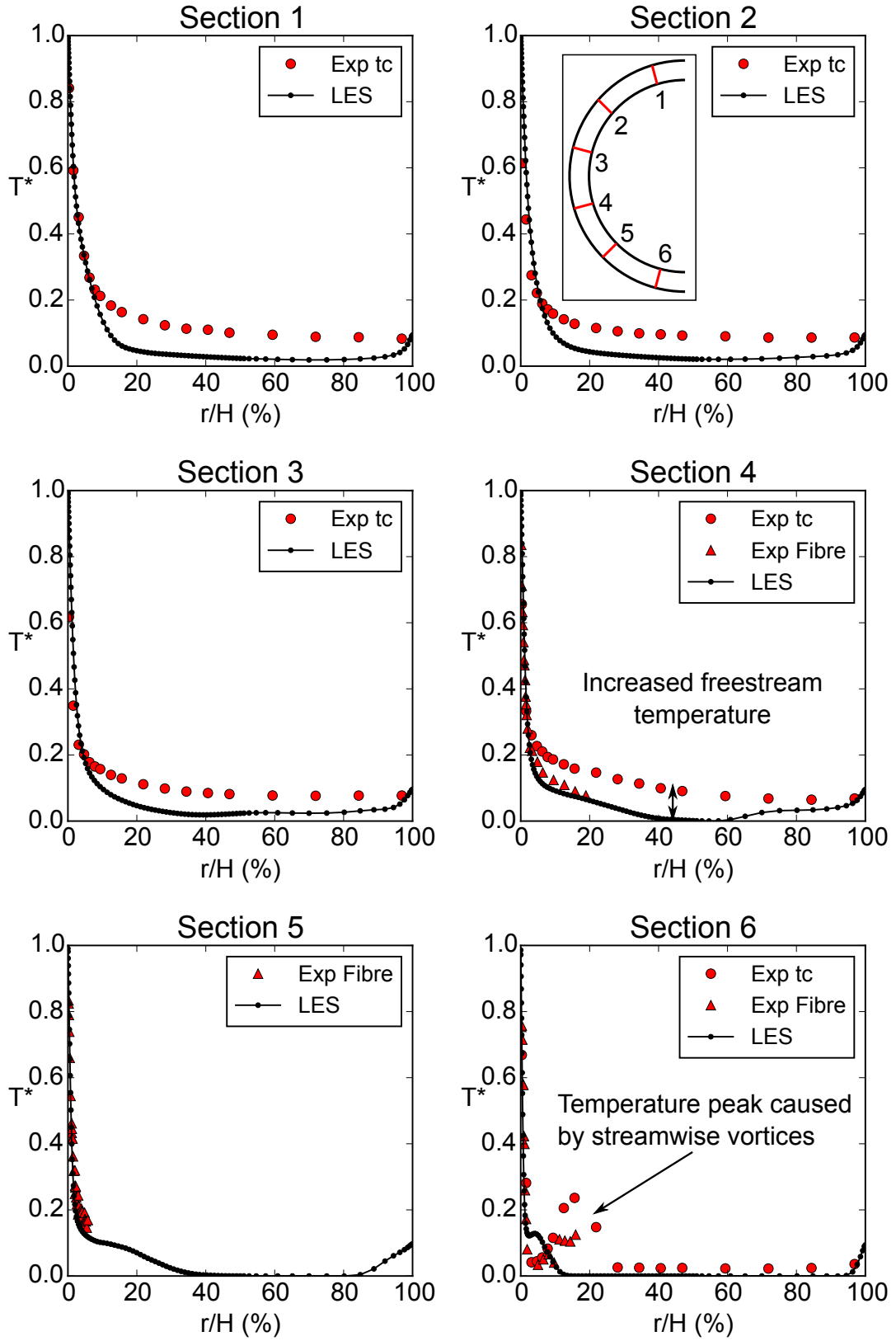
At the furthest upstream measurement location, section 6, a peak in the experimental temperature profile can be observed. This peak is caused by the effect of streamwise vortices pulling hot fluid from the boundary layer into the free stream. These vortices then increase the mixing between the boundary layer and the free stream, which can be observed by the increased free stream temperature downstream. The temperature peak predicted by the LES is smaller than that measured in the experiment, this is due to the difficulty in predicting the formation and growth of the streamwise vortices numerically. The smaller vortices predicted by the LES have had an effect on the free stream temperature, as the smaller vortices have reduced the mixing between the free stream and boundary layer. This lower free stream temperature in the LES predictions can be seen in all of the downstream sections, showing that the vortices can effect the flowfield at large downstream distances.





**Figure 3.7:** Time average temperature and velocity contours from LES, on mid-plane of casing rig

It can be seen that at all six locations that whilst there is some disagreement with the free stream temperature the LES is predicting well the temperature profile near to the outer wall. It is the temperature gradient at the outer wall that is setting the wall heat flux, which is a key output of the simulation. The fibre experimental data, available at sections 6, 5 & 4, has sufficient spatial resolution with which to calculate the wall temperature gradient. The gradient was calculated through a linear best fit through the first few points which made up the linear region of the temperature profile. The wall heat flux is then given by  $\dot{q} = -\lambda \frac{dT}{dr}$ . The calculated wall heat flux from the fibre temperature data and the LES is given in Table.3.1. It can be seen that in section 6 where the streamwise vortices are present the LES is significantly over-predicting the heat transfer. This is due to the difficulties in predicting the streamwise vortices and their large effect on the wall heat transfer. Further downstream in the duct, where the vortices have less effect, the prediction of heat transfer is improved. At section 4 the difference between the experiment and LES is then less than 10%.



**Figure 3.8:** Thermocouple and fibre experimental and LES temperature profiles at six measurement locations

| Section | Experiment | LES  | Difference |
|---------|------------|------|------------|
| 6       | 622        | 1186 | 90.6%      |
| 5       | 640        | 780  | 21.8%      |
| 4       | 637        | 587  | -7.8%      |

**Table 3.1:** Experiment and LES wall heat fluxes ( $W/m^2$ )

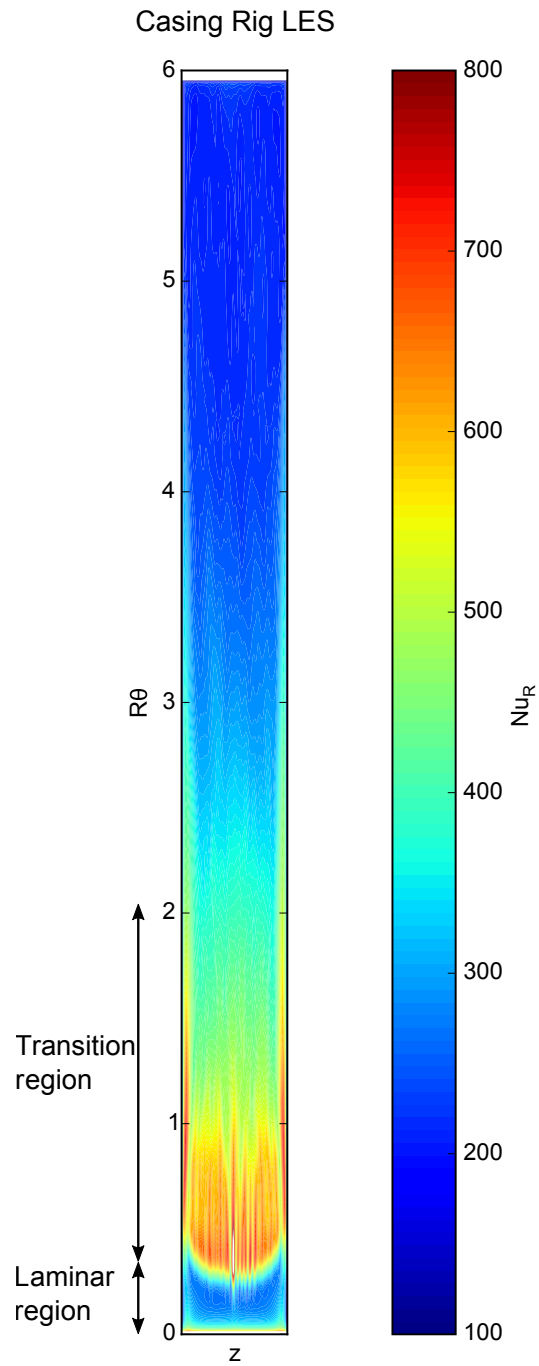
### 3.4.3 Outer Wall Heat Transfer

The flow in the casing rig can also be understood by looking at a profile of the outer wall heat transfer. A time averaged profile of non-dimensional outer wall heat transfer from the LES simulation is shown in Fig.3.9. The flow at the start of the heated outer wall is laminar. The laminar region can be seen as the initial region on the heated wall with lower heat transfer. At a distance  $\approx 0.25$  m downstream from the start of the outer wall, the flow begins to transition to turbulence. The transition region is visible as the region of increased outer wall heat transfer. In this transition region streamwise streaks of alternating high and low heat transfer can be seen, these streaks in heat transfer are caused by the formation of streamwise vortices. Finally downstream of the transition region the coherent streamwise vortices break down and the flow enters the turbulent regime.

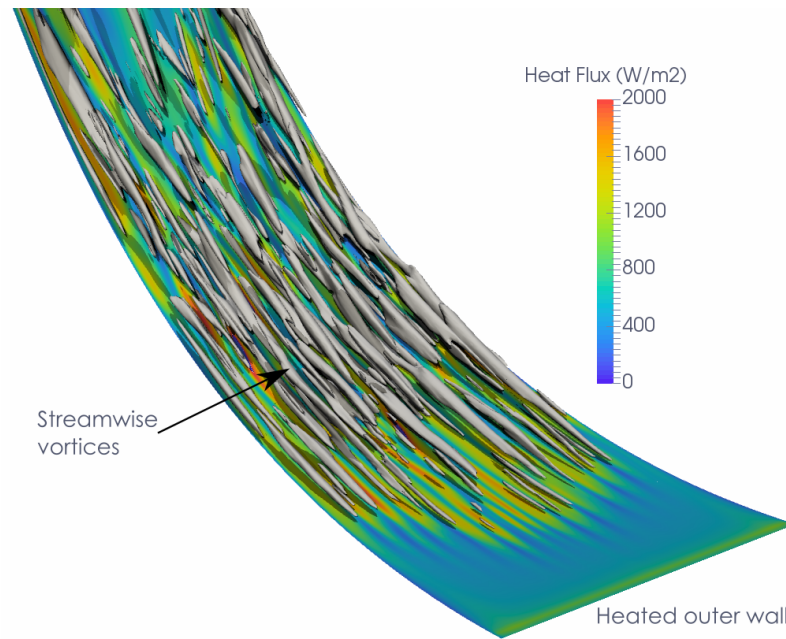
### 3.4.4 Streamwise Vortices

The streamwise vortices in the transition region form when a laminar boundary layer flows over a heated surface. When flow is on top of a heated surface there is a component of the buoyancy force that is normal to the boundary layer flow direction. It is this force that causes the boundary layer to wrap up into counter-rotating pairs of vortices. These vortices have been observed by several authors in the literature including Cheng and Kim [12] and Sparrow and Husar [86]. The formation of streamwise vortices is the transition mechanism for this flow. This differs from the heated vertical wall where transition occurs through the Tollmien-Schlichting wave instability [49]. It was found by Lloyd and Sparrow [49] that at plate angles greater than  $17^\circ$  from the vertical, the vortex instability was the dominant method of transition over the wave instability. The wave instability was dominant at plate angles of less than  $14^\circ$  from the vertical.

The streamwise vortices can be visualised by looking at iso-surfaces of Q-criterion



**Figure 3.9:** Contours of outer wall heat transfer in casing rig from time averaged LES

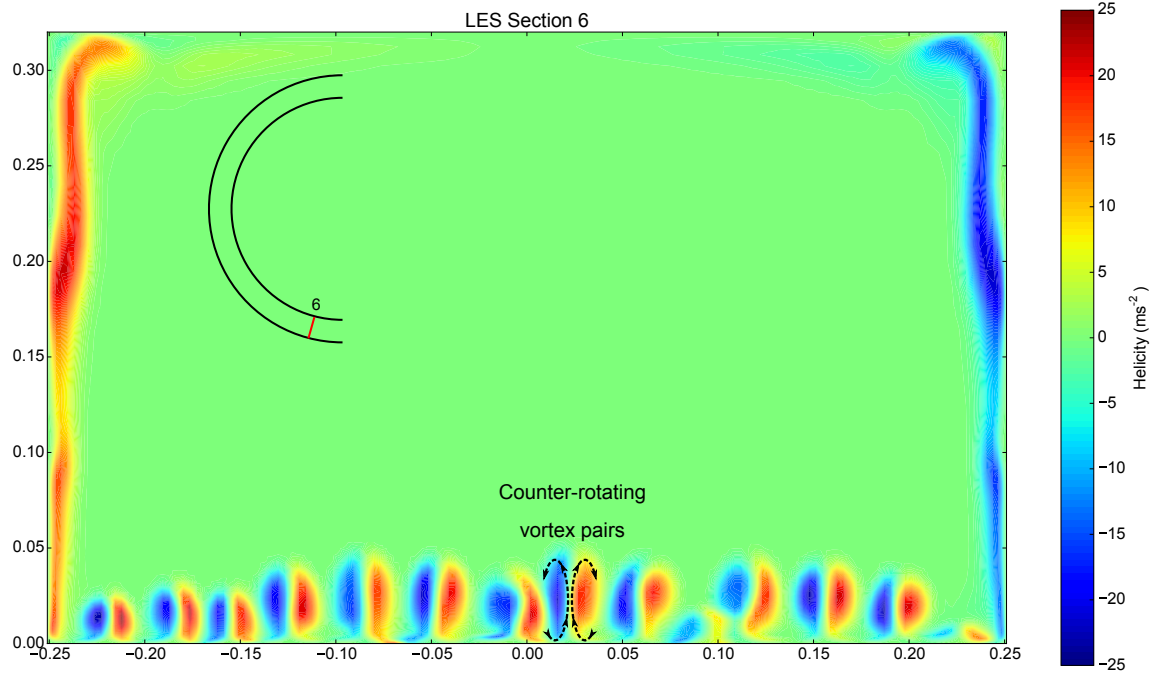


**Figure 3.10:** Contours of Q-criterion showing streamwise vortices next to heated wall

[29], this is shown in Fig.3.10. The vortices develop next to the heated wall and form in a streamwise direction. It can be seen that the location of the vortices align with the streaks of enhanced wall heat transfer. Further downstream of the transition region the number of visible vortices is reduced, and the vortices start to lose their streamwise orientation.

Looking at a contour of helicity in Fig.3.11, the streamwise vortices can be visualised. The contour is at section 6 which is located in the transition region. Helicity is a measure of the streamwise vorticity and can be used to visualise vortex cores, the sign of the helicity shows the direction of rotation of the vortices. It can therefore be seen that the streamwise vortices in the casing rig are forming into counter rotating pairs, which is consistent with the literature results.

The effect of the streamwise vortices on the wall heat transfer is shown schematically in Fig.3.12. In regions where the flow from the counter rotating vortex pairs impinges on the heated surface, the heat transfer is increased. This is due to the vortex motion compressing the thermal boundary layer. The thinner thermal boundary layer then has steeper wall temperature gradients giving higher wall heat transfer. Similarly in the regions where the vortex motion moves fluid away from the heated surface, the thermal boundary layer is thickened. This thicker thermal boundary layer has a lower wall temperature gradient and

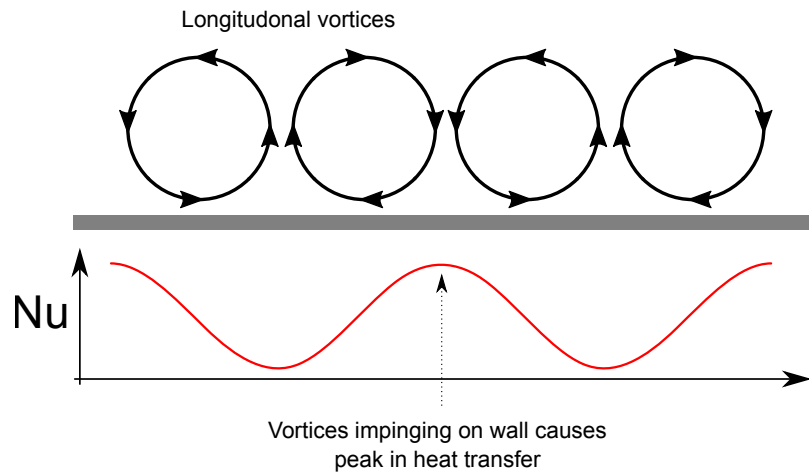


**Figure 3.11:** Contours of helicity at section 6 of casing rig, looking into rig

hence a lower heat transfer. These regions of increased and decreased heat transfer give the characteristic streamwise stripes in contours of wall heat transfer.

The effect of the thermal vortices thinning and thickening the thermal boundary layer can be seen by looking at temperature contours from a cross section of the rig, Fig.3.13. The temperature contours are taken from section 6 of the rig and compare data from the LES simulation with experimental data taken using the fibre measurement system. The experimental fibre data was only obtained in the central part of the duct due to access limitations in the rig. It can be seen that the LES has predicted a very similar temperature contour to the experimental data with the vortex in the experiment being larger, showing that the simulation has predicted a similar vortex formation compared to the experiment.

Looking the temperature contours it can be seen that there are distinctive mushroom shaped structures close to the heated wall. These structures are due to the vortices pulling hot fluid away from the thermal boundary layer and into the free stream. At the stem of the mushroom shape the thermal boundary layer is thickest. In this region is where the vortices are moving hot fluid away from the wall, corresponding to the low heat transfer region. In the region between the mushroom shapes the boundary layer is thinnest, this



**Figure 3.12:** Schematic showing effect of streamwise vortices on wall heat transfer

is where the vortices are impinging on the wall. The action of the vortices increases the mixing between the thermal boundary layer and the free stream, therefore increasing the free stream temperature.

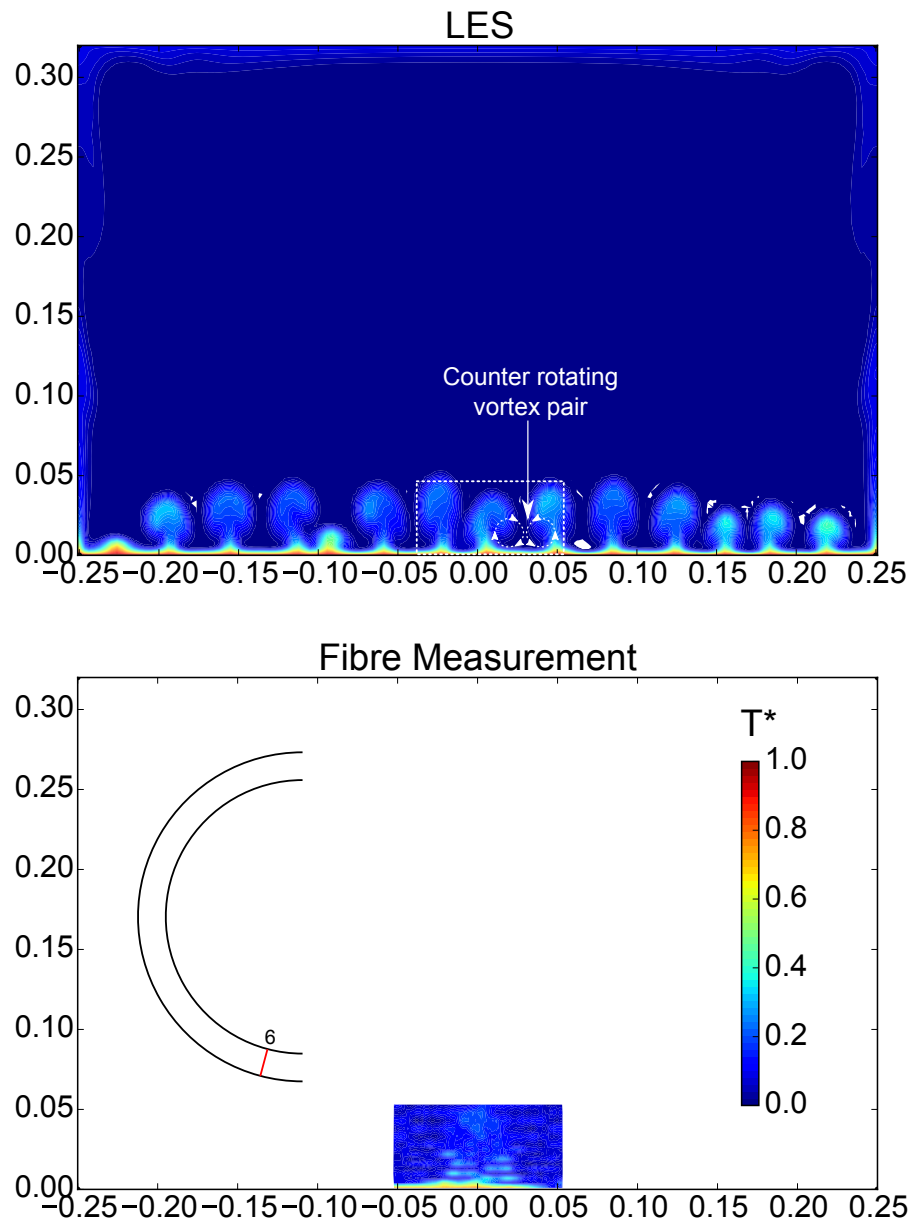
### 3.5 Conclusions

As CFD validation data is very difficult to obtain from a real engine during shutdown a new natural convection test rig was constructed by MHI.

The gas turbine casing rig had geometry that was representative of a large industrial gas turbine and had the same diameter as a real engine. The rig operated at the same Rayleigh Number as a real engine as it matched the engine temperature difference and size. This meant that the rig would give an engine representative flowfield. The curved outer wall of the rig was heated to represent the hot outer casing of a gas turbine. It was the buoyancy forces from this heated outer wall which drove the flow through the rig. CFD validation data was obtained using rakes of thermocouples and a fibre temperature sensor. This allowed temperature profiles and wall heat fluxes to be measured.

In order to better understand the flowfield inside of the casing rig, an LES study was performed. The simulation allowed temperature and velocity contours to be visualised that could be obtained from the experimental data.

It has been shown how the flowfield in the rig with engine representative geometry



**Figure 3.13:** LES and experimental temperature contours at Section 6 of casing rig



differs from a vertical wall rig operating at the same Rayleigh Number. The transition mechanism in the vertical wall is via a wave instability whereas in the casing rig transition is due to the formation of streamwise vortices. It was shown that the vortices have a significant effect on the heat transfer and cause local heat transfer enhancement as they impinge on the outer wall.

The thermal vortices also have an effect on the temperature in the free stream outside of the outer wall thermal boundary layer. The vortices cause increased mixing between the boundary layer and the free stream, raising the free stream temperature. The effect of this raised free stream temperature can be seen far downstream of the vortices. It was found that it was difficult to predict the formation of the vortices using the LES simulation. The vortex visualised in the experiment was larger than the LES prediction, this meant that the LES under-predicted the free stream mixing and temperature peak caused by the vortices.

The temperature gradient at the heated outer wall defines the heat transfer, which was a key output of the simulation. The predicted heat flux from the LES simulation was found to be giving good agreement with the experimental data, apart from where the vortices were present.

# Chapter 4

## Tall Cavity Study

*There are very few experimental validation cases for natural convection where the turbulent heat fluxes ( $\overline{u'T'}$ ) were measured. This is due to the difficulty in measuring the fluxes, as a simultaneous measurement of temperature and velocity fluctuations at the same location is required.*

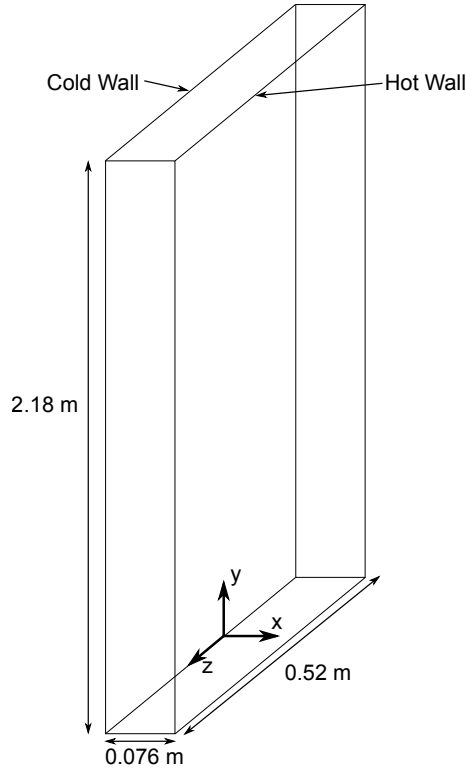
*Both mean and rms data from an experimental test case of a differentially heated tall cavity is available from Betts and Bokhari [6]. For this case both temperature and vertical velocity data is available at various locations inside of the cavity. The temperature and velocity data was not measured simultaneously and so the turbulent heat flux terms are not available.*

*A simulation was performed using the tall cavity geometry which was validated using the available experimental data. This simulation was then able to provide details of other flow features and statistics which were not available from the experimental data, such as the turbulent heat flux terms.*

*A Direct Numerical Simulation (DNS) study was conducted in order to obtain resolved turbulent statistics. The DNS results were also used to visualise the flowfield inside of the cavity and identify key flow features such as laminar and transition regions.*

### 4.1 Background

The differentially heated closed tall cavity case was investigated experimentally by Betts and Bokhari [6]. The cavity, shown schematically in Fig.4.1, had dimensions  $2.18 \times 0.076 \times$



**Figure 4.1:** Schematic of tall cavity test case, modified from [6]

0.52 m giving an aspect ratio of over 28:1. The two vertical walls were kept at a constant temperatures, one was heated electrically and the other cooled with a water jacket. Temperature differences of 19.6°C and 39.9°C between the hot and cold walls were used for the experiment, giving Rayleigh numbers of  $0.86 \times 10^6$  and  $1.43 \times 10^6$  respectively, based on the cavity width. The wall boundary conditions in this experiment were designed to be simple to implement in CFD simulations, this is one of the reasons that the tall cavity was chosen as a validation case.

Measurements were taken of both the temperature and velocity in the cavity. Profiles of both mean and rms values were obtained at several different heights in the cavity. Temperature measurements were taken using a single k-type thermocouple traversed across the cavity and velocity measurements were taken using a LDA system with seeding provided by smoke particles. The large dataset available was another reason that the tall cavity was chosen as a validation case.

## 4.2 Numerical Method

As the turbulent heat fluxes and other turbulent statistics were a key output of this simulation it was decided to use a Direct Numerical Simulation (DNS). This is due to the fact that a DNS simulation resolves the turbulent fluctuations down to the smallest length-scales, in contrast to an LES simulation where only the larger scales are resolved and the smaller scales are modelled. If an LES simulation was used there would be some uncertainty in the turbulent statistics, as a contribution to the statistics would be due to the effect of the sub-grid scale model.

The solver used in this DNS study was the high order spectral element code NEK5000 [63]. The computational domain used the complete tall cavity geometry and consisted of  $140 \times 10 \times 30$  elements in the height, width and depth respectively. The solver used a high order accuracy numerical scheme which was 5th order spatial accurate and 3rd order time accurate.

The Navier Stokes equations are solved in a non-dimensional form in the NEK5000 solver. To non-dimensionalise the equations the cavity width  $L$  is used as the length-scale. The buoyant velocity  $U = \sqrt{g\beta(T_{hot} - T_{cold})L}$  is used as the velocity scale, and the non-dimensional time-scale is then  $\tau = L/U$ . The temperature is non-dimensionalised as  $\theta = (T - T_{cold})/(T_{hot} - T_{cold})$ . Where  $T_{hot}$  and  $T_{cold}$  are the hot and cold wall temperatures respectively.

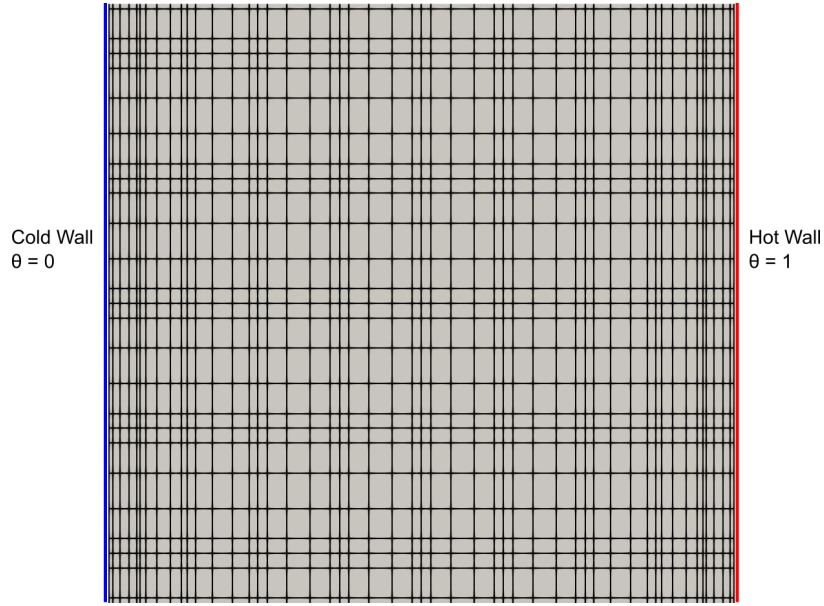
The Boussinesq approximation is used for the density formulation. The density is then treated as a constant, simplifying the Navier Stokes equations. The non-dimensionalised continuity, momentum and energy equations are then given by Equations.(4.1, 4.2 and 4.3) respectively. Where  $Gr$  is the Grashof number  $Gr = g\beta(T_{hot} - T_{cold})L^3/\nu^2$  and  $Pr$  is the Prandtl number  $Pr = \nu/\alpha$ .

$$\nabla \cdot \mathbf{u} = 0 \quad (4.1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla = -\nabla p + \frac{1}{\sqrt{Gr}} \nabla^2 \mathbf{u} + \theta \quad (4.2)$$

|            |                              |
|------------|------------------------------|
| $T_{hot}$  | 34.7°C                       |
| $T_{cold}$ | 15.1°C                       |
| $L$        | 0.076 m                      |
| $\beta$    | $3.36 \times 10^{-3} K^{-1}$ |
| $U$        | 0.221 m/s                    |
| $\tau$     | 0.343 s                      |
| $Gr$       | $1.165 \times 10^6$          |
| $Pr$       | 0.713                        |

**Table 4.1:** Values used to non-dimensionalise Navier Stokes



**Figure 4.2:** Mid-plane section of mesh used for DNS simulation

$$\frac{\partial \theta}{\partial t} + \mathbf{u} \cdot \nabla \theta = \frac{1}{\sqrt{GrPr}} \nabla^2 \theta \quad (4.3)$$

The values used to non-dimensionalise the Navier Stokes equations are given in Table.4.1.

The size of domain was  $28L \times 1L \times 6L$  in the height, width and depth respectively. The  $140 \times 10 \times 30$  elements were distributed uniformly in the height and depth directions but in the cavity width the elements were clustered towards the walls, as can be seen in Fig.4.2. As the simulation was run with 5th order spatial accuracy, each element was split into  $5 \times 5 \times 5$  cells. This gave a total cell count for the domain of  $5.25 \times 10^6$  cells.

The domain boundaries consisted of the six cavity walls, where a no-slip velocity con-

dition was applied. The hot and cold walls both had a constant temperature boundary condition applied, at the cold wall  $\theta = 0$  and at the hot wall  $\theta = 1$ . For the top and bottom walls the temperature varied linearly from 0 to 1 across the cavity width. The front and back cavity walls had an adiabatic thermal boundary condition applied.

The simulation used a non dimensional time-step of  $5 \times 10^{-3}$  in order to keep the CFL number  $\approx 0.5$  and the calculation stable. The simulation was allowed to fully develop before averages were calculated over a following 2 million time-steps. The time taken for the flow to complete one circulation, of rising up the heated cavity wall and falling down the cooled wall, was  $\approx 400\tau$ . This means that the averages were calculated over 25 circulation periods, which equates to 57 minutes of flow time.

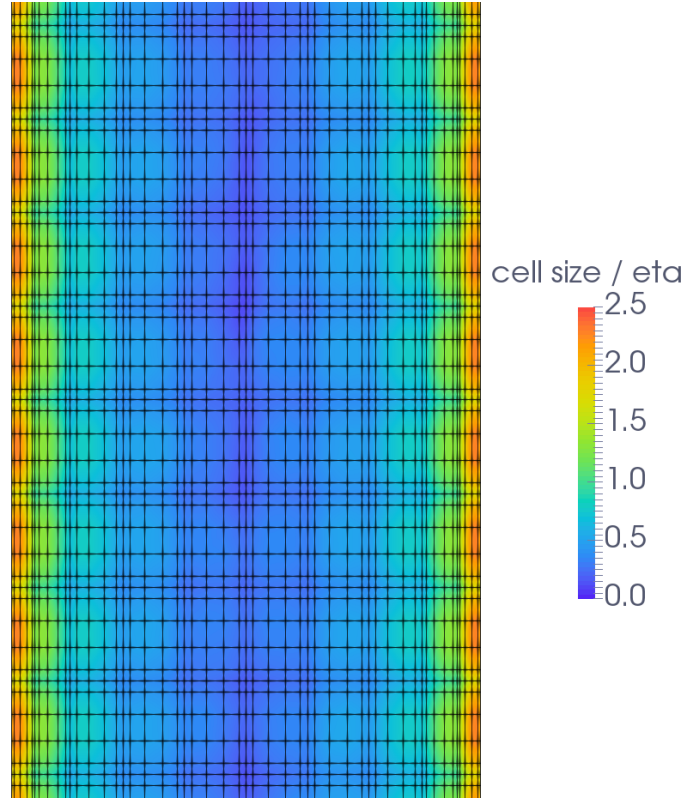
## 4.3 Simulation Validation

In order to validate the DNS simulations the results will be compared to the available experimental data. As an additional quality check the mesh size will be compared to the Kolmogorov scales, to check if the DNS has been sufficiently well resolved. The turbulent spectra will also be compared to standard correlations.

### 4.3.1 Numerical Mesh Validation

The Kolmogorov scales are the characteristic scales of the smallest turbulent motions [68]. The mesh used for the DNS simulation is compared to the Kolmogorov length scale. If the simulation is well resolved the mesh size should be of the same order as the Kolmogorov scale. The Kolmogorov length scale is defined as  $\eta = (\frac{\nu^3}{\epsilon})^{\frac{1}{4}}$ , where  $\nu$  is the kinematic viscosity and  $\epsilon$  is the dissipation of turbulent kinetic energy. The mesh cell size is defined as the cube root of the cell volume.

Looking at a contour of the ratio of cell size to Kolmogorov scale on cavity mid-plane in Fig.4.3 shows that the largest ratio is less than 2.5. With the bulk of the domain having a ratio of less than one. This low ratio of cell size to Kolmogorov scale gives confidence that the turbulent eddies are being resolved and not filtered out by a coarse grid. The wall shear length scale is given by  $\Delta y^+ = U_\tau \Delta y / \nu$ , where  $U_\tau = \sqrt{\tau_{wall} / \rho}$  is the friction

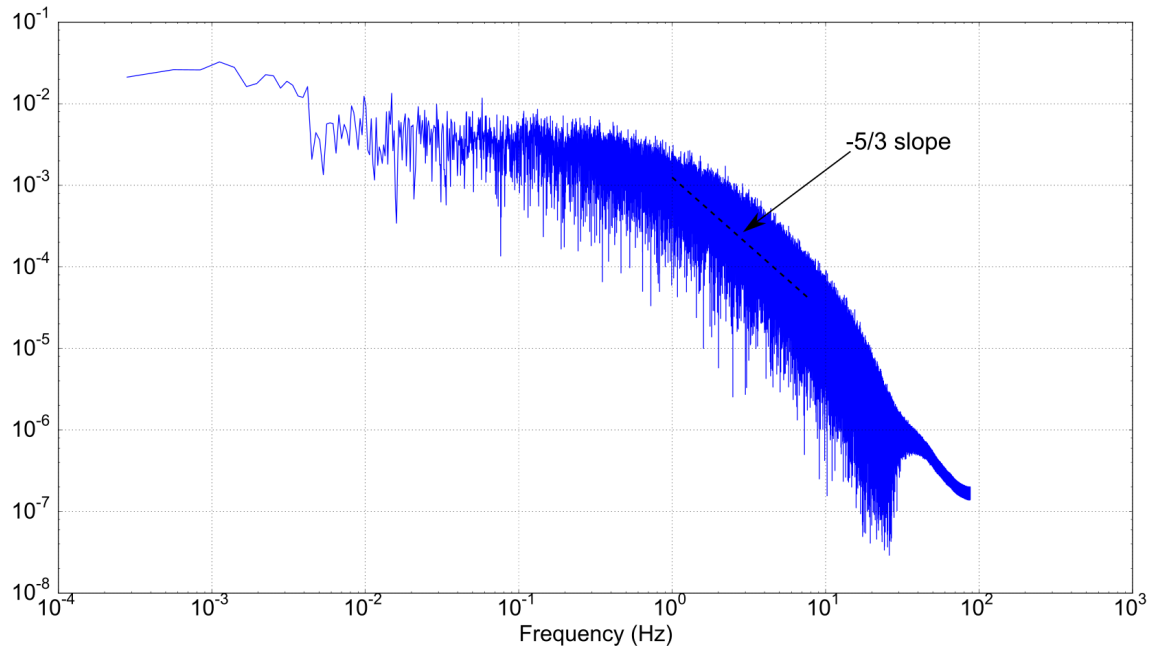


**Figure 4.3:** Ratio of cell size to Kolmogorov length-scale at cavity mid height

velocity. The calculated mesh spacings based on wall shear at the hot wall are 0.8, 8 and 8 for  $\Delta x^+$ ,  $\Delta y^+$ ,  $\Delta z^+$ , the wall normal, streamwise and spanwise directions respectively. These low values give further confidence that this DNS is sufficiently resolved [71].

A spectral plot of velocity fluctuation against frequency is given in Fig.4.4. The velocity was sampled at a point on the cavity mid-plane at 50% height and a location 8% of the cavity width from the cold wall. It can be seen that the initial decay of turbulent energy follows a  $-5/3$  slope, which is consistent with the energy spectra for isotropic turbulence [68].

The Kolmogorov time-scale is given by  $\tau_\eta = (\frac{\nu}{\epsilon})^{\frac{1}{2}}$ . The smallest value of the Kolmogorov time-scale on the cavity mid-plane is  $89 \times 10^{-3}$ , the simulation time-step of  $5 \times 10^{-3}$  is therefore over an order an magnitude smaller than the Kolmogorov time-scale. This means that the DNS has sufficient temporal resolution to resolve the fastest turbulent eddies.



**Figure 4.4:** Spectrum plot of DNS velocity at a point on the cavity mid plane at 50% height and 8% of the cavity width

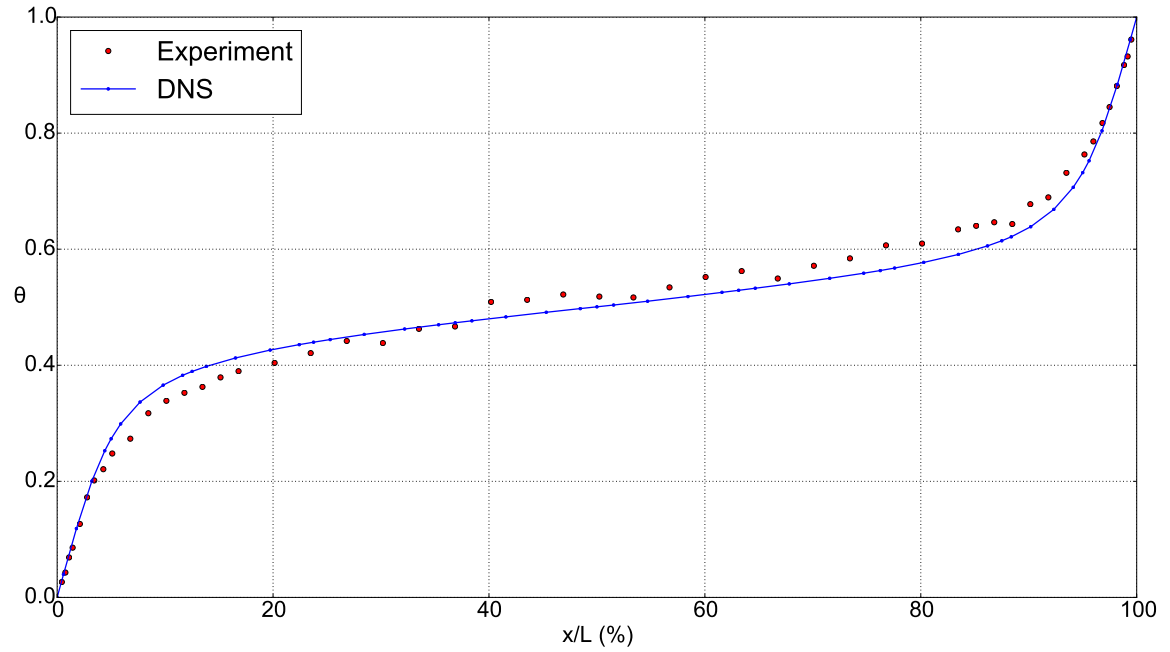
### 4.3.2 Experimental Data Validation

#### Temperature validation

Experimental validation data is available for the mean and rms temperatures at several cavity locations. A comparison of the mean temperature profile on the mid-plane of the cavity at 50% height can be seen in Fig.4.5. It can be seen that there is good agreement between the experimental data and the DNS data. The prediction of the temperature gradient near the walls is also important as the wall temperature gradient sets the wall heat transfer. As can be seen in Table.4.2 there is good agreement between the two datasets, particularly as the calculation of the temperature gradient is susceptible to noise in the experimental data. To reduce the effect of noise the gradient was calculated by a linear best fit through the 2nd to the 5th point away from the wall.

A plot of rms temperature fluctuations at 50% cavity height is shown in Fig.4.6. It can be seen that the DNS is predicting temperature fluctuations which are larger than the experimentally measured values. The profile of temperature fluctuation is also different, the experiment having a very flat profile across the cavity width and the DNS having distinct

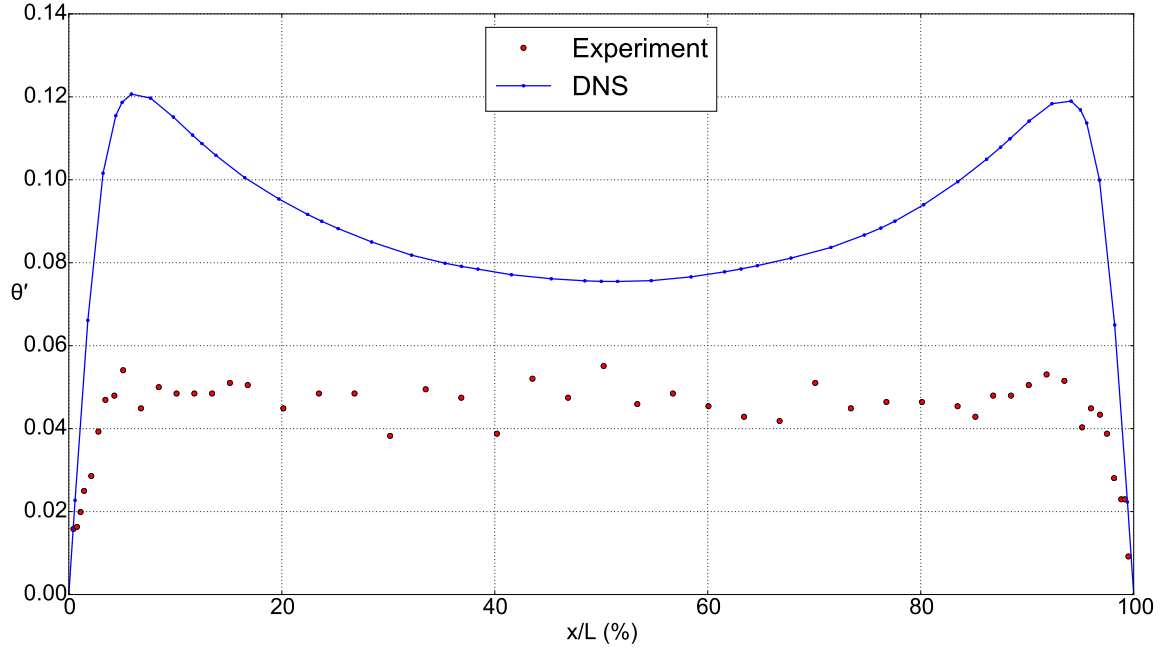




**Figure 4.5:** Non-dimensional mean temperature profile on cavity mid-plane at 50% cavity height

| y/H (%) | Experiment | DNS   | Difference |
|---------|------------|-------|------------|
| 95      | -65.5      | -62.4 | -4.7%      |
| 90      | -52.9      | -50.2 | -5.2%      |
| 70      | -31.0      | -41.4 | 33.7%      |
| 60      | -32.9      | -39.9 | 21.1%      |
| 50      | -42.0      | -40.2 | -4.3%      |
| 40      | -47.4      | -39.1 | -17.5%     |
| 30      | -35.6      | -36.9 | 3.6%       |
| 10      | -21.9      | -15.6 | -28.7%     |
| 5       | -16.9      | -11.2 | -33.7%     |

**Table 4.2:** Experimental and DNS cold wall heat flux at different cavity heights ( $W/m^2$ )

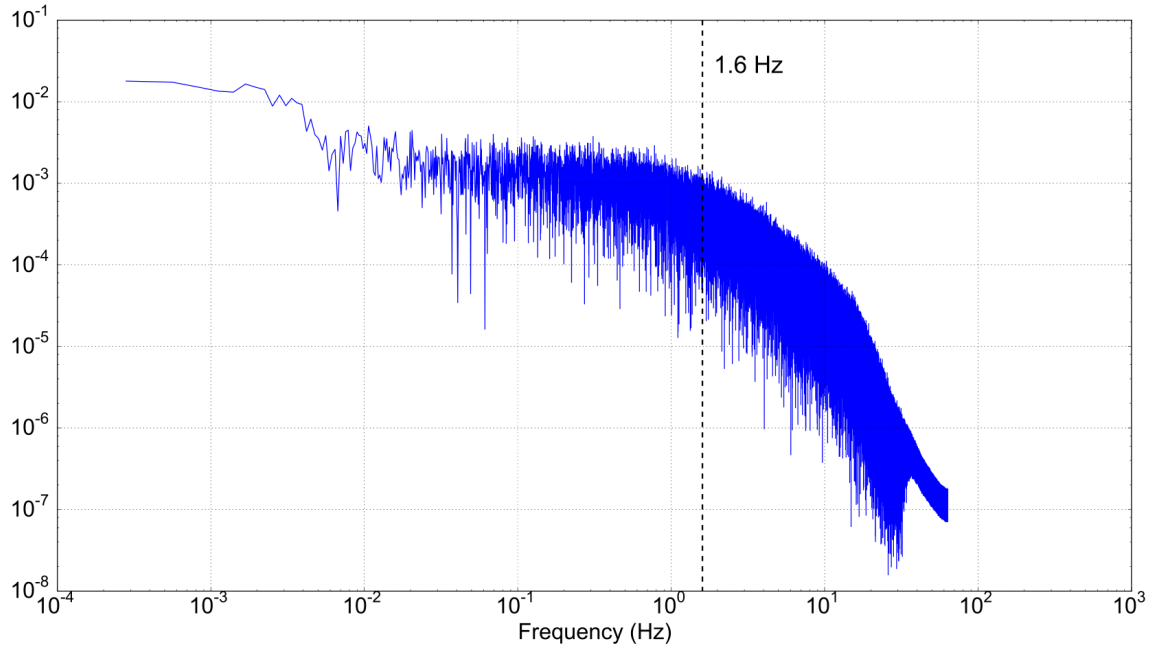


**Figure 4.6:** Non-dimensional rms temperature profile on cavity mid-plane at 50% cavity height

peaks close to the walls at 8% and 92% of the cavity width. These peaks in temperature fluctuation correspond to the peaks in vertical velocity magnitude.

The discrepancy in the amplitude of temperature fluctuations could be down to the sampling rates used for the DNS and experiment. The DNS used a non-dimensional time-step of  $5 \times 10^{-3}$  for numerical stability reasons, this equates to a time-step of 1.715 ms or a sample rate of 583 Hz. In the experimental temperature data collection, 2048 samples were recorded in 128 s [6], which equates to a sample rate of 16 Hz. As a rule of thumb a sample rate should be at least ten times the signal frequency in order to resolve the signal amplitude [74]. This is higher rate than the Nyquist criterion of twice the highest signal frequency as the requirement is to resolve signal amplitude and not only frequency. This means that the DNS can resolve signal amplitudes up to 58.3 Hz and the experiment can resolve signal amplitudes up to 1.6 Hz. The thermocouple in the experiment will also be acting as a low pass filter to reduce the amplitude of the higher frequency fluctuations.

The flow history from the DNS simulation was monitored at a point in the cavity mid plane at 50% height and 8% of the cavity width, corresponding to the location of the peak in rms temperature fluctuation. A frequency spectrum plot of the temperature data from



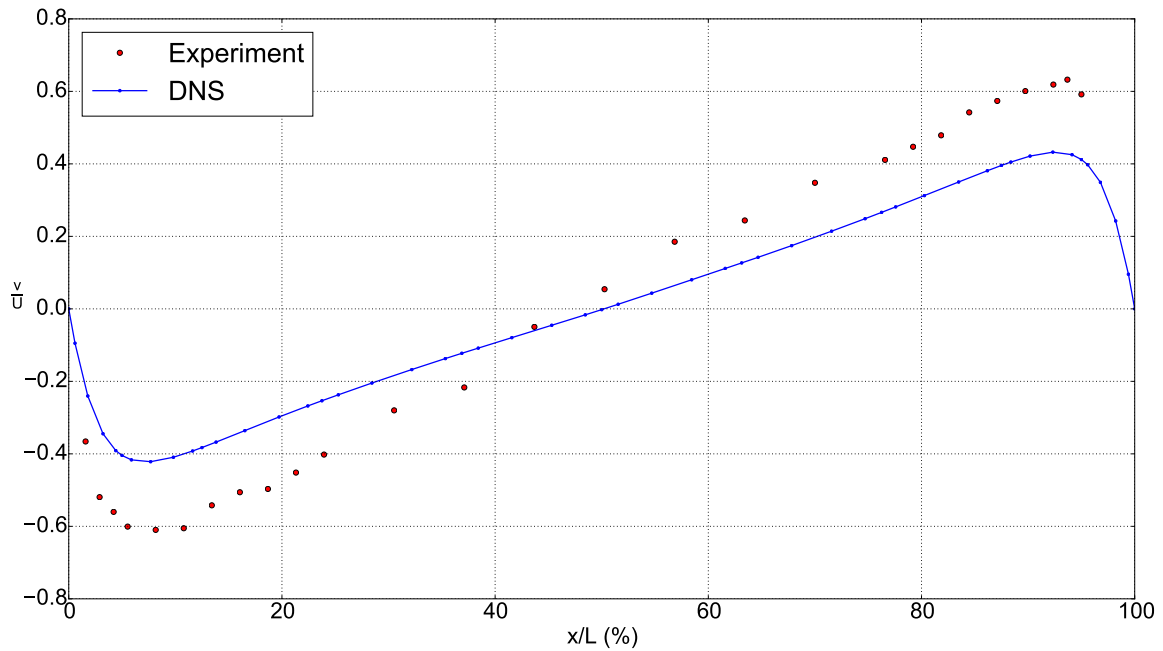
**Figure 4.7:** Spectrum plot of DNS temperature signal at a point on the cavity mid plane at 50% height and 8% of the cavity width

this point is shown in Fig.4.7. It can be seen that there is significant energy present in the temperature signal at frequencies above 1.6 Hz. This could be the cause of the lower value of rms temperature fluctuations found in the experiment. As the higher frequency fluctuations were not captured by the experiment, this could give the lower measured rms temperature.

At the point on the cavity mid plane at 50% height and 8% of the cavity width, the DNS data was low pass filtered to represent the low sampling rate and thermocouple response time of the experimental data. A 1st order 0.1 Hz filter was found to give a non-dimensional rms temperature fluctuation of 0.05, the same as the experimental measurements.

### Velocity validation

Experimental data is also available for mean and rms vertical velocity profiles at several cavity heights. Comparing the DNS to the experimental data for the mean velocity at 50% cavity height in Fig.4.8, shows good agreement between the two datasets. The biggest difference between the two datasets occurs at the peak velocities close to the heated walls, however this difference is only around 0.04 m/s. This is likely to be within the measurement



**Figure 4.8:** Non-dimensional mean vertical velocity profile on cavity mid-plane at 50% cavity height

error of the LDA system, however the measurement error was not quoted.

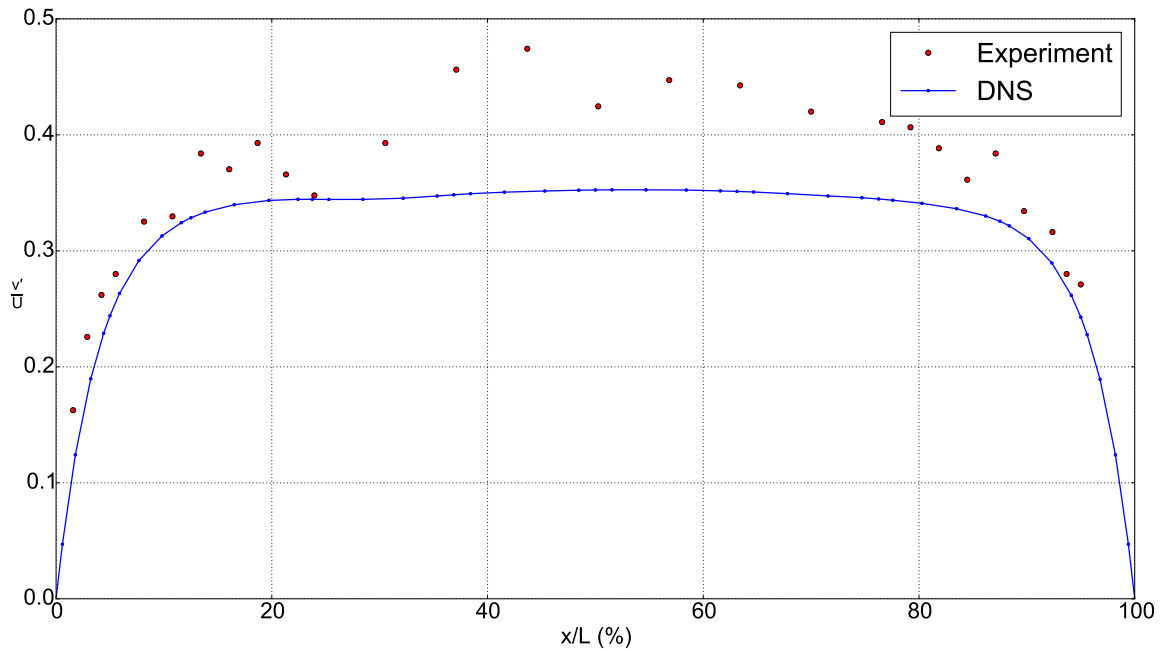
Experimental profiles are also available for rms vertical velocity fluctuations. This data is compared to the DNS data at 50% cavity height in Fig.4.9, and it can be seen that there is good agreement between the two datasets. The maximum difference between the datasets occurs in the centre of the cavity, and is around 0.02 m/s. This difference is again likely to be within the experimental error margin.

The combination of numerical validation and validation against experimental data gives confidence in the DNS that is correctly predicting the tall cavity flowfield. The DNS can then be used to obtain data such as the turbulent heat fluxes that are not available experimentally.

## 4.4 Tall Cavity Flowfield

### 4.4.1 Mean Flowfield

The flowfield inside of the cavity is driven by buoyancy forces. The buoyancy forces set up a recirculating flowfield where the flow is heated and rises upwards at the hot wall and

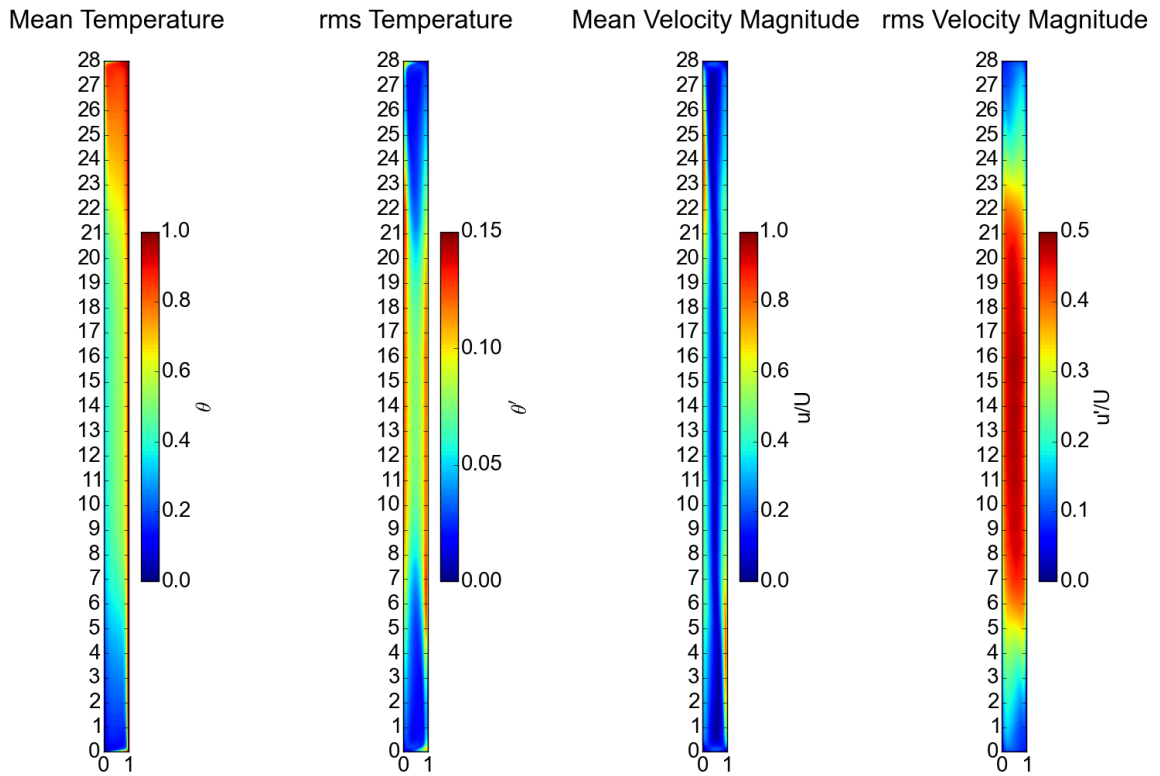


**Figure 4.9:** Non-dimensional rms vertical velocity profile on cavity mid-plane at 50% cavity height

is cooled and sinks down at the cold wall.

It can be seen by looking at the mean velocity in Fig.4.10, that the bulk of the flow movement is close to the hot and cold walls, with the peak velocities occurring at the bottom of the hot wall and the top of the cold wall. In the region at the centre of the cavity the mean velocity is very low. Thermal stratification occurs in the cavity with hot fluid collecting at the top of the cavity and cold fluid towards the bottom. These thermally stratified regions act to slow down the rising hot fluid as it approaches the top of the cavity and similarly with the cold fluid at the bottom of the cavity.

The rms temperature and velocity contours are also shown in Fig.4.10. Despite the mean velocity in the centre of the cavity being almost zero, the rms velocity fluctuations in the centre are relatively large. Similarly the mean temperature in the centre of the cavity is relatively uniform with a non dimensional temperature of 0.5, yet there are significant rms temperature fluctuations in this region. These regions of large fluctuations suggests that the flow is more complex than a simple recirculating flow.

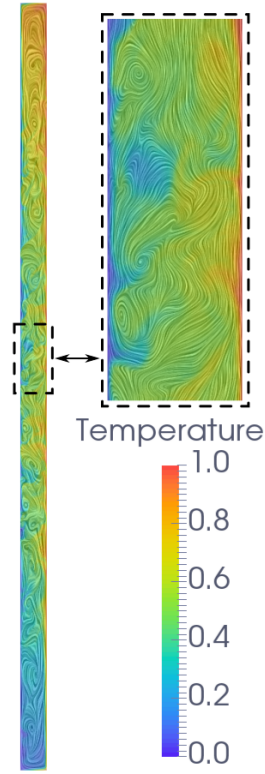


**Figure 4.10:** Mean and rms temperature and velocity at cavity mid-plane

#### 4.4.2 Instantaneous Flowfield

The instantaneous velocity and temperature field is shown in Fig.4.11. The velocity field is shown using a Line Integral Convolution (LIC) technique and coloured by temperature. The flow inside of the cavity can be seen to contain lots of turbulent eddies which extend into the centre of the cavity. It is these eddies that are responsible for causing the fluctuations in temperature and velocity seen in the centre of the cavity. These turbulent eddies average out over time to give the simple mean flowfield found in Fig.4.10.

The turbulent structures inside of the cavity can also be visualised using iso-surfaces of Q-criterion, shown in Fig.4.12. Only the bottom half of the cavity is shown due to the symmetry of the case. The central core of the cavity can be seen to contain a large number of turbulent structures. There is a region located at the bottom of the heated wall where the turbulent structures are not visible. This is where the boundary layer starts growing upwards along the heated wall. A similar low turbulence region is also visible at the top of the cold wall. The flow in these regions is laminar and transitions to turbulence as it flows



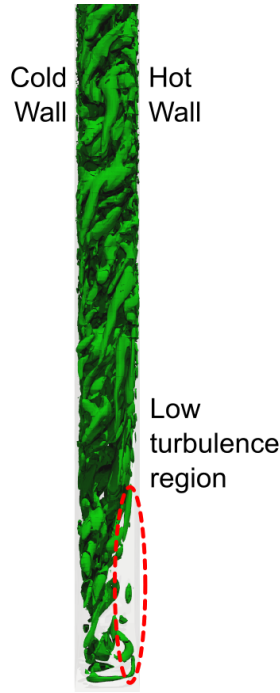
**Figure 4.11:** Instantaneous temperature and velocity at cavity mid-plane

along the wall.

It is these turbulent structures that require an eddy resolving technique such as DNS or LES in order to be captured. The turbulent fluctuations caused by these structures are modelled in RANS simulations and the choice of turbulence model can therefore have a significant impact on the accuracy of the numerical solution.

#### 4.4.3 Wall Heat Transfer

The non-dimensional heat transfer on the hot wall of the tall cavity is shown in Fig.4.13. The Nusselt number is defined as  $Nu = \frac{\partial \theta}{\partial x} \frac{L}{(\theta_{hot} - \theta_{cold})}$ . Towards the bottom of the hot wall is the laminar region, this can be seen as the region with more uniform heat transfer. This region also corresponds to the region with no turbulent structures shown in Fig.4.12. Turbulent streaks are visible in this region, which is the start of the transition of the flow into a turbulent boundary layer flow. The turbulent flow can be seen by its effect on the wall heat transfer, with the characteristic peaks and troughs of heat transfer indicative of



**Figure 4.12:** Iso-surface of Q-criterion in bottom half of the cavity

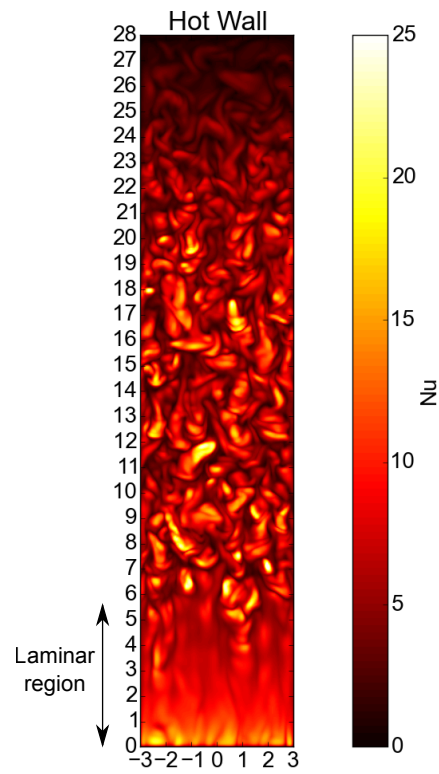
turbulent flow. Towards the top of the hot wall and the bottom of the cold wall, the thermal stratification causes the flow velocity to reduce. This reduction in velocity also causes a reduction in the wall heat transfer, which can be seen at the top of the cavity.

## 4.5 Turbulent Heat Flux

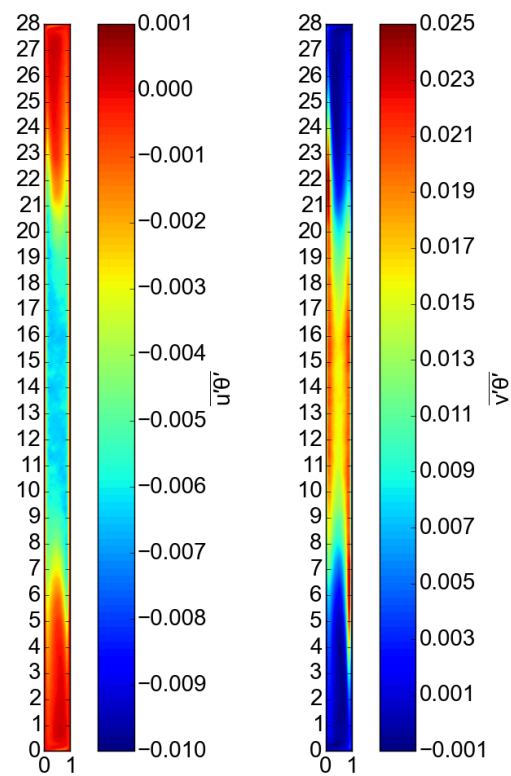
The turbulent heat flux terms  $\overline{(u'_i \theta')}$  are difficult to measure, as they require a simultaneous measurement of temperature and velocity fluctuation at the same location. Experimentally measured turbulent heat fluxes are therefore not available for the tall cavity case, but the turbulent heat fluxes can be obtained from the validated DNS data. Contours in the cavity mid-plane of the horizontal and vertical components of the turbulent heat flux are shown in Fig.4.14. It can be seen the largest values of turbulent heat fluxes occur around the cavity mid height, this is where the largest rms temperature and velocity fluctuations occur.

The assumption made by the SGDH turbulent heat flux model is that the turbulent heat flux term is proportional to the mean temperature gradient. The mean horizontal and vertical temperature gradients from the DNS data are shown on the cavity mid-plane in

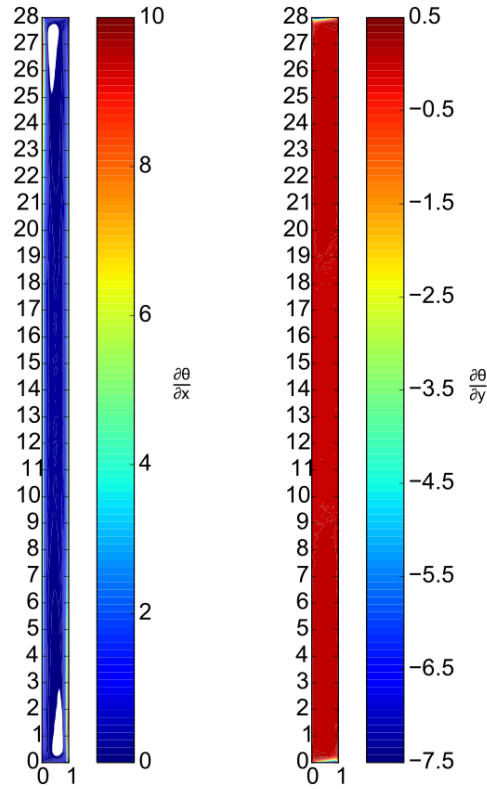




**Figure 4.13:** Contours of Nusselt Number on cavity hot wall



**Figure 4.14:** Contours of non-dimensional turbulent heat flux on cavity mid-plane



**Figure 4.15:** Temperature gradients on cavity mid-plane

Fig.4.15. The horizontal temperature gradient is close to zero in the centre of the cavity with the larger gradient occurring closer to the hot and cold walls. If the assumption that the turbulent heat flux is proportional to the mean temperature gradient is used, then this would give low values for  $\overline{u'\theta'}$  in the centre of the cavity. It was seen in Fig.4.14 however that the  $\overline{u'\theta'}$  turbulent heat flux has the largest magnitude in the centre of the cavity. Similarly the vertical temperature gradient is small except at the very top and bottom of the cavity. The vertical turbulent heat flux  $\overline{v'\theta'}$  however has the largest values at the cavity mid-height with the value reducing towards the cavity top and bottom. This shows that the assumption that the turbulent heat flux is proportional to the mean temperature gradient is not valid for this cavity flow.

The resolved turbulent heat fluxes from the DNS simulation can now be used to help validate turbulent heat flux RANS models.

## 4.6 Conclusions

There are very few experimental validation cases for natural convection where the turbulent heat fluxes ( $\overline{u'T'}$ ) were measured. A simulation was therefore performed in order to calculate turbulent heat fluxes for a natural convection test case. The test case chosen was the differentially heated tall cavity investigated by Betts and Bokhari [6]. This case was chosen as both mean and rms data for the temperature and vertical velocity was available at various locations inside of the cavity. The temperature and velocity data was not measured simultaneously and so the turbulent heat flux terms are not available but the individual rms components could be used to validate the simulation.

A Direct Numerical Simulation (DNS) study was conducted to simulate the cavity flowfield. A DNS simulation was chosen over a computationally cheaper method in order to resolve rather than model the turbulent statistics.

The DNS results were used to visualise the flowfield inside of the cavity, it was found that the centre of the cavity contained lots of turbulent eddies which were responsible for creating the turbulent fluctuations in the cavity. The wall heat transfer was also visualised and it was possible to identify the laminar and turbulent regimes of the wall boundary layer.

Comparisons of the DNS data with the experimental data gave good agreement for the mean temperature and velocity. It was found that the DNS predicted a higher level of temperature fluctuation than the experimental data. This was likely down to the contribution of the higher frequency temperature fluctuations that were captured in the DNS but not the experimental data.

The turbulent heat flux terms were then extracted from the DNS study. It was found that the turbulent heat fluxes were not proportional to the mean temperature gradient as is sometimes assumed. These turbulent heat flux terms can now be used for validation of turbulent heat flux RANS models.

## Chapter 5

# Turbulent Heat Flux Modelling

*In order to improve predictions of natural convection heat transfer using RANS simulations, more advanced turbulent heat flux models were implemented into ANSYS Fluent. The RANS simulations were validated using data from three test cases, the heated vertical plate, the differentially heated tall cavity and the gas turbine casing rig.*

*For each of the three validation cases, results from RANS simulations were compared to the validation datasets. It was found that the baseline SGDh model consistently under-predicted the heat transfer by more than 20%, making this model unsuitable for engineering design calculations. The implemented GGDH and AFM models were found to perform better than the baseline, giving errors in wall heat transfer of around 10%. This places the models within the target of 20% error and therefore makes the models suitable for engineering design use. The results from the GGDH and AFM models were similar to each other, meaning the extra complexity, cost and reduced numerical stability of the AFM model over the GGDH model hasn't led to a significant improvement in heat transfer predictions.*

*A modification to the GGDH model was developed to improve heat transfer predictions in regions of stratified flow. This GGDH+ model gave improved predictions in the top half of the casing rig test case.*

*It has been concluded that the GGDH model is the best choice for engineering design calculations. It is giving improved predictions over the SGDh model but without the complexity of the AFM model. The GGDH+ model may also help for flows featuring thermal stratification.*

## 5.1 Implementation

The new turbulent heat flux models were implemented in Fluent using User Defined Functions (UDFs). The first step was to suppress the internal calculation of the turbulent heat flux which uses the built in SGDH model. With the SGDH model  $\overline{u'_i T'} \propto 1/Pr_t$ , so setting the turbulent Prandtl number to a large value will set the internally calculated turbulent heat fluxes to zero.

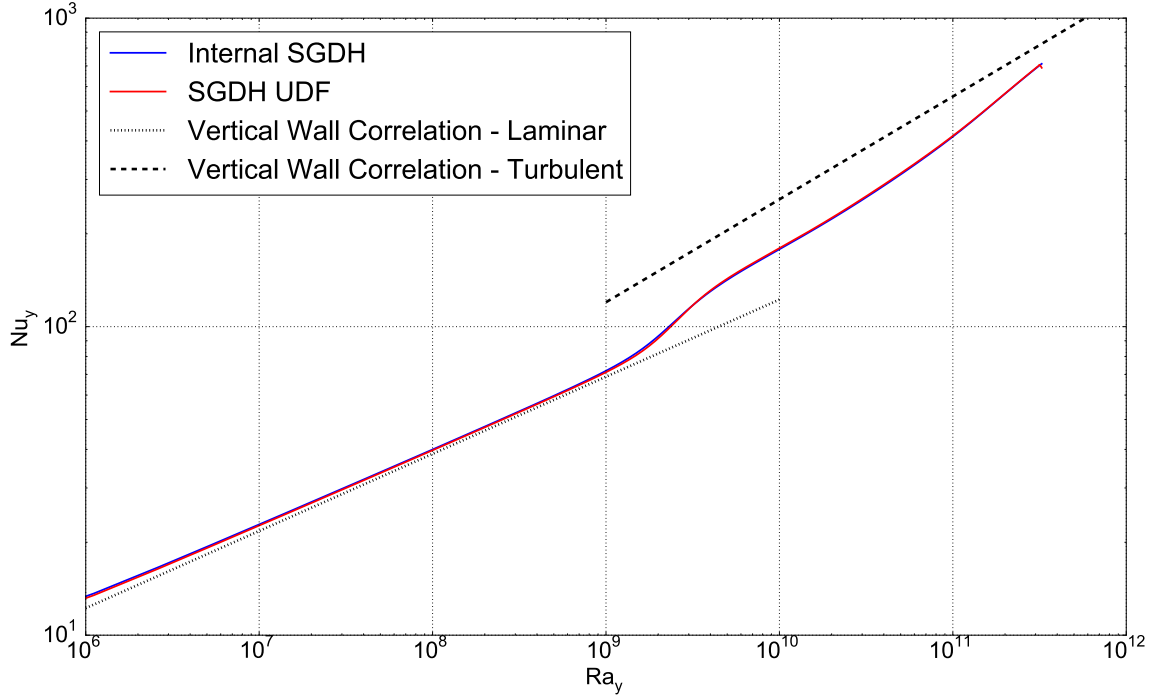
The new turbulent heat flux term can then added back into the energy equation. If the Navier Stokes energy equation is written as in Equation.5.1, the turbulent heat flux term  $\frac{\partial(\overline{u'_i T'})}{\partial x_i}$  can be treated as an energy source term. This source term is then added into Fluent using using a UDF source term function.

$$\frac{\partial \overline{T}}{\partial t} + \frac{\partial \overline{U_i T}}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \alpha \frac{\partial \overline{T}}{\partial x_i} \right) - \frac{\partial (\overline{u'_i T'})}{\partial x_i} \quad (5.1)$$

This method of implementing new turbulent heat flux models using UDF was then validated. An unmodified Fluent case uses the SGDH turbulent heat flux model, therefore a case using the SGDH model implemented using a UDF should give the same result. A SGDH model was then implemented using a UDF and the results compared to an unmodified baseline case. The results from a vertical plate test case are shown in Fig.5.1, comparing the heat transfer from the baseline to a case using a UDF. The results from the two cases are almost identical, validating this method of implementing turbulent heat flux models in Fluent. The new GGDH and AFM turbulent heat flux models could then be implemented using the same method.

The GGDH model could be closed using the terms from the mean flow and turbulence model, but the AFM model required a temperature variance term in order to be closed. A scalar transport equation for the temperature variance term  $(\overline{T'^2})$  was implemented into Fluent using a user defined scalar (UDS).

The temperature variance transport equation from Kenjeres and Hanjalic [38] is given by Equation.5.2 . Where  $D_{T'}$  is the diffusion term and  $P_{T'}$  and  $\epsilon_{T'}$  are the production and dissipation terms respectively. The empirical constant  $C_{T'} = 0.09$  and  $f_\mu$  is a damping function given by  $f_\mu = \exp(-3.4/(1 + Re_t/50)^2)$ , where  $Re_t$  is the turbulent Reynolds



**Figure 5.1:** Vertical plate heat transfer using unmodified internal SGD model and UDF model

Number,  $Re_t = \frac{k^2}{\nu \epsilon}$ .

$$\frac{D(\rho \overline{T'^2})}{Dt} = D_{T'} + 2\rho(P_{T'} - \epsilon_{T'}) \quad (5.2)$$

$$D_{T'} = \frac{\partial}{\partial x_j} \left( C_{T'} f_\mu \rho \frac{k^2}{\epsilon} \frac{\partial \overline{T'^2}}{\partial x_j} + \mu \frac{\partial \overline{T'^2}}{\partial x_j} \right) \quad (5.3)$$

$$P_{T'} = -\overline{u'_j T'} \frac{\partial T}{\partial x_j} \quad (5.4)$$

$$\epsilon_{T'} = \frac{\epsilon \overline{T'^2}}{2kR} \quad (5.5)$$

In order to maintain numerical stability when using the new turbulent heat flux models, several techniques were used. Under-relaxation of the energy source term was used and was found to improve stability. Under-relaxation factors were in the range 0.01 to 0.1, the lower factors increased stability at the cost of the increased number of iterations required

to reach convergence. Another technique used was to set bounding values, the temperature variance term was bounded to make sure it could not go negative.

## 5.2 Vertical plate validation case

The case of natural convection over a heated vertical plate has been studied by several authors including, Siebers et. al. [78] and Tsuji and Nagano [97]. The heated plate is useful as a validation case as it is one of the few test cases available in the literature that operates at a Rayleigh Number of the same order as an industrial gas turbine ( $10^{10}$ ).

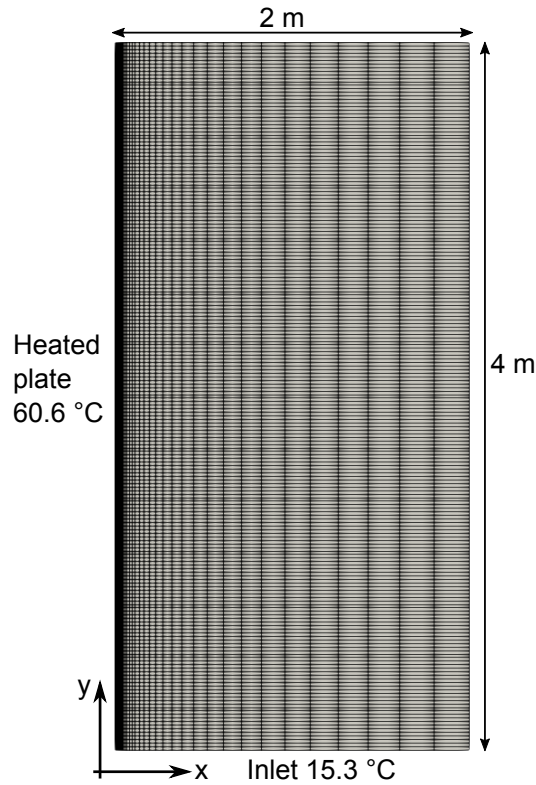
For validation of the RANS simulations, correlations for the plate heat transfer are available as well as correlations for thermal boundary layer profiles. Experimentally measured values for the turbulent heat flux terms are also available, which is unusual for a natural convection test case.

### 5.2.1 Numerical method

Numerical simulations were performed on the vertical plate test case using ANSYS Fluent v15.0. Steady RANS simulations were conducted in order to represent the type of simulations commonly used in industry. To obtain a baseline a simulation was conducted without any UDF and therefore used the SGDH turbulent heat flux model. Further simulations were run using the GGDH and the AFM turbulent heat flux models implemented using UDF.

The computational mesh used for the vertical wall case is shown in Fig.5.2. The domain was  $4 \times 1 \times 2$  m in the height, width and depth from the plate. This was to match the  $4 \times 1$  m heated plate used by Tsuji and Nagano [97]. The mesh depth of 2 m kept the ratio of boundary layer thickness to mesh depth less than 0.1 at the top of the plate. This was to avoid the side-wall boundary condition affecting the boundary layer growth. The mesh consisted of  $5 \times 10^5$  nodes with the spacing being biased towards the heated plate in order to keep the wall  $y^+ < 1$ .

The boundary condition for the heated plate was set at a constant temperature of  $60.6^\circ\text{C}$  to match the Tsuji and Nagano data [97]. The bottom of the domain used a pressure inlet boundary condition at ambient pressure and  $15.3^\circ\text{C}$ , with the outlet at the top of the domain



**Figure 5.2:** Cross section of vertical plate computational mesh

also being at ambient pressure. The side-wall parallel to the heated plate used a symmetry or inviscid-wall flow boundary condition and was adiabatic. The front and back walls were set as a periodic boundary condition.

The numerical scheme used was a coupled pressure velocity scheme with second order upwind spatial discretization. The turbulence model used was Menter's  $k - \omega SST$  model [55].

In order to judge if the solution had converged a surface monitor was created on the heated plate. The heat flux integrated over the plate was monitored at each iteration. When the value stopped changing with subsequent iterations it was judged that the plate heat flux and hence the thermal boundary layer had reached a converged solution. As a secondary check for convergence the error in the global energy balance was checked, the solution was judged to have converged if the imbalance was less than 0.1%.



## 5.2.2 Results & Discussion

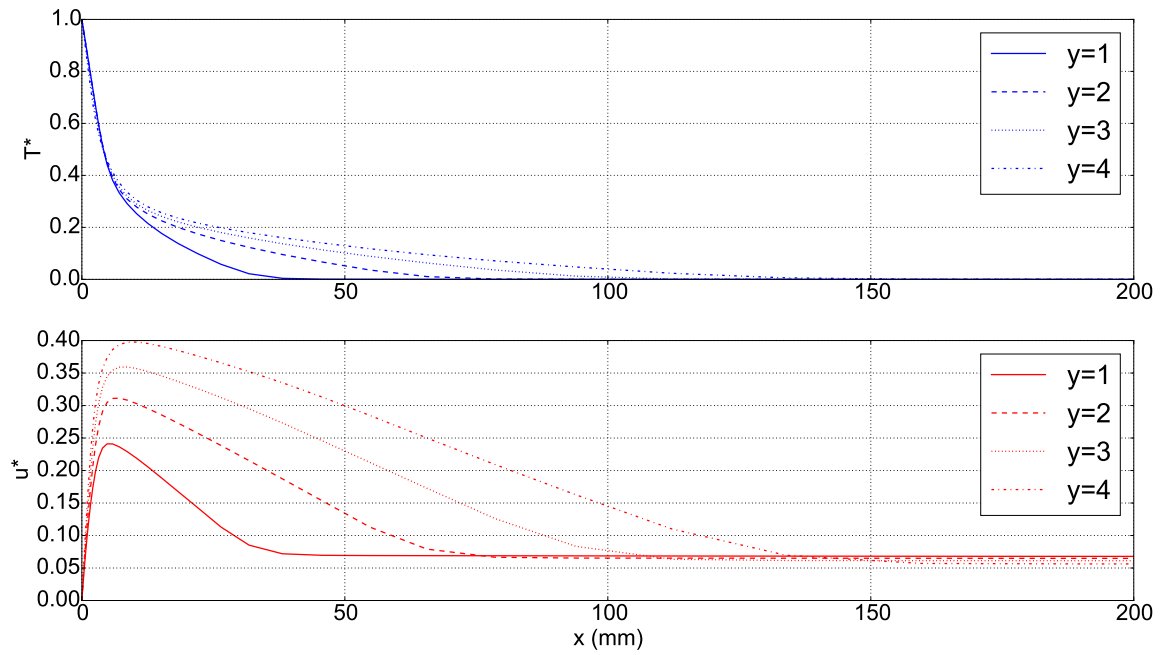
### Flowfield

The flowfield in the vertical plate case is that of a boundary layer growing as it flows upwards along the heated plate. Both momentum and thermal boundary layers develop along the plate, this can be seen in the temperature and velocity profiles from a RANS simulation in Fig.5.3. Profiles are given at distances 1,2,3 & 4 m from the bottom of the heated plate, these distances correspond to Rayleigh Numbers of  $Ra_y = 5.1 \times 10^9$ ,  $4.1 \times 10^{10}$ ,  $1.4 \times 10^{11}$  &  $3.3 \times 10^{11}$  respectively.

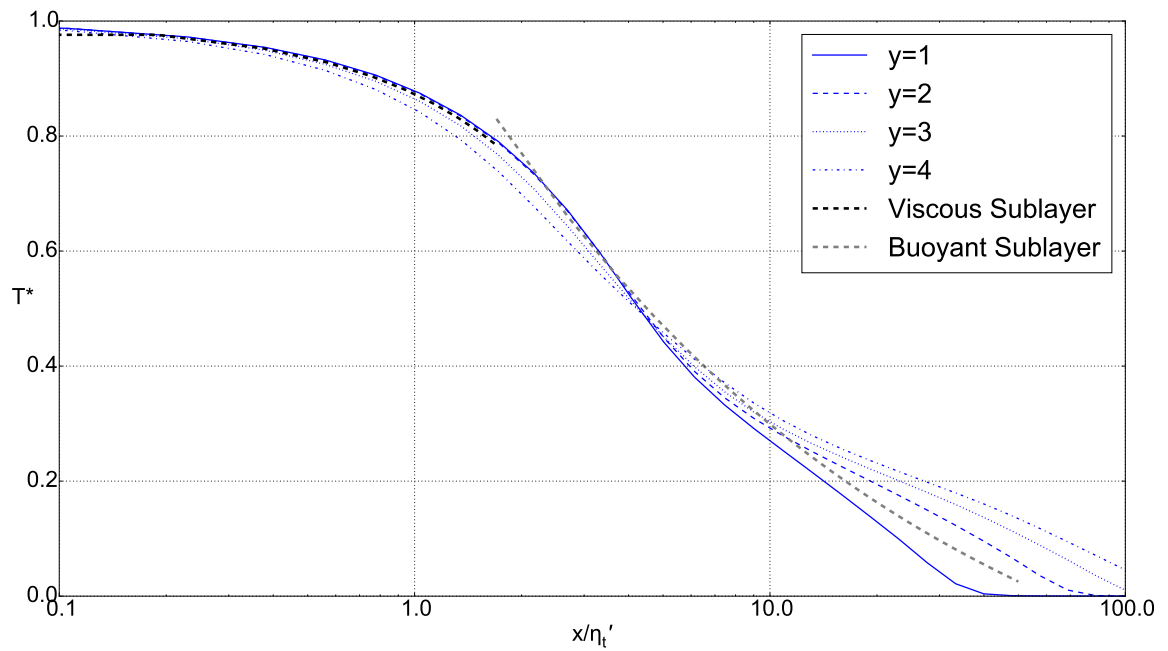
The temperature profiles have been non-dimensionalised as  $T^* = (T - T_\infty)/(T_{plate} - T_\infty)$ . It can be seen that the temperature of the air starts off at the plate temperature before decaying to the free stream temperature. The initial decay in temperature away from the plate can be seen to be linear. This is the constant heat flux region of the boundary layer as described by George and Capp [35], as  $\frac{\partial T}{\partial x}$  is a constant and  $\dot{q} = -\lambda \frac{\partial T}{\partial x}$ .

A length scale for the inner region of a turbulent natural convection thermal boundary layer was proposed by George and Capp [35] and modified by Siebers et. al. [78]. The length scale  $\eta'_t = \eta_t \left( \frac{\lambda_{plate}}{\lambda_\infty} \right) \left( \frac{T_{plate}}{T_\infty} \right)^{0.14}$  where  $\eta_t = (\alpha^2/g\beta(T_{plate} - T_\infty))^{1/3}$ , and  $\lambda_{plate}$  and  $\lambda_\infty$  are the thermal conductivity of the air at the plate and free stream respectively. The inner region of the thermal boundary layer was split into a viscous sublayer and a buoyant sublayer with temperature correlations given for each. The temperature profiles from a RANS simulation are compared to the correlations in Fig.5.4. It can be seen that the results from the simulation are very good compared to the experimental correlation for both the viscous sublayer and the buoyant sublayer.

The velocity profiles are also shown in Fig.5.3, the velocity has been non-dimensionalised by the buoyant velocity to give  $u^* = u/\sqrt{g\beta(T_{plate} - T_\infty)H}$  where  $H$  is the height of the heated plate. The velocity profile starts at zero at the plate before rising to a peak and then decaying back down to zero. As the flow rises up the plate the peak velocity can be seen to increase and move away from the plate as the boundary layer thickens.



**Figure 5.3:** Temperature and Velocity profiles at different heights (in m) from bottom of the heated plate



**Figure 5.4:** Inner region thermal boundary layer profiles

## Plate Heat Transfer

Profiles of heat transfer on the centreline of the heated plate are shown in Fig.5.5. The profiles calculated using the three turbulent heat flux models are compared to experimental correlations from Tsuji and Nagano [97]. The Nusselt Number ( $Nu_y$ ) and Rayleigh Number ( $Ra_y$ ) are defined using the coordinate  $y$ , the distance from the leading edge of the heated plate.

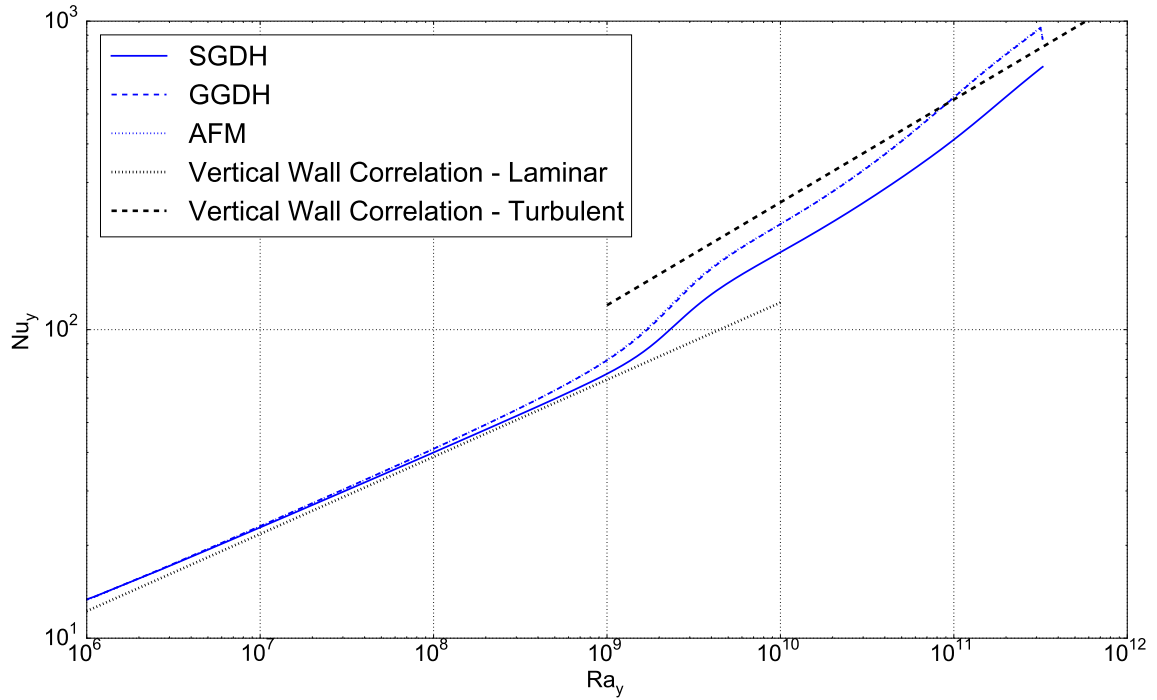
The boundary layer at the start of the heated plate is initially laminar and the heat transfer profile follows the trend  $Nu_y \propto Ra_y^{1/4}$ . It can be seen that in the laminar region all three models are giving similar results and correctly predicting the trend of heat transfer. The transition region occurs at Rayleigh Numbers between  $10^9$  and  $10^{10}$  and it can be seen that all three models have predicted transition occurring in this region.

The heat transfer in the vertical plate boundary layer increases after it transitions to turbulence. The heat transfer then follows the trend  $Nu_y \propto Ra_y^{1/3}$ . It can be seen that the simulation using the SGDH model has predicted a heat transfer trend that is the correct slope but the value of heat transfer is lower than the experimental correlation. This result was expected as it is known that the SGDH model cannot correctly predict the turbulent heat fluxes parallel to the heated plate. The GGDH and AFM turbulent heat flux models are also predicting the correct slope of the heat transfer profile but are giving improved predictions for the heat transfer compared to the correlations.

## Turbulent Heat Flux

Experimentally measured values of the turbulent heat flux terms  $\overline{u'T'}$  and  $\overline{v'T'}$  are compared to the values predicted by the turbulent heat flux models in Fig.5.6. The values are compared along a line normal to the plate at  $y=2.54$  m, equivalent to  $Ra_y = 8.33 \times 10^{10}$ .

The turbulent heat flux normal to the plate ( $\overline{u'T'}$ ) has been predicted well by all three turbulent heat flux models. The SGDH model is predicting a reasonable value for this term as the temperature gradient normal to the plate is significant, and the SGDH model assumes the turbulent heat flux is proportional to the temperature gradient. The GGDH and AFM models also have a similar term which is proportional to the temperature gradient and hence



**Figure 5.5:** Heat transfer from vertical plate case using different turbulent heat flux models

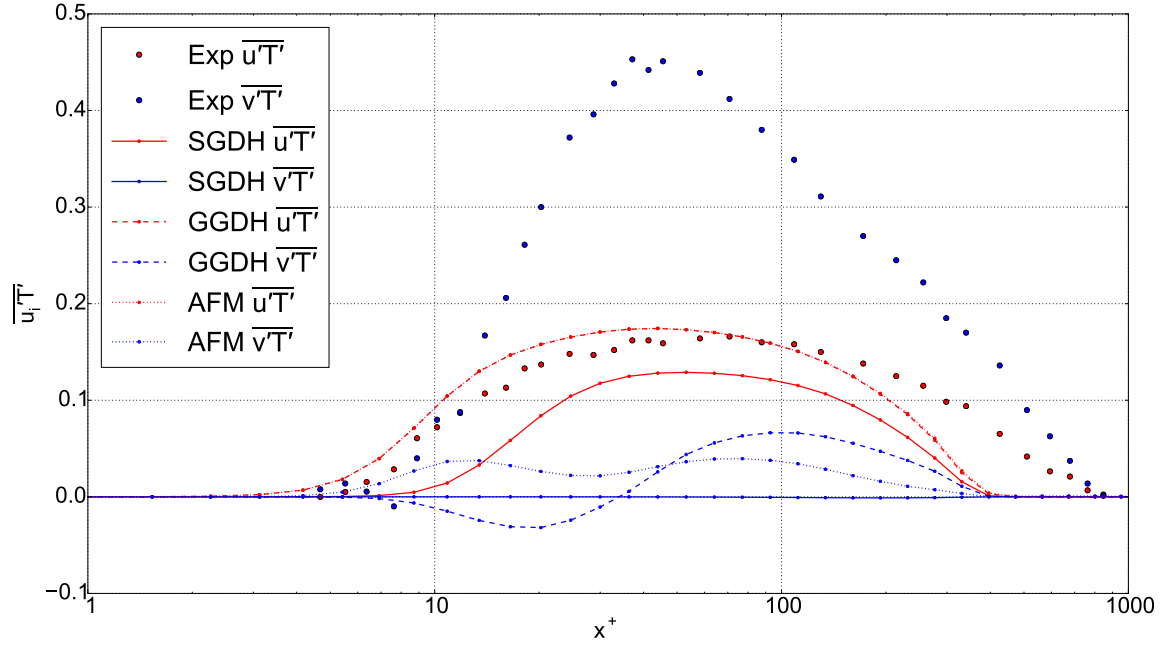
have also accurately predicted the turbulent heat flux.

However the turbulent heat flux term parallel to the plate ( $\overline{v'T'}$ ) is predicted to be almost zero by the SGDH model. This is due to the temperature gradient parallel to the plate being small and hence the predicted turbulent heat flux also being very small. The GGDH and AFM models have predicted non-zero values for the  $\overline{v'T'}$  turbulent heat flux but the magnitude of the predicted value is still lower than the experiment. This is a significant improvement in the prediction of the turbulent heat flux compared to the SGDH model and is responsible for the improved predictions of the plate heat transfer.

### 5.2.3 Vertical plate conclusions

Simulations were conducted on the heated vertical plate geometry using the baseline SGDH turbulent heat flux model as well as the implemented GGDH and AFM turbulent heat flux models. The flowfield in the heated plate case is that of a boundary layer growing as the flow rises up the plate, and the temperature profiles in the thermal boundary layer were shown to agree well with available experimental correlations.

The heat transfer from the plate was compared to experimental correlations for both the



**Figure 5.6:** Vertical plate turbulent heat fluxes profiles at  $y=2.54$  m

baseline case using the SGDH model and for the GGDH and AFM models. It was found that in the laminar region all three models agree well with the heat transfer correlation. In the turbulent region of the flow the models predicted the correct slope of the  $Nu_y - Ra_y$  plot but the SGDH under-predicted the value of the heat transfer. Whereas the GGDH and AFM models both gave very similar predictions to each other, which were closer to the experimental data.

The values of the turbulent heat flux predicted by the three models were compared to the experimentally measured values. For the wall normal turbulent heat flux ( $\overline{u'T'}$ ) it was found that all three models gave good predictions compared to the experimental data. For the turbulent heat flux parallel to the wall ( $\overline{v'T'}$ ) the values predicted by the SGDH model were almost zero, much lower than the experimental data. This is due to the temperature gradient parallel to the plate being very small and the SGDH assumes the turbulent heat flux is proportional to the temperature gradient. This very low prediction of the turbulent heat flux explains the low prediction of wall heat transfer. The GGDH and AFM models gave better predictions of the  $\overline{v'T'}$  turbulent heat flux compared to the SGDH model, but the predictions did not match the same peak as the experimental data. This improved prediction

of the turbulent heat flux is responsible for the improvement in the prediction of the plate heat transfer.

It was shown that the implemented turbulent heat flux models can give improved predictions of turbulent heat fluxes and wall heat transfer for a high Rayleigh Number natural convection flow.

## 5.3 Tall cavity validation case

The second validation case is the differentially heated tall cavity. This was studied by Betts and Bokhari [6], and was chosen as a validation case as a large dataset is available for both temperature and velocity. This has been supplemented by the results from the DNS simulation discussed in Chapter 4.

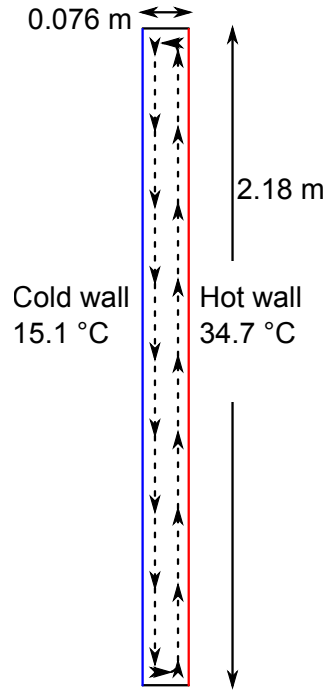
### 5.3.1 Numerical method

Steady RANS simulations were conducted using the tall cavity geometry. RANS simulations were chosen due to being commonly used in industry. A baseline case was run using the SGDH model which was compared to cases using the GGDH and AFM turbulent heat flux models.

The RANS simulations were conducted using ANSYS Fluent v15.0 using the pressure based solver, with the  $k - w$  SST turbulence model. The computational domain consisted of the whole cavity and was  $2.18 \times 0.076 \times 0.52$  m in the height, width and depth respectively, shown schematically in Fig.5.7. The mesh was a structured hexahedral type consisting of  $150 \times 50 \times 75$  nodes, with the nodes clustered towards to hot and cold walls to keep the  $y^+ < 1$ .

Wall boundary conditions were used on all six sides of the cavity with the no-slip velocity condition. The cold and hot walls used constant temperature boundary conditions of  $15.1^\circ\text{C}$  and  $34.7^\circ\text{C}$  respectively. The top and bottom walls had a temperature profile that varied linearly between the cold and hot temperatures and the front and back walls used adiabatic boundary conditions.

To ensure that the simulations had converged sufficiently a surface monitor on the hot



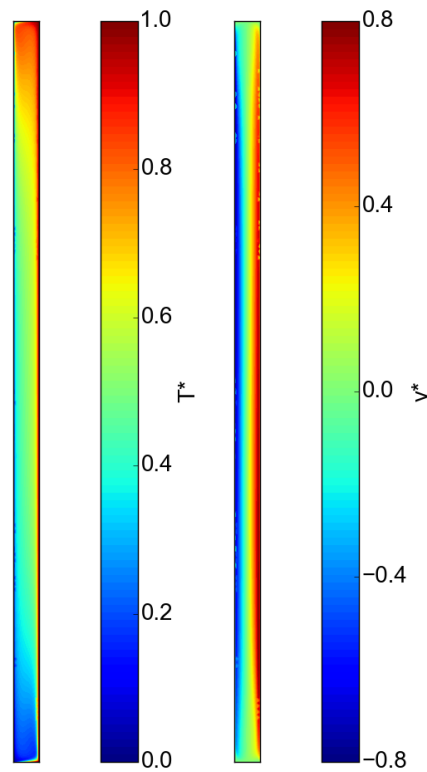
**Figure 5.7:** Schematic cross section of tall-cavity

wall was used. The wall heat flux was integrated over the surface and plotted at each iteration, when the value no longer changed the solution was deemed to have converged. The overall heat balance was also checked to make sure the global heat flux imbalance was less than 0.1%.

## 5.3.2 Results & Discussion

### Flowfield

The flowfield predicted by the steady RANS simulations is that of a recirculating flow inside of the cavity, this can be seen in Fig.5.8. The temperature has been non-dimensionalised as  $T^* = (T - T_c)/(T_h - T_c)$ , where  $T_c$  and  $T_h$  are the cold and hot wall temperatures. The vertical velocity has been non-dimensionalised by the buoyant velocity,  $v^* = v/\sqrt{g\beta(T_h - T_c)L}$  where the length-scale  $L$  is the cavity width. The steady RANS simulations were not able to resolve the turbulent eddies that were found in the DNS simulation, the effects of these eddies on the flow and heat transfer therefore have to be included in the turbulence model and the turbulent heat flux model.



**Figure 5.8:** Contours of temperature and vertical velocity on cavity mid-plane

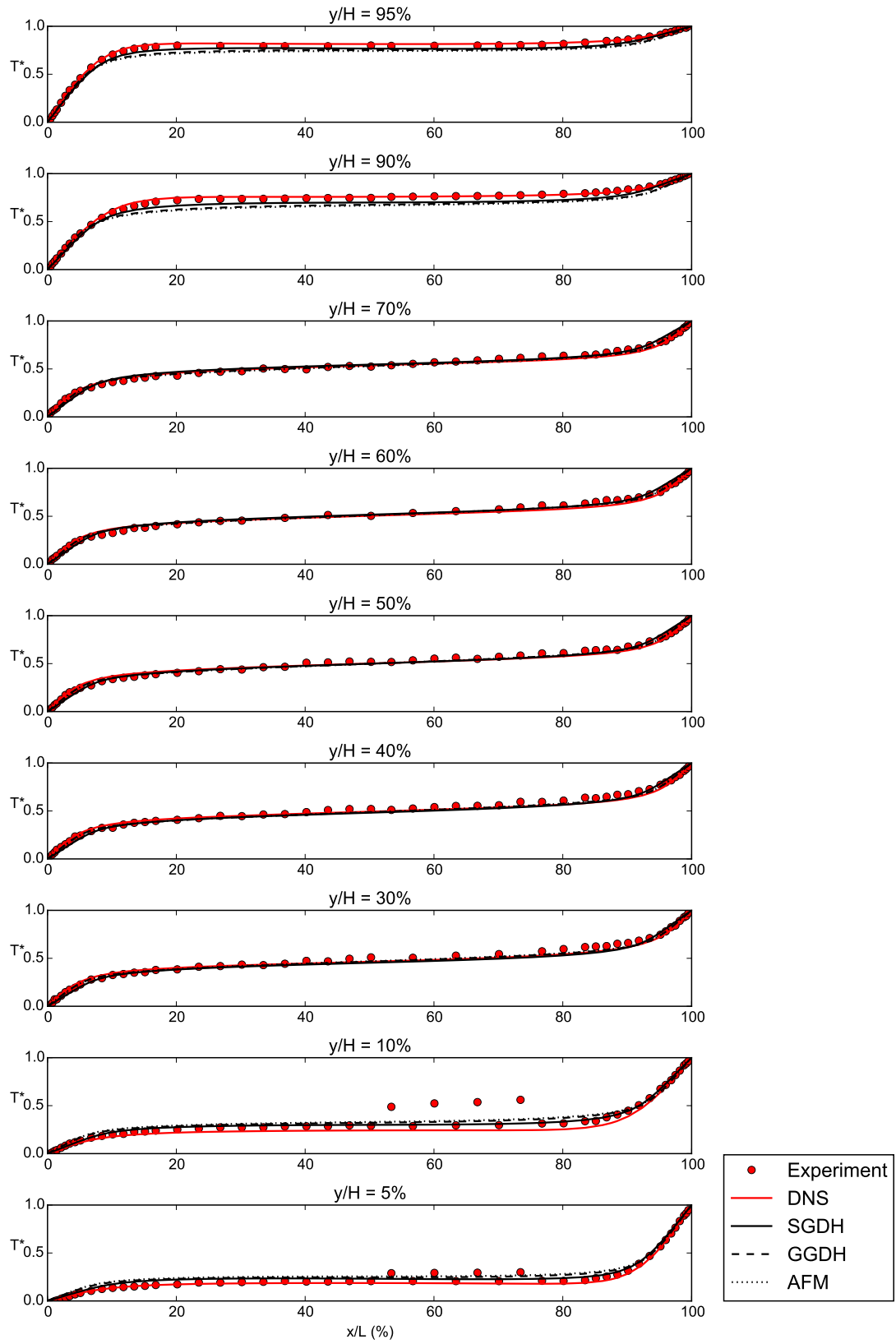
### Temperature profiles

Non-dimensional temperature profiles at nine cavity heights are shown in Fig.5.9. The results from the RANS simulations using the three turbulent heat flux models are compared to the experimental data and the results from the DNS simulation. It can be seen that there is good agreement between the temperature profiles at each of the cavity heights, meaning that the RANS simulations can predict the mean temperature profiles. This shows that the SGDH model can be used if only the general temperature field is required from an engineering calculation, as there is little benefit from the more complex turbulent heat flux models.

### Heat Transfer

The temperature profiles close to the cavity cold wall are shown in Fig.5.10, the temperature gradient next to the wall defines the heat transfer, an important result for the RANS simulations to predict. It can be seen that the differences in gradient are relatively small but





**Figure 5.9:** Temperature profiles on mid-plane of tall cavity at different vertical heights

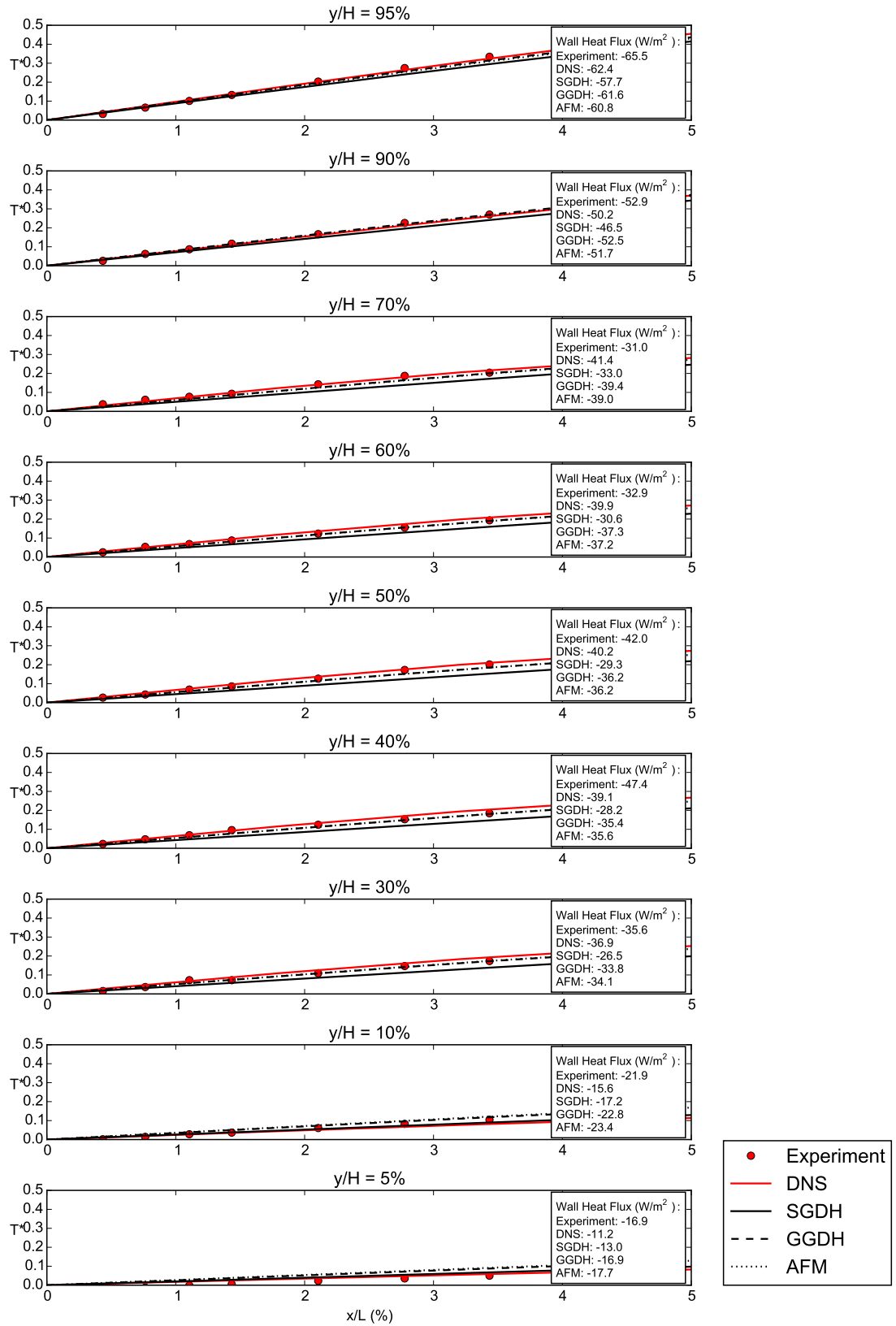
the temperature profile from the simulation using the SGDH model is consistently lower than the experiment and DNS. The temperature profiles from the simulations using the GGDH and AFM models have a near wall gradient that are closer to the experiment.

The differences in the wall heat transfer predicted by the RANS simulations compared to the DNS data are given in Table.5.1. The DNS dataset was chosen as the baseline dataset over the experimental data as the gradient calculated from the experimental data is subject to measurement errors, and small errors in the temperature measurement can have a significant effect on the temperature gradient. The DNS data gives a straight line for the temperature profile next to the wall, making the gradient calculation more reliable. It can be seen that at most of the locations the SGDH model is under-predicting the heat transfer by over 20% whereas the GGDH and AFM models are giving predictions that are within 10%, this accuracy is then sufficient for the results to be used for engineering design. The large difference between the DNS and RANS simulations at the bottom of the cavity is likely due to the lower values of heat transfer in this region, therefore small errors in the temperature gradient will have a relatively larger effect on the heat transfer.

The GGDH and AFM models are both giving very similar results to each other, this means that the additional complexity of the AFM model is not giving an additional benefit in heat transfer predictions. The AFM model is computationally more expensive than the SGDH and GGDH models, as it requires an equation to be solved for the temperature variance. The AFM model is also computationally less stable than the other models requiring lower relaxation factors. This would likely make the GGDH model the best compromise of complexity against accuracy for engineering calculations. The SGDH model could still be useful for engineering simulations, due to its numerical robustness low computational cost, for results where relative differences and trends in heat transfer are required.

## Velocity profiles

Vertical velocity profiles non-dimensionalised by the buoyant velocity  $\left(\sqrt{g\beta(T_h - T_c)L}\right)$  are shown at nine heights in the cavity in Fig.5.11. It can be seen that all three RANS simulations are giving good predictions for the mean velocity profiles. This shows that the more complex turbulent heat flux models are not needed if the general flowfield is all that



**Figure 5.10:** Near wall temperature profiles on mid-plane of tall cavity at different vertical heights

| y/H (%) | SGDH   | GGDH  | AFM   |
|---------|--------|-------|-------|
| 95      | -7.6%  | -1.4% | -2.6% |
| 90      | -7.2%  | 4.7%  | 3.2%  |
| 70      | -20.3% | -4.7% | -5.7% |
| 60      | -23.4% | -6.5% | -6.7% |
| 50      | -27.1% | -9.8% | -9.8% |
| 40      | -28.0% | -9.6% | -9.1% |
| 30      | -28.1% | -8.4% | -7.6% |
| 10      | 10.0%  | 45.6% | 49.9% |
| 5       | 15.5%  | 50.6% | 57.4% |

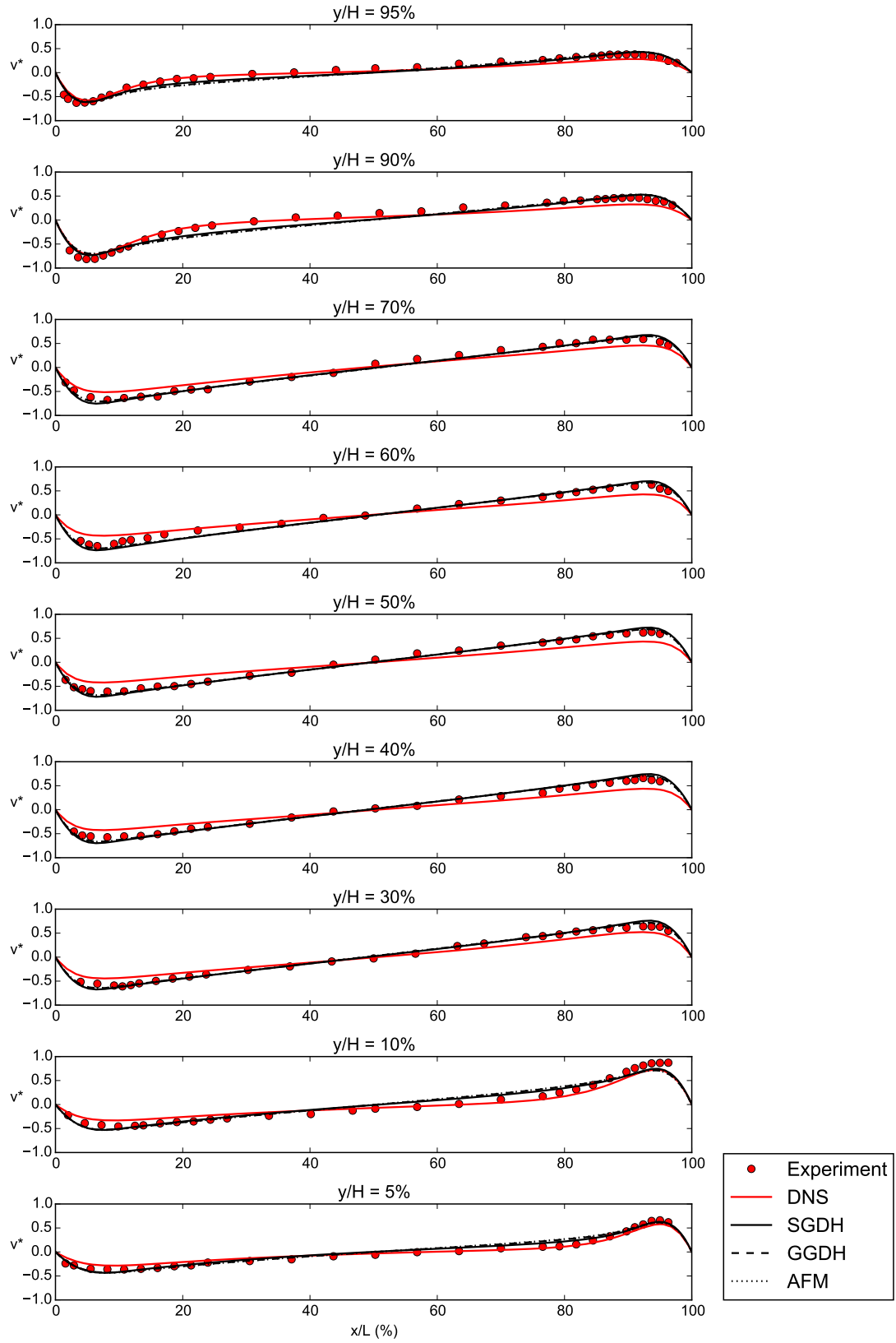
**Table 5.1:** Differences in predicted wall heat transfer from DNS data

is required from a simulation. This also shows that the implemented turbulent heat flux models are not having an adverse effect on the velocity field.

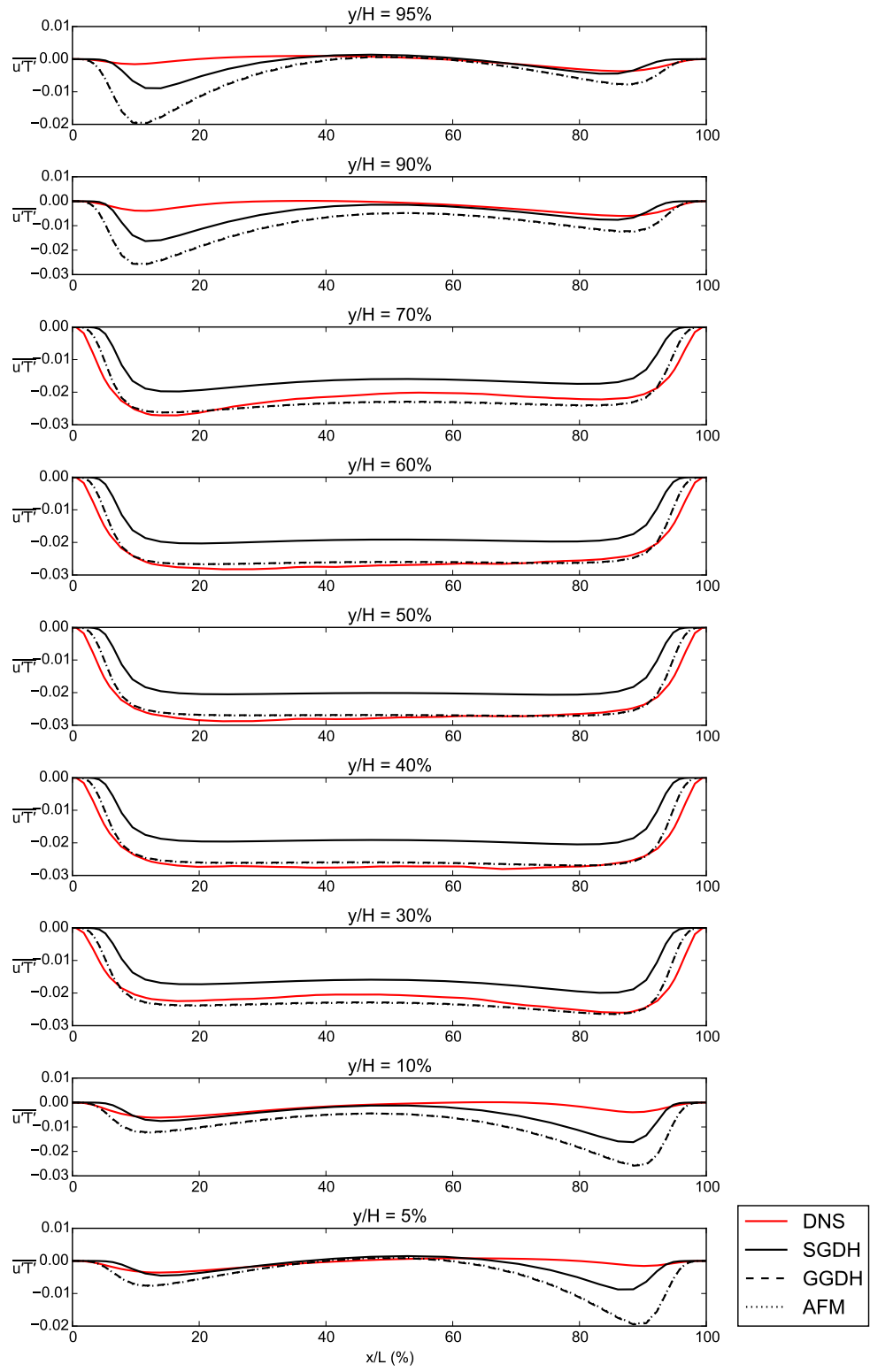
### Turbulent Heat Flux profiles

Values of the turbulent heat flux predicted by the different models are compared to resolved turbulent heat fluxes from the DNS dataset. Profiles of the  $\overline{u'T'}$  term are shown at nine cavity heights in Fig.5.12. In the profiles around the cavity mid-height it can be seen that the SGDH model is under-predicting the magnitude of the turbulent heat flux compared to the DNS dataset. The GGDH and AFM models however are giving predictions that are very close to the DNS data at locations around the cavity mid-height. The bottom of the hot wall and the top of the cold wall are the start of the thermal boundary layers, it can be seen in this region that all three models have predicted turbulent heat fluxes that have a greater magnitude than the DNS data.

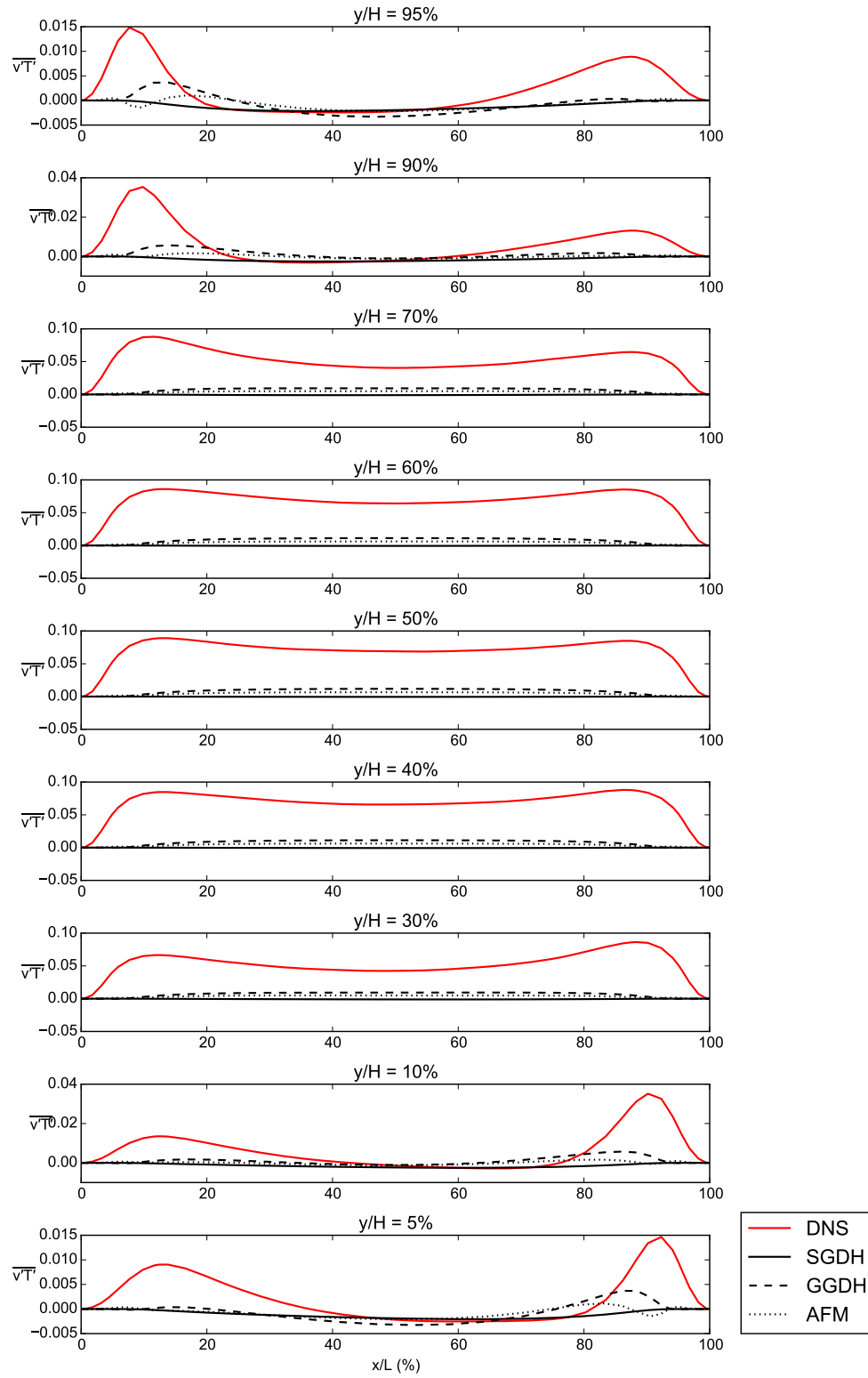
Profiles of the  $\overline{v'T'}$  turbulent heat flux term are shown in Fig.5.13. It can be seen that in all of the profiles the turbulent heat flux models have under-predicted the  $\overline{v'T'}$  term compared to the DNS data. The SGDH model has predicted almost zero for the  $\overline{v'T'}$  term as the vertical temperature gradient is relatively small compared to the temperature gradient across the cavity width. The GGDH and AFM models have predicted larger values for the term but this is still significantly lower than the DNS value. This under-prediction of the  $\overline{v'T'}$  term could be the reason why the RANS simulations are still giving values of wall heat transfer that are  $\approx 10\%$  low when compared to the DNS data.



**Figure 5.11:** Vertical velocity profiles on mid-plane of tall cavity at different vertical heights



**Figure 5.12:** Profiles of  $\overline{u'T'}$  turbulent heat flux in tall cavity



**Figure 5.13:** Profiles of  $\overline{v'T'}$  turbulent heat flux in tall cavity

It was found that for both the  $\overline{u'T'}$  and  $\overline{v'T'}$  terms the GGDH and AFM models are giving very similar results. The additional complexity and computational expense of the AFM model is not leading to a significant improvement in turbulent heat flux and hence heat transfer predictions. This would suggest that the GGDH model is a better choice for engineering calculations than the AFM model.

### 5.3.3 Tall cavity conclusions

Steady RANS simulations were conducted on the differentially heated tall cavity geometry using three different turbulent heat flux models. A baseline simulation used the SGDH turbulent heat flux model and was used for comparison against simulations using the GGDH and AFM turbulent heat flux models. The RANS simulations were validated with both experimental data and data from the DNS simulation detailed in Chapter.4.

It was found that the general temperature profile was well predicted by all three RANS simulations. The velocity profiles were also well predicted, showing that the implemented turbulent heat flux models were not having any adverse effect on the temperature and velocity fields.

The prediction of the wall heat transfer is often a key output of engineering simulations. The heat transfer predictions using the baseline SGDH model were found to be under predicted by often more than 20%, making them unsuitable for engineering design calculations. The heat transfer predictions using the implemented GGDH and AFM models however were found to reduce the under-prediction to less than 10%, which is acceptable for engineering design use.

The values of the turbulent heat flux predicted by the models were compared to resolved values from the DNS dataset. For the  $\overline{u'T'}$  term the GGDH and AFM were giving very similar results to the DNS dataset at the cavity mid-height, with the SGDH model under-predicting the magnitude of the term. For the  $\overline{v'T'}$  term however all three models were under predicting compared to the DNS data. The SGDH model prediction was almost zero for the  $\overline{v'T'}$  term, with the GGDH and AFM giving non-zero values but were still lower than the DNS. This under-prediction of the  $\overline{v'T'}$  term is likely to be the reason for the remaining under-prediction of the wall heat transfer.



## 5.4 Casing Rig validation case

The casing rig validation case provides data which is closest to that found in a gas turbine during shutdown. The rig uses representative geometry and operates at the same Rayleigh Number as a real engine. Temperature data from the rig will be combined with velocity and turbulent heat flux data from LES simulations in order to validate RANS simulations. The casing rig is a challenging test case for the RANS simulations as it features transition and the formation of streamwise vortices as well as buoyancy forces aiding and inhibiting turbulence.

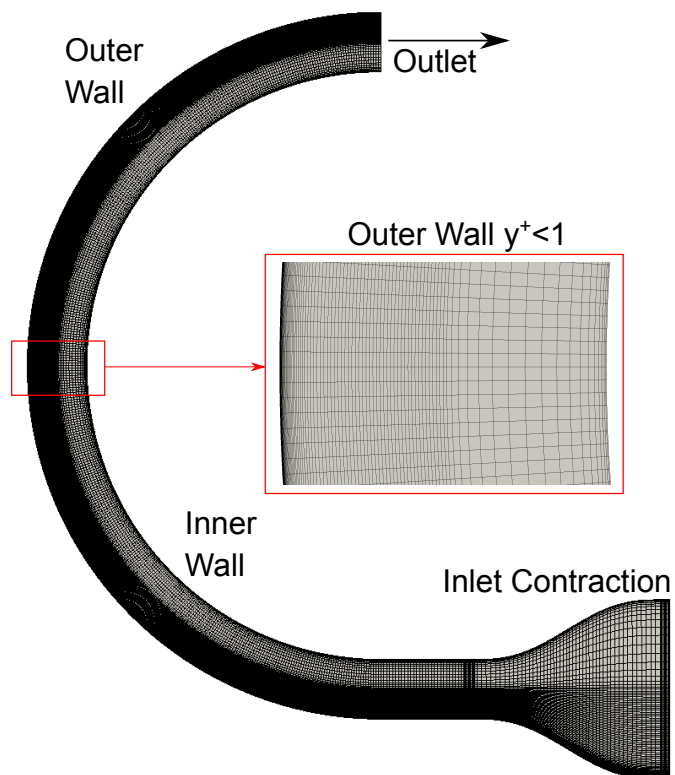
### 5.4.1 Numerical method

Steady RANS simulations were performed on the casing rig geometry using the three different turbulent heat flux models. The computational mesh used was the same as that previously used for the LES study of the rig, this allows a direct back to back comparison with the LES but is not representative of the coarser meshes that would be used by engineers in industry.

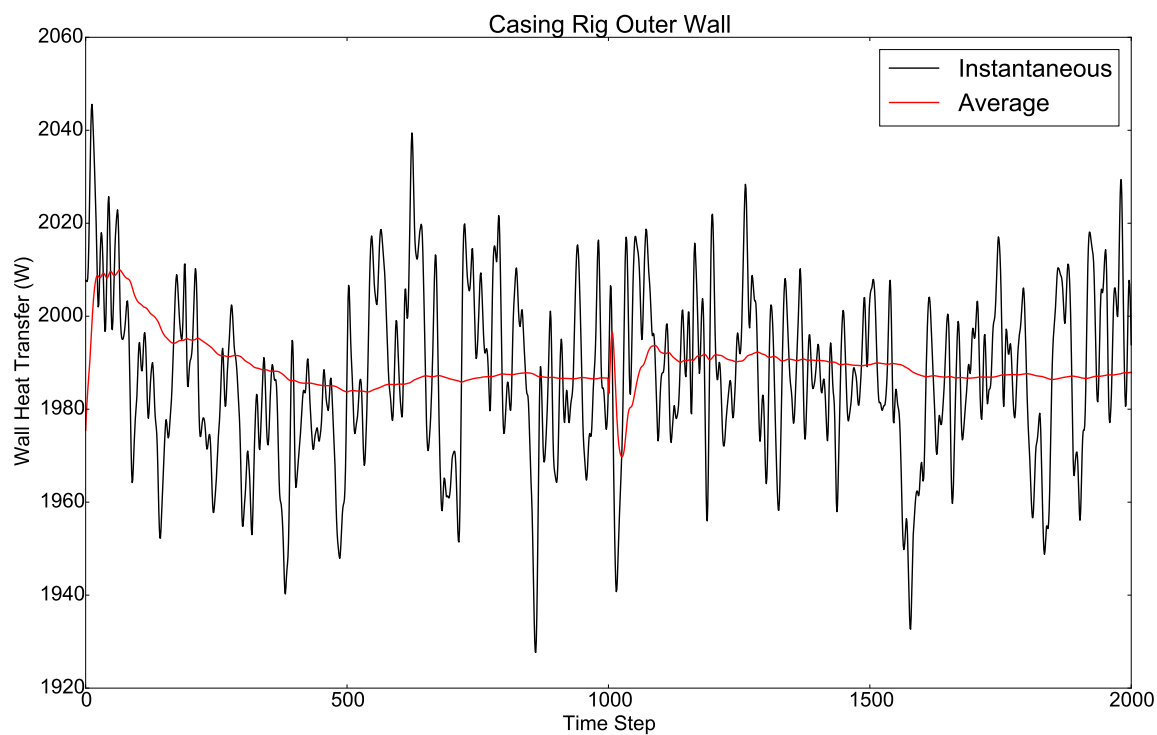
The computational mesh used for the RANS simulations is shown in Fig.5.14. The mesh was a structured hexahedral type consisting of  $7.0 \times 10^6$  cells, the mesh was clustered towards the outer wall in order to keep the wall  $y^+ < 1$ .

The solver used was ANSYS Fluent v15, run as a pressure based coupled solver using the  $k - \omega$  SST turbulence model. The temperature boundary conditions were taken from the casing rig test case.

It was found that the simulations needed to be run in an unsteady manner in order to obtain convergence. The simulations were run with a 0.1 s time-step for 1000 time-steps, 20 though flow periods, to allow for initial transients to decay. Averages were then taken over a further 1000 time-steps. A typical surface monitor of the outer wall heat transfer can be seen in Fig.5.15. Whilst simulations were successfully run using the SGDh and GGDh turbulent heat flux models, it was found not to be possible to get the casing rig case to converge when using the AFM model. The simulation using the AFM model would diverge even when run with low relaxation factors and small time-steps.



**Figure 5.14:** Casing rig cross section of computational mesh



**Figure 5.15:** Surface monitor of outer wall heat transfer

## 5.4.2 Results & Discussion

### Temperature Profiles

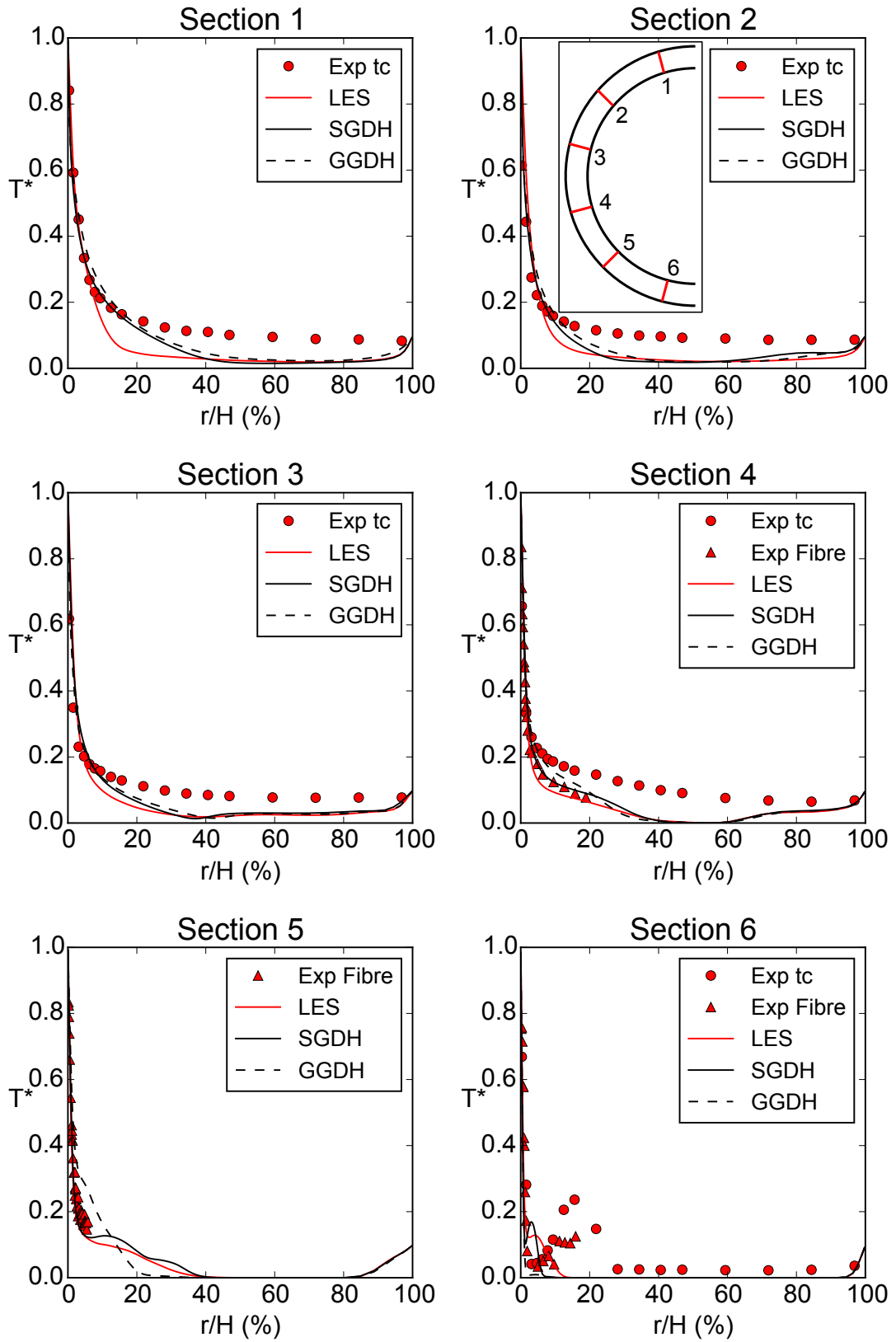
Time averaged temperature profiles from the RANS simulations are compared to the experimentally measured temperatures and the time averaged LES temperature profiles. The results are shown at the six measurement locations around the casing rig in Fig.5.16. The temperature has been non-dimensionalised as  $T^* = (T - T_{inlet}) / (T_{outer\ wall} - T_{inlet})$ , with the distance from the outer wall  $r$  being non-dimensionalised by the duct height  $H = 0.32\text{ m}$ .

Firstly comparing the results of the RANS simulations to the LES, it can be seen that in general the RANS simulations are agreeing well with the LES temperature profile. Comparing the RANS simulations to the experimental data it can be seen that at section 6, the furthest upstream section, the peak in the temperature profile caused by the streamwise vortices is not as pronounced as in the experimental data. The streamwise vortices have a significant effect on the mixing between the thermal boundary layer and the cold free stream, this effect was not fully captured by the LES and RANS simulations, this is giving lower free stream temperatures compared to the experiment.

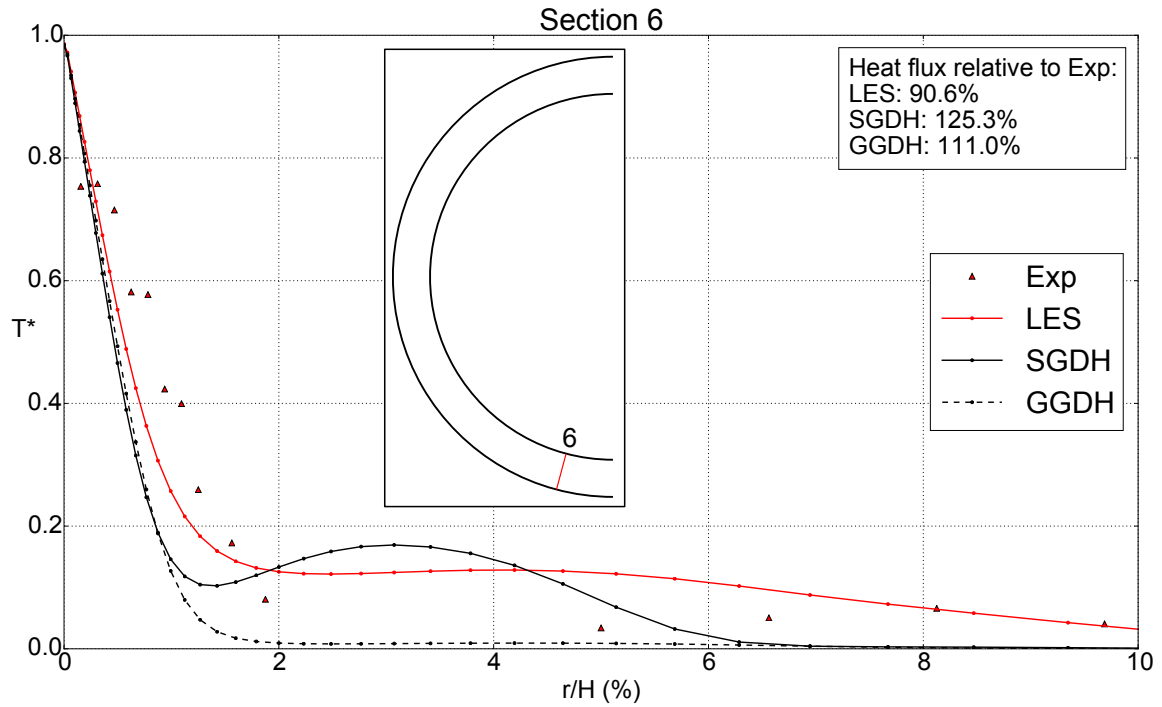
### Heat Transfer

A key output of the RANS simulations is the accurate prediction of the wall temperature gradient and hence the wall heat transfer. Temperature data measured using the fibre measurement system has sufficient spatial resolution to calculate the wall temperature gradient, and this data is available at sections 6, 5 & 4. It is difficult to obtain an accurate measurement of the temperature gradient in the experiment, as small changes in the measured temperature can have a large effect on the gradient. The gradient was calculated by fitting a straight line through the data, this was to minimise the effect on the gradient of noise from the measurement data. The first point closest to the wall was not used and the linear fit was through the next 5 points. The integral method for calculating the gradient from Ebadi et. al. [18] was also tested and found to give very similar results to the linear best fit method.

The near wall temperature profile at section 6 is shown in Fig.5.17. Due to the presence



**Figure 5.16:** Temperature profiles at six measurement locations in casing rig



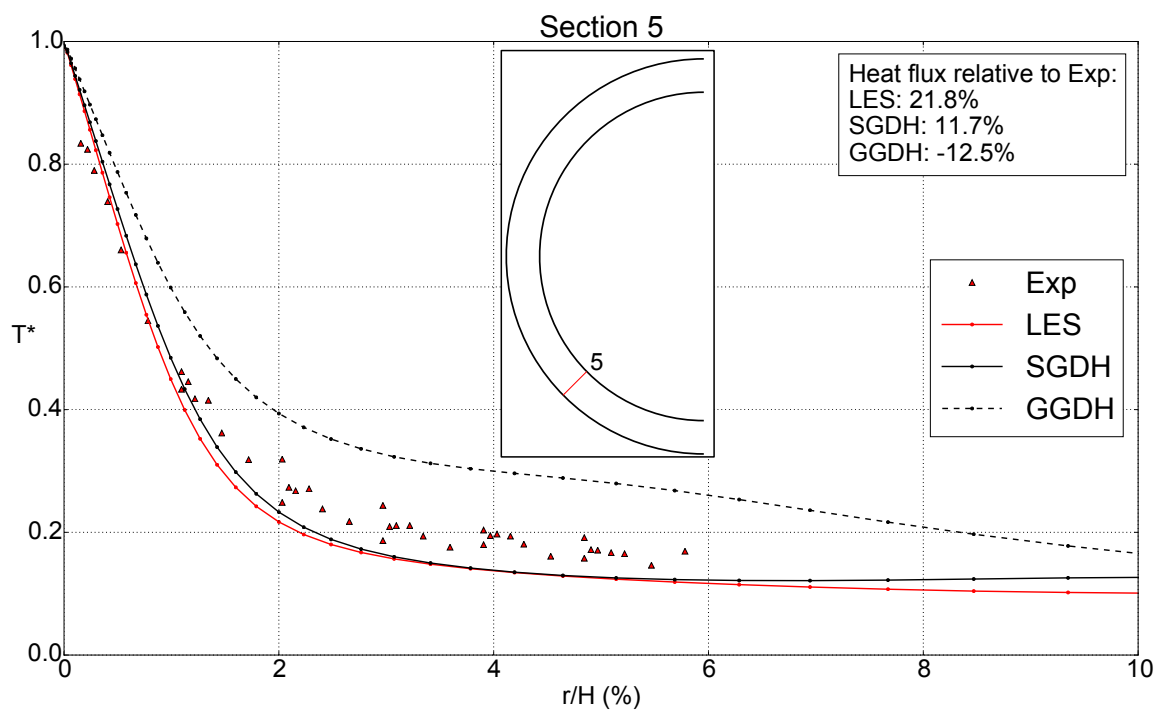
**Figure 5.17:** Wall temperature gradient at section 6

of the streamwise vortices at section 6, it is difficult to accurately predict the heat transfer at this location, as can be seen by the large differences from the experimental heat transfer.

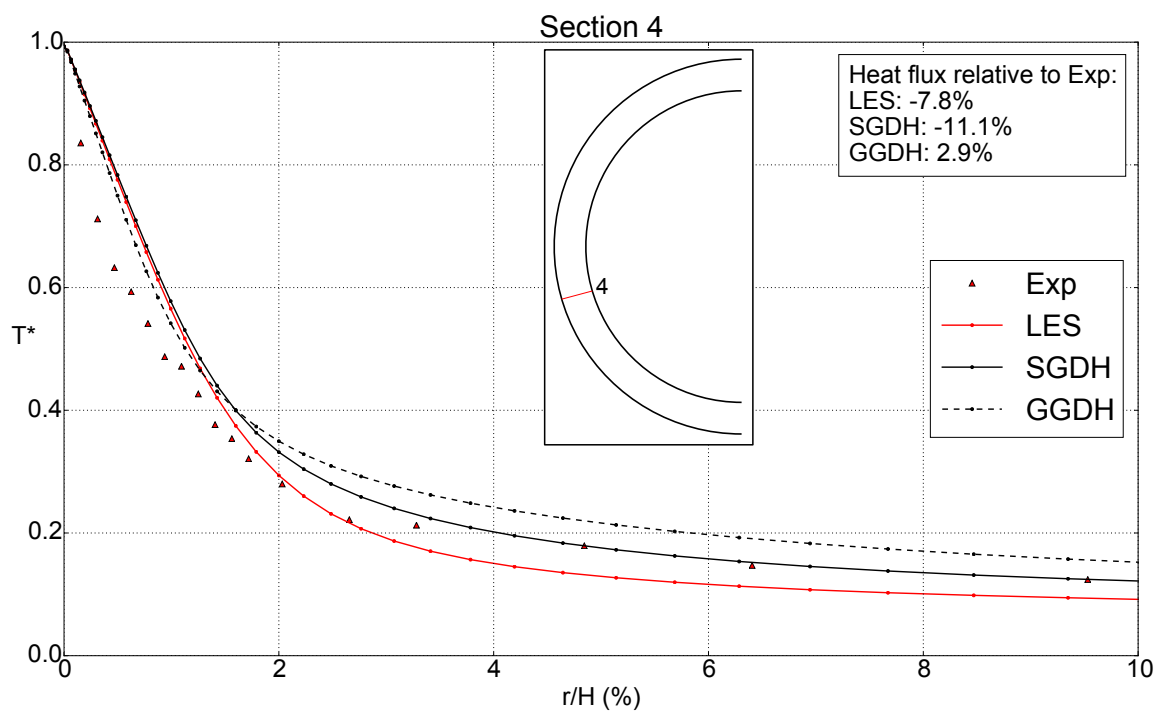
The near-wall temperature profiles at section 5, further downstream in the casing rig, are shown in Fig.5.18. The SGDH model has predicted a temperature profile that is close to that of the experimental data, giving a good prediction of the wall heat flux. The GGDH model is predicting a thicker thermal boundary layer but is still predicting a heat transfer that is close to the experiment. The thicker profile predicted by the GGDH model could be due to the streamwise vortices still having an effect on the temperature profile at section 5.

Section 4 is the furthest downstream location where the fibre temperature data is available. It is the furthest from the effect of the streamwise vortices and closest to that of a turbulent natural convection boundary layer. The temperature profiles at section 4 are shown in Fig.5.19. The SGDH model is now starting to under-predict, but is still within the required 20%. This under-prediction of the heat flux by the SGDH model is the trend that is found when the SGDH is used for turbulent natural convection flows.

Contours of the time averaged outer wall heat transfer from the LES and RANS sim-



**Figure 5.18:** Wall temperature gradient at section 5



**Figure 5.19:** Wall temperature gradient at section 4

ulations are given in Fig.5.20. The Nusselt Number has been calculated using the outer wall radius  $Nu_R = hR/\lambda$  and the conductivity is based on the inlet temperature. The general form of the heat transfer distribution is a small initial region of very high heat transfer where the cold fluid first comes into contact with the hot wall and the thermal boundary layer is very thin. As the laminar boundary layer grows and thickens the heat transfer then reduces. The boundary layer then transitions through the formation of streamwise vortices, these vortices cause a significant increase in the wall heat transfer when the vortices impinge onto the outer wall. After the streamwise vortices decay the turbulent boundary layer grows and thickens, causing the heat transfer to be reduced.

Comparing the results of the RANS simulations to the LES simulation it can be seen that the vortex formation and transition location is occurring later in the RANS simulations. The effect of the vortices is also propagating further downstream in the RANS simulations, this was seen in the temperature profile at section 5. In the top half of the cavity the reduction in heat transfer as the turbulent boundary layer thickens and forms a stably stratified region is not present in the RANS simulations, giving a higher heat transfer in the top of the cavity. The GGDH model is also predicting a higher heat transfer than the SGDH model in this region and is not predicting a reduction in heat transfer due to the stable stratification.

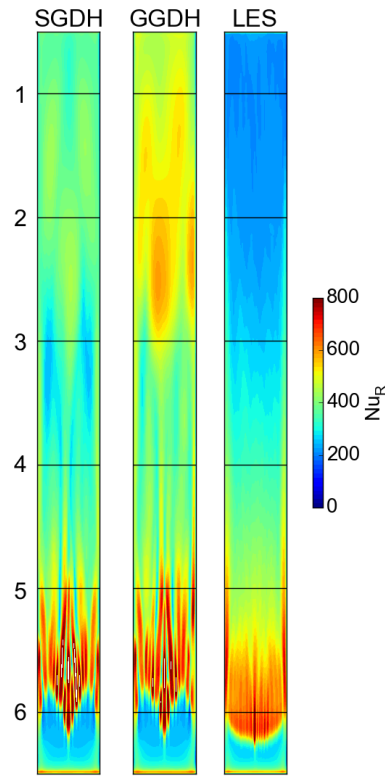
### Turbulent Heat Fluxes

As no experimental data is available for the turbulent heat fluxes in the casing rig, the terms predicted by the RANS simulations using the different turbulent heat flux models are compared to the terms resolved by the LES simulation. The terms are compared in polar coordinates as the circumferential  $\overline{U'_\theta T'}$  and outer wall normal  $\overline{U'_r T'}$  terms.

Profiles of the circumferential turbulent heat flux term  $(\overline{U'_\theta T'})$  are shown at the six measurement locations in Fig.5.21. It can be seen that in general the RANS models are under-predicting the streamwise<sup>1</sup> turbulent heat flux. The SGDH model is predicting almost zero values at all six locations, this is due to the streamwise temperature gradient being small. The GGDH model is also under predicting the streamwise turbulent heat flux,

---

<sup>1</sup>The circumferential velocity is much larger than the radial velocity, hence the circumferential turbulent heat flux can be considered the streamwise turbulent heat flux.



**Figure 5.20:** Time averaged heat transfer contours on casing rig outer wall

but at the locations further downstream in the rig the model is performing better and is predicting the same trend as the LES.

Profiles of the wall normal turbulent heat flux  $(\overline{U'_\theta T'})$  are shown in Fig.5.22 at the six measurement locations in the casing rig. It can be seen that there is a trend that in the upstream sections of the rig the RANS models are under-predicting the wall normal turbulent heat flux whereas at the downstream sections the RANS models are over-predicting the turbulent heat flux. The reason for this trend is due to the effect of buoyancy on the turbulent fluctuations. In the upstream sections of the rig the flow is on top of the heated wall and the buoyancy force acts to increase the turbulent fluctuations. In the downstream sections of the rig the flow is underneath the heated wall giving a stable stratified flow, the buoyancy force then acts to damp out the turbulent fluctuations. This effect is resolved by the LES simulation but is not modelled by the RANS simulations. The effect on the flow can be seen as transition occurs later in the RANS simulations compared to the LES. At the top of the rig the wall heat transfer has been over-predicted by the RANS simulations due



to the over-prediction of the wall normal turbulent heat flux.

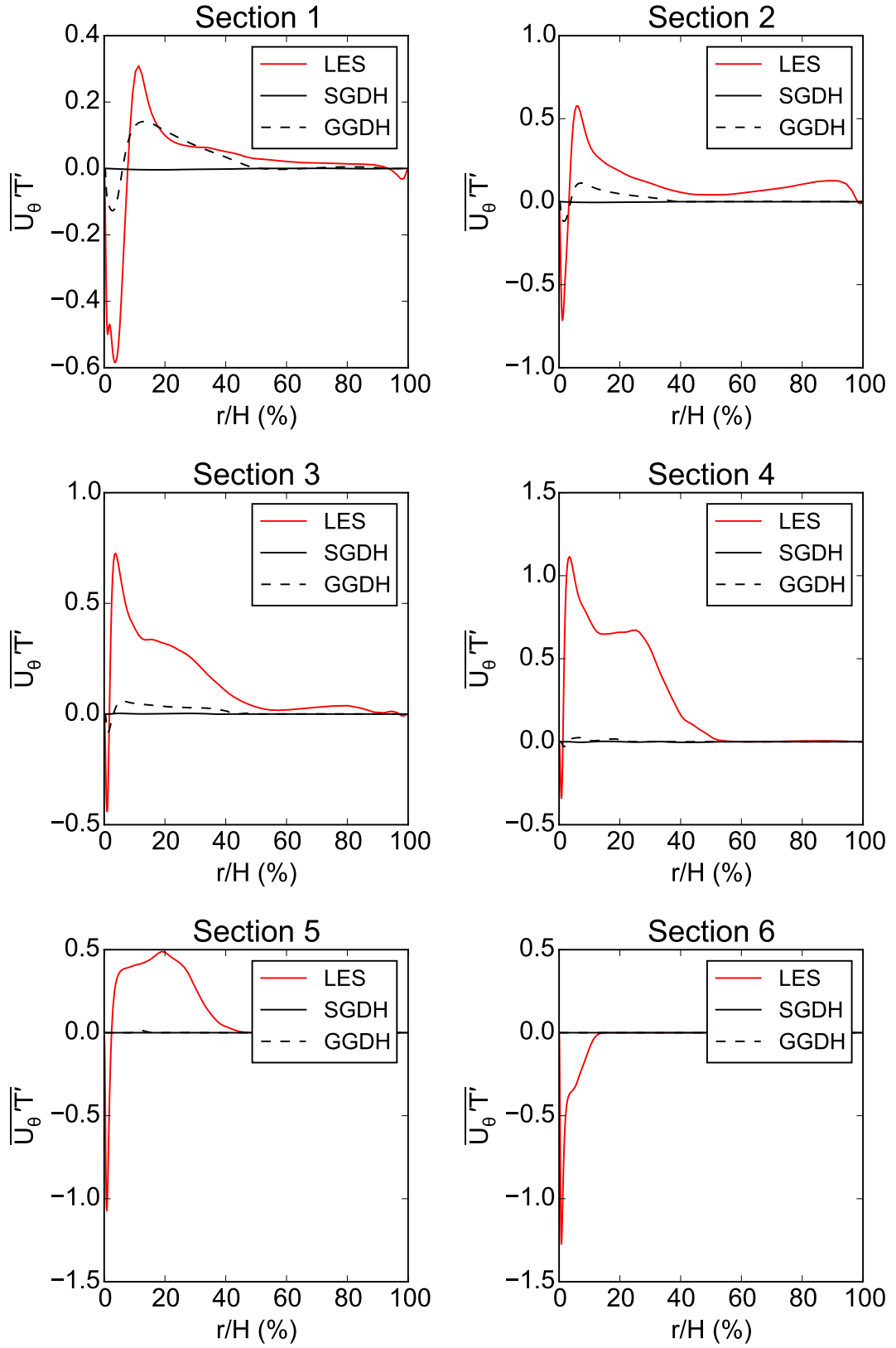
The casing rig is a challenging case for a models to predict the turbulent heat fluxes. This is due to the complexity of transition through the formation of streamwise vortices and the effect of buoyancy aiding and damping turbulent fluctuations. It has been found that the RANS simulations tend to under predict the streamwise turbulent heat flux with the SGDH model giving an almost zero value. The wall normal turbulent heat flux showed under-prediction in the lower half of the rig and over-prediction in the top half.

### 5.4.3 Buoyancy Modification

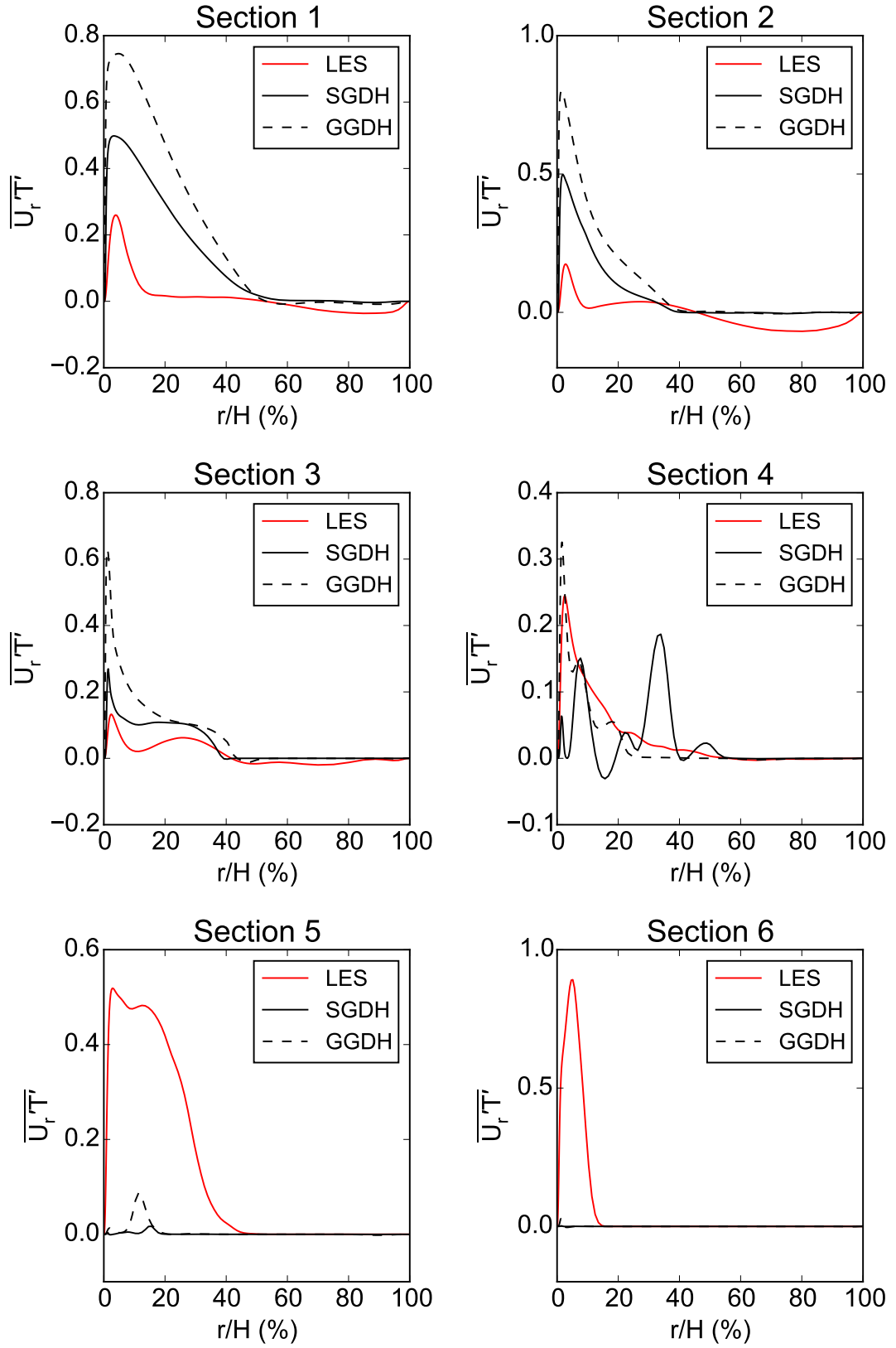
It was found that the RANS simulations did not predict the reduction in heat transfer at the top of the casing rig. This was found to be due to the over-prediction of the wall normal turbulent heat fluxes at the top of the casing rig. The SGDH and GGDH turbulent heat flux models do not contain any buoyancy terms, so cannot model the reduction in turbulent fluctuations in stably stratified regions. The AFM model has one buoyancy term but this model requires an extra equation for the temperature variance to be solved. This extra computational cost combined with the instability of the AFM model does not make this an attractive option. What is desired is a modification to the turbulent heat flux model to allow for the effects of buoyancy to be modelled.

A modification to the GGDH model has been developed in order to include the effects of buoyancy in the model. The GGDH model was chosen as the starting point as it performs better than the SGDH model but without the instability of the AFM model. The baseline GGDH model has a constant  $C = 0.15$  that scales the predicted turbulent heat flux. This constant has been replaced by the function given by Equation.5.6.

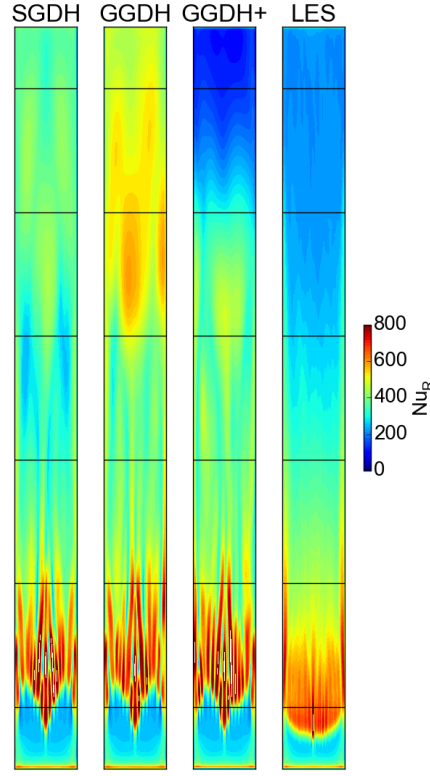
In Equation.5.6 the term  $\tanh \left| \frac{u}{v} \right|$  determines if the flow is over a horizontal or vertical surface. Where  $u$  is the velocity perpendicular to the gravity vector and  $v$  is the velocity parallel to the gravity vector. If the flow is parallel to a vertical surface then the term will become 0 and the function will revert to  $C = 0.15$ . The term  $\operatorname{sgn} \left( g_j \frac{\partial T}{\partial x_j} \right)$  determines if the flow is above or below a heated surface. If the flow is below a heated surface the term will be -1, and +1 for flows above a heated surface. The function will therefore increase the turbulent heat flux when the flow is above a heated surface and decrease the turbulent heat



**Figure 5.21:** Circumferential turbulent heat flux term in casing rig



**Figure 5.22:** Wall normal turbulent heat flux term in casing rig



**Figure 5.23:** Time averaged heat transfer contours on casing rig outer wall

flux when the the flow is below a heated surface. This modification to the GGDH model will be referred to as GGDH+.

$$C = 0.15 \left( 1 + \tanh \left| \frac{u}{v} \right| \operatorname{sgn} \left( g_j \frac{\partial T}{\partial x_j} \right) \right) \quad (5.6)$$

A RANS simulation was run using the GGDH+ model, with the same numerical method as described previously. The outer wall heat transfer is compared to the other RANS simulations and the LES results in Fig.5.23. The modification to the GGDH model can be seen to have reduced the heat transfer in the top half of the cavity compared to the baseline GGDH model. The heat transfer distribution is now closer to that predicted by the LES simulation.

The effect of the GGDH+ model on the predicted circumferential and wall normal turbulent heat fluxes is shown in Fig.5.24 and Fig.5.25 respectively. For the circumferential turbulent heat flux the predictions are still low compared to the LES and have been further reduced towards the top of the domain. For the wall normal turbulent heat flux the results

|            | Increase over baseline |
|------------|------------------------|
| SGDH (UDF) | 6.4                    |
| GGDH       | 5.9                    |
| GGDH+      | 5.7                    |

**Table 5.2:** Fluent UDF increase in computational cost per time-step

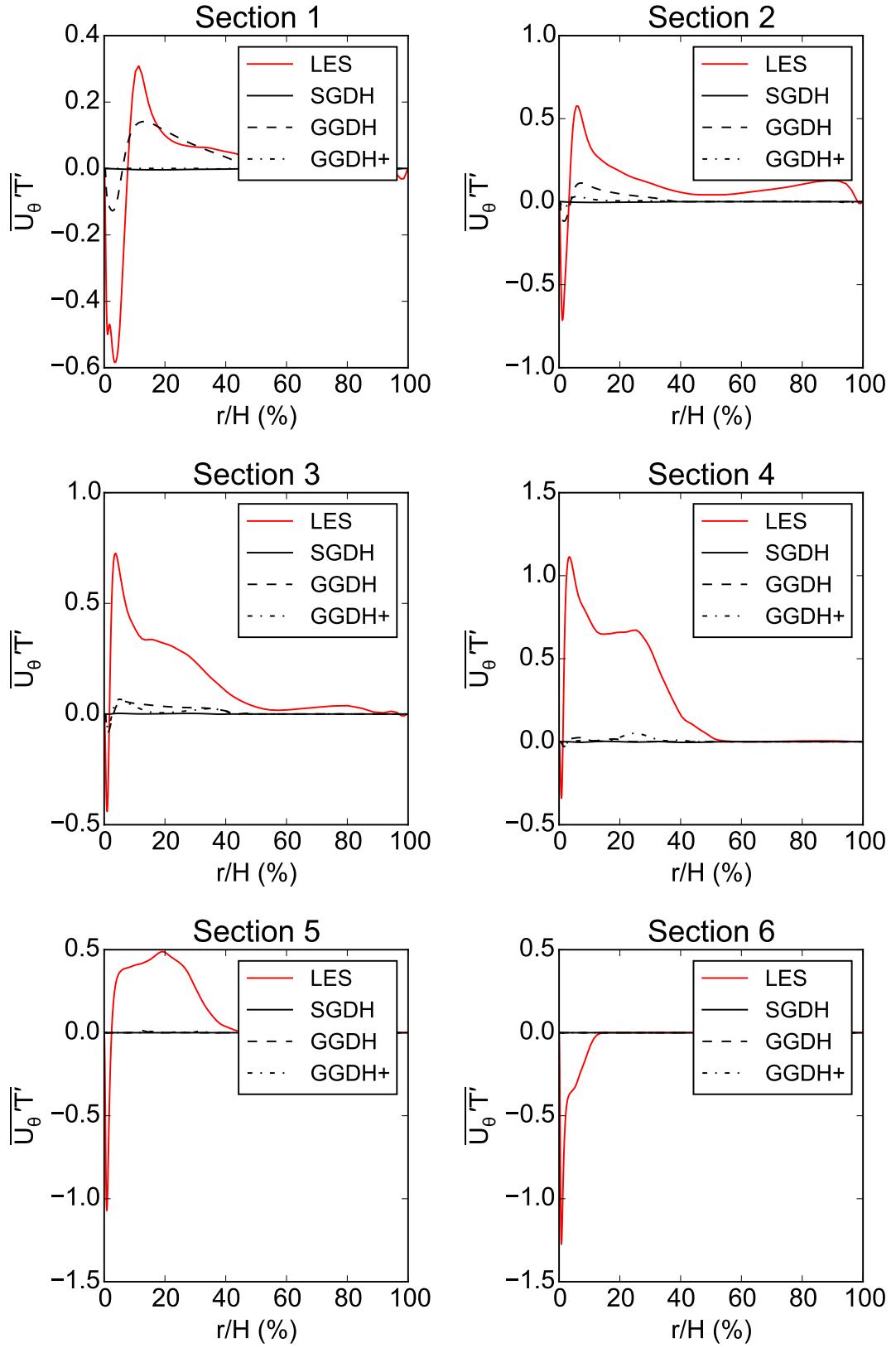
are similar to the baseline GGDH model in the bottom half of the rig. Towards the top of the rig the modification is working as desired as is reducing the turbulent heat flux term to be closer to the LES value. At section 1 the modification has reduced the turbulent heat flux too much and the value is close to zero.

This study has demonstrated that the modification to the GGDH term is able to improve the wall heat transfer in the region where the flow is underneath a heated surface. Outside of this region the modification had little effect over the baseline GGDH model. This modification can therefore be used in combination with the GGDH model to give improved predictions for cases with stably stratified flows.

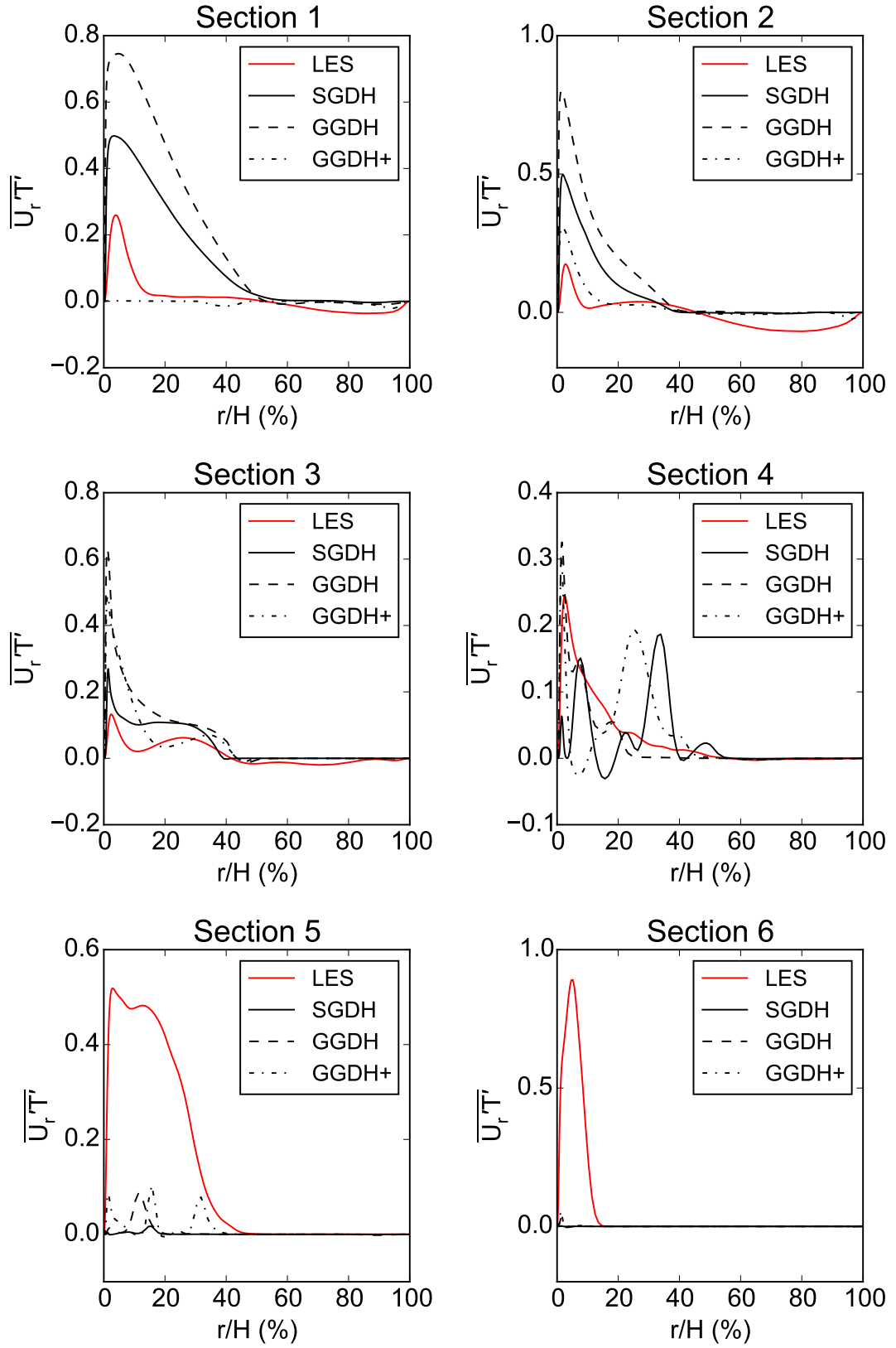
#### 5.4.4 Computational Cost

The aim of this investigation was to produce numerical predictions at a computational cost available to engineers. The increase in computational time when using the UDF are compared to a baseline case without any UDF, the results are given in Table.5.2. It can be seen that the increase in computational cost is due to the use of UDF rather than more complex turbulent heat flux models. This can be seen as the SGDh model using the UDF takes over six times longer than the baseline case, also using the SGDh model. If the more advanced GGDH and GGDH+ models were implemented directly into Fluent then it may be possible to use these models with little computational cost increase. It was found that in all of these cases a time-step of 0.1 s was stable and the simulations took a similar number of time-steps to converge.

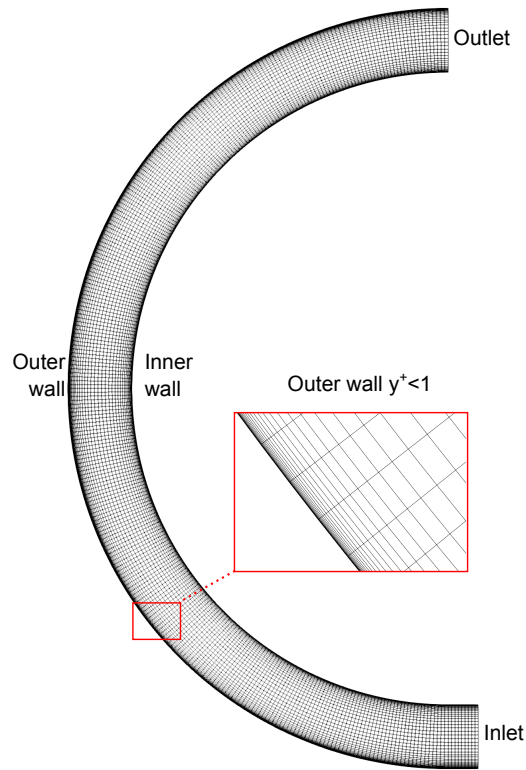
The baseline casing rig case with a 7 million cell mesh took  $\approx 45$  s per time-step on a 32 core engineering desktop PC, or 25 hours for 2000 time-steps. This is a reasonable computational time for an engineer, but the 6 days for a case using the UDF would likely be too long, and would instead require time on a more powerful cluster.



**Figure 5.24:** Circumferential turbulent heat flux term in casing rig



**Figure 5.25:** Wall normal turbulent heat flux term in casing rig



**Figure 5.26:** Cross section of reduced casing rig computational mesh

#### 5.4.5 Reduced Mesh study

The mesh used for the initial RANS simulations was the same mesh as used in the LES study of the casing rig. This allowed for a direct comparison of the RANS with the LES but this mesh is not representative of the type of mesh that would be used by engineers in industry when performing RANS simulations. A study has therefore been conducted using a reduced mesh of the casing rig geometry.

The computational domain will include only the measurement section of the casing rig and will no longer include the inlet section. The mesh refinement will also be reduced, giving a coarser mesh. A cross section of the reduced mesh is shown in Fig.5.26, the mesh contains  $7.7 \times 10^5$  cells compared to  $7.0 \times 10^6$  in the larger mesh. This order of magnitude reduction in mesh size causes a significant reduction in computational cost, both in CPU time and memory requirement. The reduced mesh still has the cells clustered towards the walls keeping the outer wall  $y^+ < 1$ .

The boundary conditions were the same as used on the previous larger mesh, except

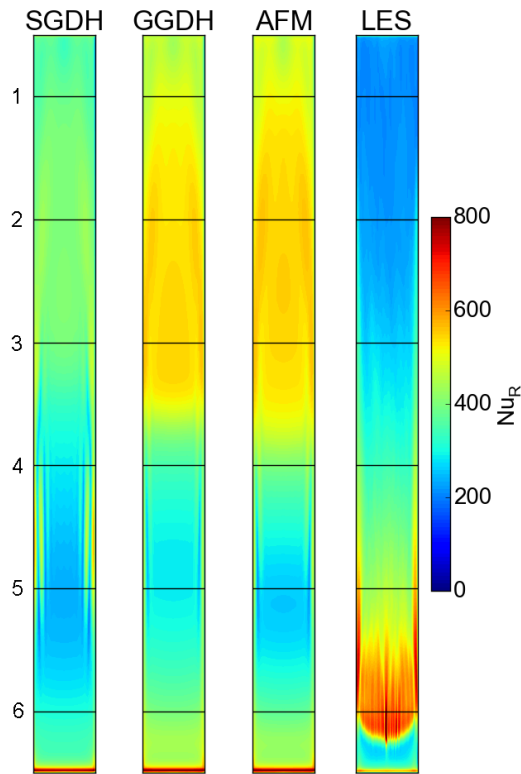


that the inlet now used a total pressure profile to represent the effect of the inlet section. The numerical set up was also kept the same as in the previous RANS simulations, but it was found that the simulations converged sufficiently well when run in a steady manner. Removing the need to run an unsteady simulation gave a large reduction in computational cost, with the baseline SGDH case taking under 10 minutes to run on a 32 core PC. It was also found that the AFM model was able to be run using reduced mesh and give a converged solution.

The outer wall heat flux profiles from the reduced mesh RANS simulations are compared to the time averaged LES results in Fig.5.27. It can be seen that the RANS simulations with the reduced mesh have not predicted the streamwise vortex formation near section 6. The predicted transition location has also moved further downstream to between sections 3 and 4, visible due to the increase in heat transfer in this location. Comparing the three RANS simulations to one another it can be seen that the GGDH and AFM models are in general predicting higher levels of heat transfer than the SGDH model. This is likely due to the prediction of larger turbulent heat fluxes. The GGDH and AFM models are giving similar predictions to one another but the GGDH model has been found to be more stable.

Looking at the temperature profiles from the reduced mesh simulations in Fig.5.28, it can be seen the RANS profiles are agreeing in general with the LES. This shows that the reduced mesh can be used to predict the general flowfield. It can also be seen that the GGDH and AFM models are giving similar predictions to one another.

It has therefore been shown that for the casing rig case it is necessary to include the inlet into the computational domain and have a fine mesh in order to capture the streamwise vortices. The reduced mesh was not able to predict the correct distribution of outer wall heat flux but could be used by engineers to predict the general flowfield. It was also found that when the AFM model case converged the results were very similar to the GGDH model case. As the AFM model has been found to be less numerically stable than the GGDH model, the GGDH model is the best of the two for engineering simulations.



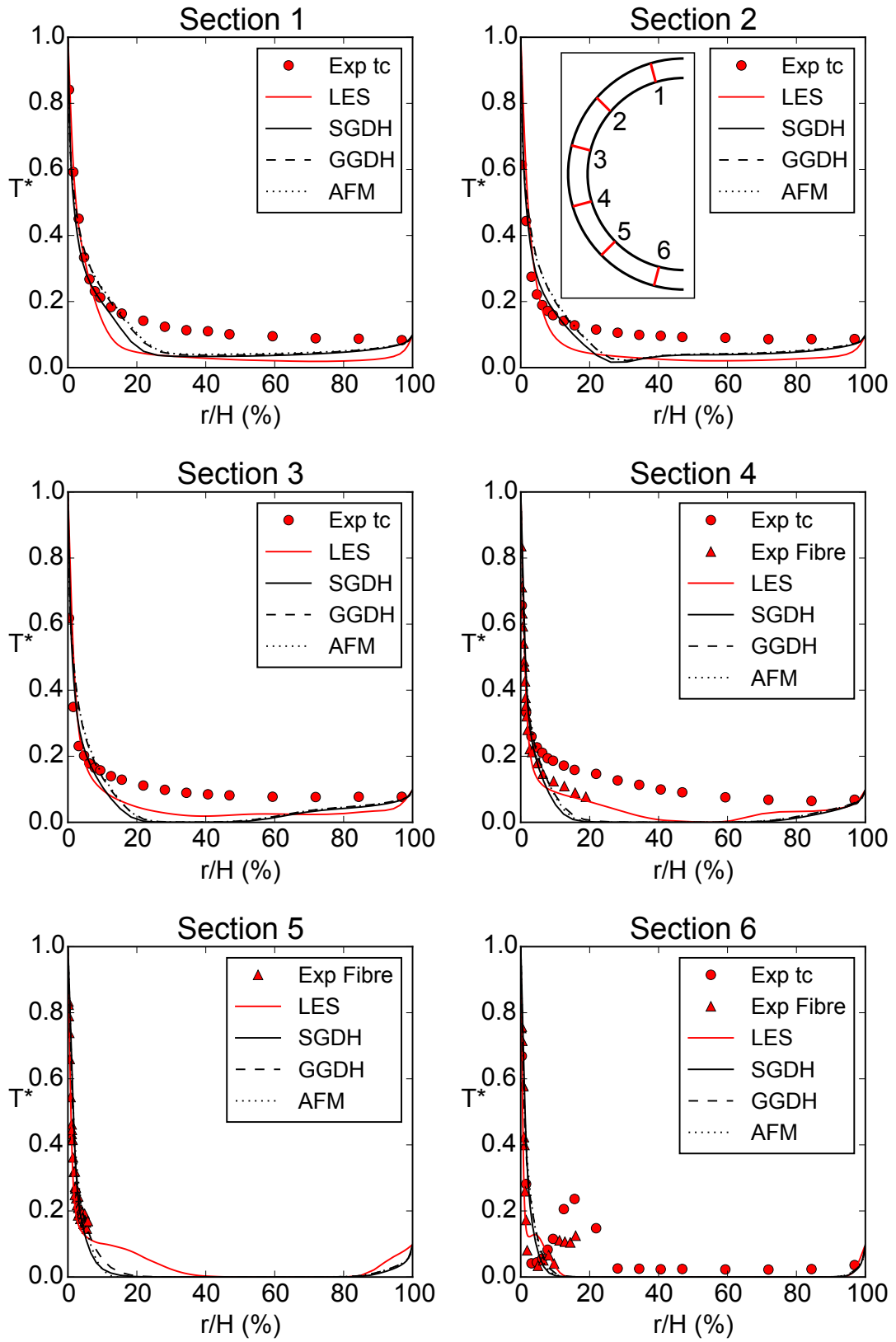
**Figure 5.27:** Outer wall heat transfer contours, RANS simulations with reduced mesh

#### 5.4.6 Casing Rig conclusions

RANS simulations were conducted on the gas turbine casing rig geometry to test the performance of different turbulent heat flux models. The casing rig is a challenging test case for the simulations as it features transition and the formation of streamwise vortices. There is also flow both above and below a heated wall, giving situations where the buoyancy forces are aiding or inhibiting boundary layer growth at different locations inside of the rig.

A baseline RANS simulation using the SGDH model was compared to simulations run with the GGDH turbulent heat flux model. It was found that it was not possible to get a simulation using the AFM model to converge. The models were compared with data from experimental measurements and from an LES simulation of the rig.

The temperature profiles predicted by the RANS simulations were similar to each other and followed the same trend as the experiment. This shows that the simplest SGDH model could be used if a qualitative prediction of the temperature field is required. The simulations predicted the streamwise vortices forming later on in the rig and hence did not predict



**Figure 5.28:** Non-dimensional temperature profiles in casing rig, RANS simulations with reduced mesh

the temperature peak found in section 6 of the experiment. This also gave lower mixing between the thermal boundary layer and the free-stream.

A key output of the simulations is the prediction of the wall heat flux, which is set by the wall temperature gradient. The temperature gradient predicted by the RANS simulations was compared to experimental data at sections 6, 5 & 4 in the rig. At section 6 the streamwise vortices were present, this made heat transfer predictions difficult and makes it hard to draw conclusions on the performance of the models. Further downstream at section 5 the effects of the vortices were still present in the simulations but both the SGDh and GGDH models were giving good wall heat flux predictions. Finally at section 4 the flow was closest to that of a turbulent boundary layer flow, both models were giving good predictions of the heat flux but the trend of the SGDh model under-predicting was starting to appear.

The turbulent heat flux terms that were predicted by the models were compared with resolved terms calculated by the LES simulation. For the streamwise term it was found that the RANS models under-predicted compared to the LES. The SGDh model performed the worst and predicted almost zero values for the streamwise term. The GGDH model performed better when predicting the streamwise heat fluxes, and at the downstream turbulent sections could predict the locations for the peaks in the profile. For the wall normal turbulent heat flux the RANS models tended to under-predict at the bottom of the rig and over-predict at the top of the rig, this was due to the effect of buoyancy on the turbulent fluctuations.

The RANS models did not predict the reduction in wall heat transfer towards the top of the casing rig caused by the development of a stratified flow. A modification to the GGDH model was developed to account for the effect of the buoyancy forces on the turbulent heat flux terms. The GGDH+ model was found to improve the predictions of wall heat flux and turbulent heat fluxes at the top of the casing.

The SGDh and GGDH temperature profiles were similar meaning the simpler model could be used for qualitative flow predictions. It was also found that the AFM was too unstable to be used in this case, which would mean that this model may not be suitable for test cases run by engineers. The best model for engineering use would therefore be the

GGDH model with the GGDH+ modification, giving a compromise between computational cost and accuracy.

## 5.5 Conclusions

RANS simulations run using three different turbulent heat flux models were validated on three separate test cases, the heated vertical plate, the differentially heated tall cavity and the gas turbine casing rig.

The heated vertical plate was used as a validation case as it operates at Rayleigh Numbers that are the same order as an industrial gas turbine during shutdown. There is also validation data from different literature sources, including measurements of turbulent heat fluxes. The differentially heated cavity was used as there is a large experimental dataset available, including measurements of mean and rms temperatures and velocities. The experimental data from the tall cavity was supplemented by a DNS simulation of the cavity, providing resolved turbulent heat flux terms for validation of the RANS modelled terms. The final validation case was the gas turbine casing rig, this test case is the closest to that of a real industrial gas turbine, operating at the same Rayleigh Number and using engine representative geometry. The temperature measurements from the experiment was supplemented by data from an LES simulation of the rig.

A method for implementing new turbulent heat flux models into ANSYS Fluent was developed. This method involved suppressing the internal turbulent heat flux model and then adding in the turbulent heat flux as a source term into the energy equation. Simulations were then run to validate this method of implementation, which was found to be successful.

For each of the three validation cases RANS simulations were run, in order to be representative of the type of low computational expense simulations that are commonly used in industry. A baseline simulation was performed which used the SGDh turbulent heat flux model, this was compared to results from simulations using the GGDH and AFM turbulent heat flux models that had been implemented into Fluent using UDFs. The results from the RANS simulations were compared to the validation data for the three test cases, looking at temperature and velocity fields, heat transfer and turbulent heat flux predictions. Accurate

prediction of wall heat transfer is a key requirement for the models as this parameter is important in engineering design calculations.

The baseline case using the SGD model was able to predict the general flow and temperature field, meaning that the SGD model is useful if the general temperature or flow field is all that is required. The GGDH model could also predict the general temperature and flow fields, showing the implemented model did not have an adverse effect on the calculation. The AFM model performed similarly to the GGDH model in the vertical plate and tall cavity cases but was too unstable to be used in the casing rig case.

For the wall heat transfer prediction it was found that the SGD model consistently under-predicted the heat transfer, giving errors of over 20%, this would make the SGD model unsuitable for engineering design calculations. The implemented GGDH and AFM models were found to perform much better in heat transfer predictions than the baseline, with an under-prediction usually around 10%. This puts the models within the 20% error margin and allows these simulations to be used for engineering design. The heat transfer predictions of the GGDH and AFM models were very similar, showing that there is little benefit for the increased cost and complexity of the AFM model, with the downside of being less stable.

When comparing the wall heat transfer in the casing rig predicted by the LES to the RANS simulation using the GGDH model, it was found that the RANS simulation over-predicted the heat transfer at the top of the casing. In this region the flow is stably stratified which reduces the heat transfer, however this effect is not included in the GGDH model. A modification to the GGDH model was developed, known as GGDH+. This modification added a factor that accounted for if the flow was underneath or on top of a heated surface, and modified the turbulent heat flux accordingly. This modification was found to improve the heat transfer and turbulent heat flux predictions over the baseline GGDH model.

The best choice of turbulent heat flux model for prediction of heat transfer in a gas turbine would therefore be the GGDH model. This is due to the increased accuracy over the SGD model and the lower cost and better stability over the AFM model. If the case is likely to involve regions of stably stratified flow then the GGDH+ model would likely be more appropriate. The SGD model is still useful for simulations where computational cost has

to be very low, such as a large design of experiments, where trends are more important than exact values or where only the general flowfield is required.

## Chapter 6

# Streamwise Vortex Study

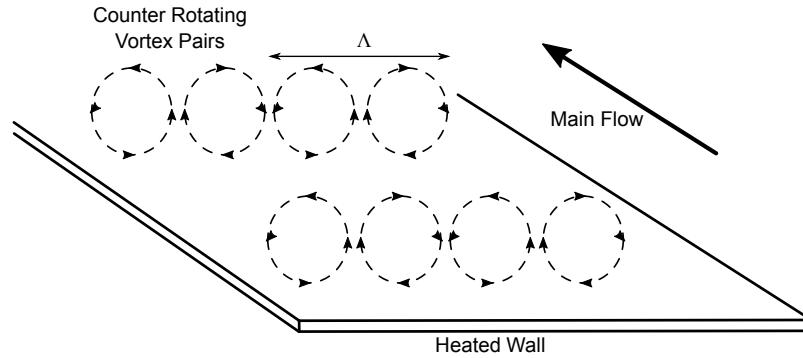
*An investigation was conducted into the formation and development of streamwise thermal vortices. The vortices are of interest as they can locally increase the heat transfer and could cause problems if they occurred in a gas turbine during shutdown. The vortices have been observed in both the casing rig experiment and numerical simulations. The formation mechanism of these vortices and what sets the vortex wavelength is not well understood, so a numerical investigation using the casing rig geometry was conducted.*

*Numerical studies using LES allowed the flow to be visualised and parameters such as helicity and vorticity to be shown. A method for the formation of the streamwise vortices was proposed, which involved the interaction of an incoming boundary layer vortex with buoyancy forces from the heated wall.*

*Simulations were also run varying several parameters such as wall temperature and fluid properties. The variation in wavelength with temperature was found to agree well with existing experimental data. It was then shown that the variation in wavelength with temperature was not due to changes in fluid properties.*

*A final simulation used a heated inner wall, so the flow was underneath a heated surface. In this case the streamwise vortices did not occur but instead a secondary flow phenomenon occurred.*





**Figure 6.1:** Schematic of thermal vortex wavelength

## 6.1 Background

Streamwise vortices were found to develop in the gas turbine casing rig, they were observed in both the experiment and the numerical investigation. The vortices formed as the laminar boundary layer from the inlet flowed over the heated outer wall of the rig. The buoyancy force normal to the flow direction caused the boundary layer to wrap up into pairs of counter rotating vortices. These vortices then acted as the transition mechanism for the boundary layer.

The vortices' impact on the heat transfer from the heated outer wall, when the vortices impinge onto the heated wall the heat transfer is enhanced. This heat transfer enhancement by the vortices gives characteristic streamwise streaks on a contour of outer wall heat transfer. The distance between these streaks is set by the vortex wavelength  $\Lambda$ , shown schematically in Fig.6.1.

The physical mechanism for the development of the streamwise vortices and the factors that set the wavelength are not well understood. There are several studies in the literature that have investigated this vortex phenomenon. An experimental investigation into the vortices was conducted by Cheng and Kim [12]. The experiment used air flowing over a heated inclined plate with the vortices being visualised using smoke. The experiment was conducted with plate angles ranging from 0 to 20° from the horizontal and temperature differences between the heated plate and the ambient air of between 15.5°C and 37.5°C. This gave Grashof numbers based on  $x$ , the distance from the plate leading edge, of  $Gr = 1.02 \times 10^6$  to  $2.13 \times 10^8$ . It was found that the inclination angle of the plate did not have

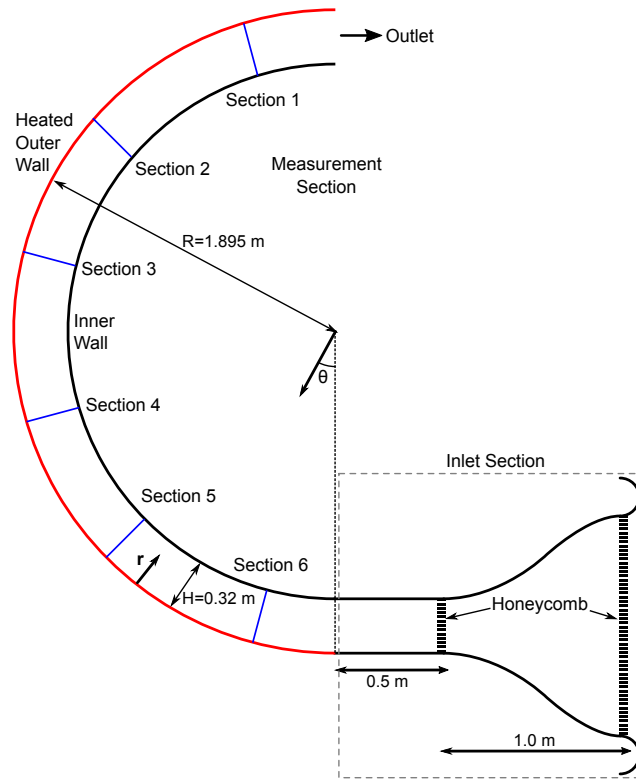
a significant effect on the wavelength of the vortices but there was an effect when the plate temperature was varied. A correlation was presented giving the ratio of critical distance to the vortex wavelength,  $x_c/\Lambda = CGr_x^{1/4}$ . Where the critical distance  $x_c$  is the distance from the start of the heated wall to when the vortices start to form and  $\Lambda$  is the vortex wavelength. The constant  $C$  had upper and lower bounds of 0.088 and 0.068 respectively.

The development of streamwise vortices over a heated plate was also investigated by Sparrow and Husar [86], this time using water as the working fluid. At a fixed  $\Delta T = 11^\circ\text{C}$  between the fluid and the heated plate the wavelength of the vortices was not found to vary significantly over the range of plate angles from  $30^\circ$  to  $60^\circ$  from the vertical. The temperature difference  $\Delta T$  was found to have an effect of the wavelength, with the wavelength decreasing as  $\Delta T$  was increased. The number of vortex streaks increased from 30 to 42 in a 20 cm width, as  $\Delta T$  was increased from  $9^\circ\text{C}$  to  $28^\circ\text{C}$ .

A more recent investigation into the streamwise vortices was conducted by Jeschke et. al. and Jeschke and Beer [34, 33]. This investigation used water flowing over an inclined plate which was heated with a constant heat flux heater. The vortices were visualised using liquid crystal thermography which allowed for quantitative measurements of the local heat transfer. The heat transfer enhancement due to the vortices impinging onto the heated plate was able to be visualised.

The studies on the streamwise vortex phenomenon often consider the problem to be that of a flow stability problem and analyse the vortices using stability theory, such as in studies by Chen and Tzuoo [11] and Tumin [100]. To the authors knowledge however there is no literature that describes the physical mechanism of the vortex formation.

The streamwise vortices show some similarity to hairpin or horseshoe vortices that form in the viscous sublayer of a boundary layer [22]. The mechanism for the formation of these hairpin vortices is due to stretching of a transverse vortex element, as described by Kline et al. [41]. It may be the case that the streamwise vortices are therefore a form of hairpin vortices.



**Figure 6.2:** Schematic of casing rig geometry

## 6.2 Numerical method

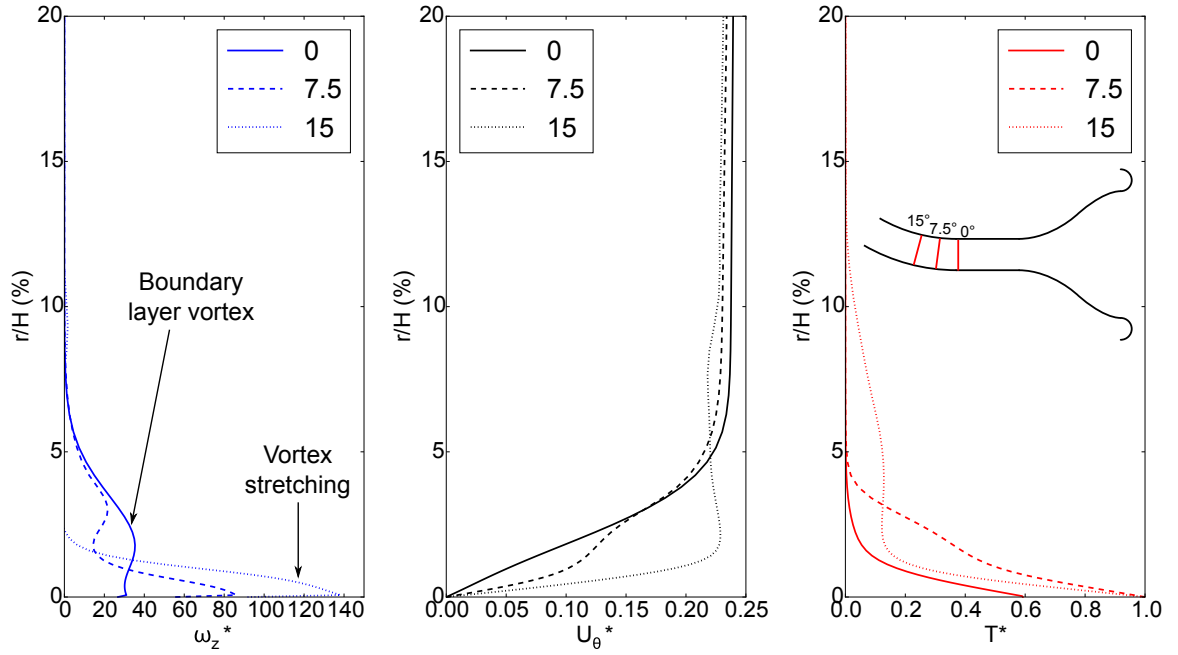
The vortices were investigated using LES simulations of the casing rig geometry, shown schematically in Fig.6.2. The computational mesh and simulation set-up were the same as that used for the LES simulation described in Chapter.3. Changes to boundary conditions and material properties were applied one at a time, with the other factors being kept constant.

For simulations where the material properties were varied, a piecewise linear function of the property with temperature was multiplied by a constant scaling factor.

## 6.3 Results & Discussion

### 6.3.1 Vortex Formation

The mechanism of the formation of the streamwise vortices was investigated using the baseline case with a 150°C outer wall temperature. The vorticity at the start of the heated

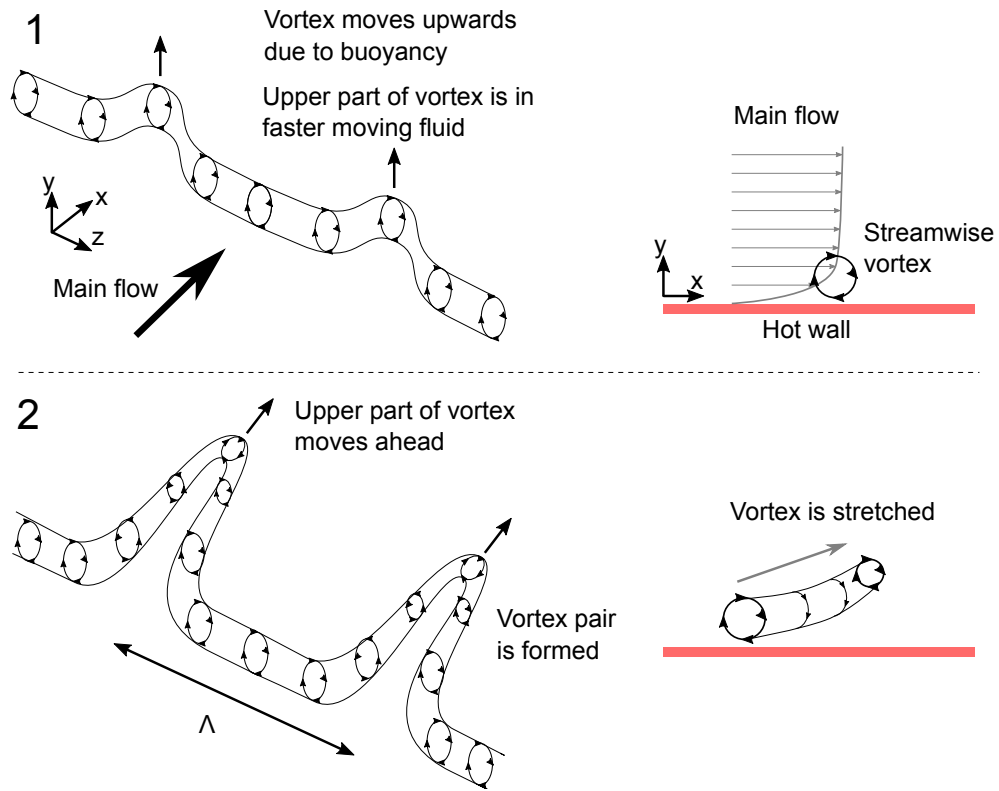


**Figure 6.3:** Non-dimensional z-vorticity, streamwise velocity & temperature at three centreline locations

outer wall is concentrated in the boundary layer which formed as the flow travels through the inlet section. The vortex is formed close to the wall and is in the spanwise  $z$  direction, this can be seen in Fig.6.3.

As the boundary layer flow is heated by the outer wall some sections of the boundary layer, containing the spanwise vortex, rise up due to the action of buoyancy forces. These rising sections of the boundary layer then enter fluid that has a higher streamwise velocity than the boundary layer. This higher velocity can be seen by looking at the streamwise velocity profiles in Fig.6.3. This higher velocity causes the section of fluid and spanwise vortex to move ahead of the fluid and spanwise vortex still in the slower moving fluid. The spanwise vortex is then stretched as the upper sections move ahead of the lower sections, this vortex stretching can be seen in Fig.6.3. This stretching causes parts of the vortex to become streamwise and form a counter rotating vortex pair. This process is also shown schematically in Fig.6.4.

In Fig.6.5 contours of non-dimensional  $z$  vorticity, circumferential velocity and temperature are given on a plane at an angle of  $7.5^\circ$  from the start of the outer wall. The vorticity has been non-dimensionalised by the buoyant velocity ( $\sqrt{g\beta\Delta TR}$ ) and the outer wall radius

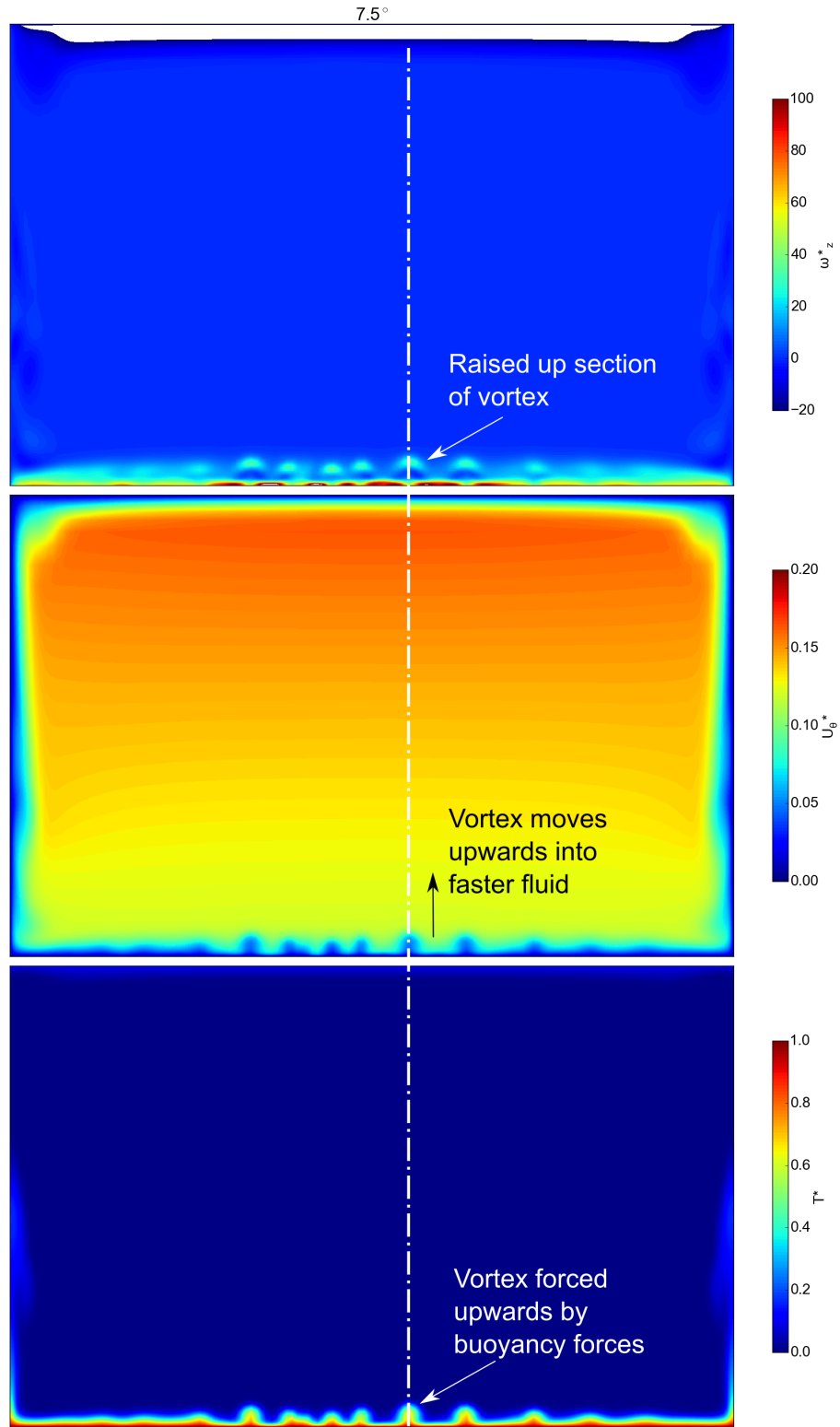


**Figure 6.4:** Schematic showing formation of streamwise vortices

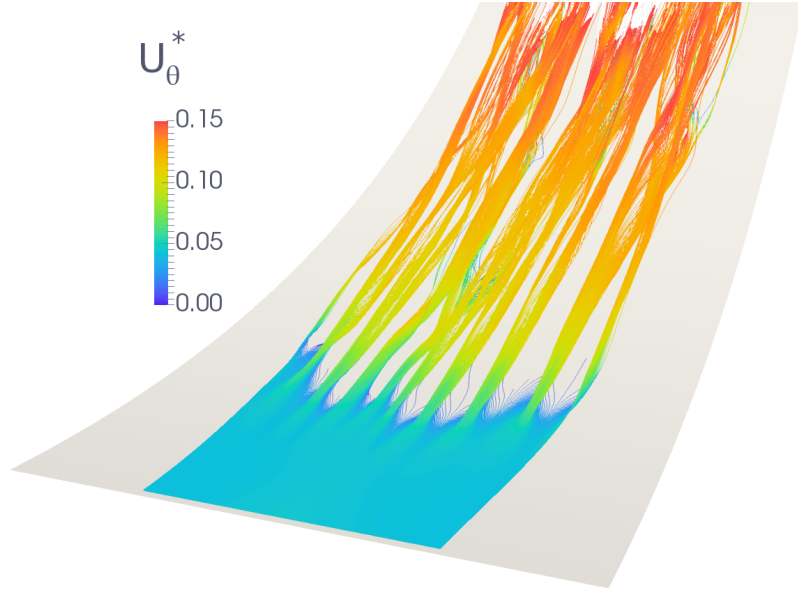
( $R$ ), the circumferential velocity has been non-dimensionalised by the buoyant velocity and the temperature is given as  $T^*$ . It can be seen that at several locations the boundary layer spanwise vortex has been raised up, in between these locations the vortex remains close to the outer wall. The raised up sections of the vortex are then in the faster moving fluid, which can be seen in the contour of streamwise velocity, and will move ahead of the sections of the vortex that remained closer to the wall. Finally this upwards movement of sections of the vortex can be seen to be driven by a section of rising hot fluid.

The effect of the rising boundary layer moving into faster moving fluid can also be seen in Fig.6.6. Stream-traces were released 5 mm from the start of the heated outer wall and coloured by non-dimensional circumferential velocity. Some of the traces can be seen to have moved further away from the wall than others and then have a higher circumferential velocity. These traces can then be seen to move ahead of the slower fluid and form the streamwise vortices.

When the boundary layer vortex is stretched and is turned streamwise the vortices form



**Figure 6.5:** Contours of non-dimensional  $z$  vorticity, streamwise velocity & temperature on a plane at  $7.5^\circ$

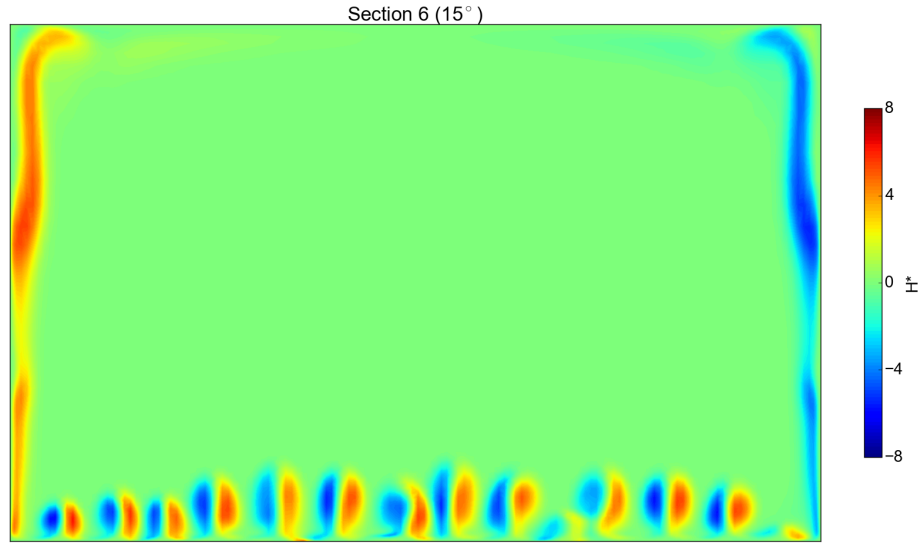


**Figure 6.6:** Stream-traces released 5 mm from outer wall, coloured by non-dimensional circumferential velocity

into counter rotating pairs. This can be seen by looking at contours of helicity further downstream at section 6, a location of  $15^\circ$  from the start of the outer wall, shown in Fig.6.7. Helicity contours show the location of streamwise vortex cores. The sign of the helicity also shows the direction of rotation of the vortices, in this case positive values represent a clockwise rotation looking into the rig. It can be seen that the vortices have formed into counter rotating pairs by the different signs of the helicity contours in each pair.

A similar vortex phenomenon is that of Görtler vortices, which can form when fluid flows over a concave surface [75]. The flow instability is due to centrifugal forces which act normal to the curving flow, this centrifugal force acts in a similar way to the buoyancy force acting normal to the wall. There will also be some centrifugal forces effecting the flow in the curved casing rig, but this can be shown to be small compared to the buoyancy forces. If the buoyancy force per unit volume is given by  $F_B = \rho_0 g \beta (T - T_0)$  and the centrifugal force per unit volume is given by  $F_{CF} = \rho V^2 / R$ , the ratio of buoyancy forces to centrifugal forces is given by  $F_B / F_{CF} = g \beta (T - T_0) R / V^2$ . For the casing rig case this comes out to  $\sim 60$ , showing that the centrifugal effect is small and the vortex formation can be said to be a buoyancy driven phenomenon.

The mechanism for the formation of the streamwise vortices has been proposed. The



**Figure 6.7:** Contours of non-dimensional helicity at section 6

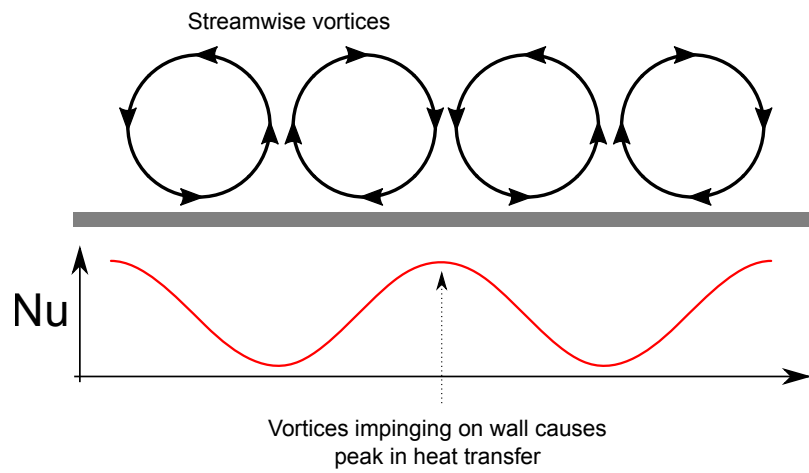
vortices require a buoyancy force that is normal and in the direction away from the wall in order to form. The mechanism that sets the initial wavelength is not yet understood.

### 6.3.2 Heat Transfer Enhancement

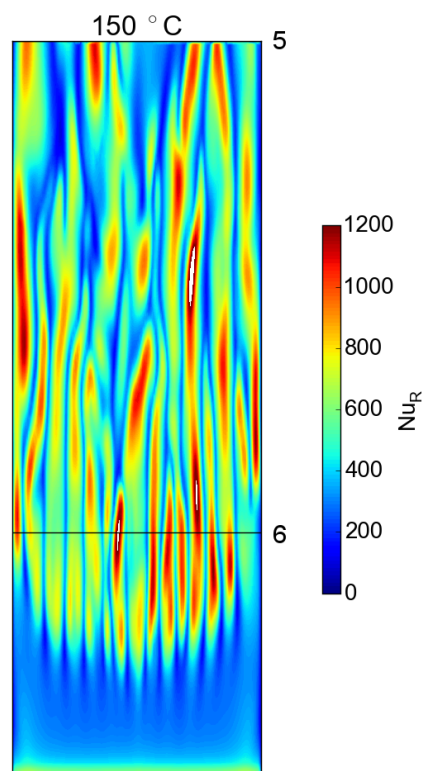
The streamwise vortices can locally have a significant impact on the wall heat transfer. When the rotating vortex pair impinges onto the heated wall the boundary layer becomes very thin and regions of high heat transfer occur. Conversely when the vortices move fluid away from the heated wall this causes a thickening of the thermal boundary layer giving reduced wall heat transfer. This effect is shown schematically in Fig.6.8. These regions of increased and reduced heat transfer give characteristic stripes in a contour of outer wall heat transfer, which can be seen in Fig.6.9.

Literature data for the local heat transfer is available from Jeschke et. al. [34]. At different streamwise Grashof Numbers the spanwise variations in Nusselt Number were plotted, it was found that the peak heat transfer was up to 1.6 times the spanwise average. A similar plot is shown in Fig.6.10, this is the variation in instantaneous wall Nusselt Number across the width of the casing rig. The heat transfer is plotted on a line  $15^\circ$  from the start of the heated wall, which is the location of section 6 of the rig. This location equates to a streamwise Rayleigh Number of  $Ra_s = 1.37 \times 10^9$  or a streamwise Grashof Number of

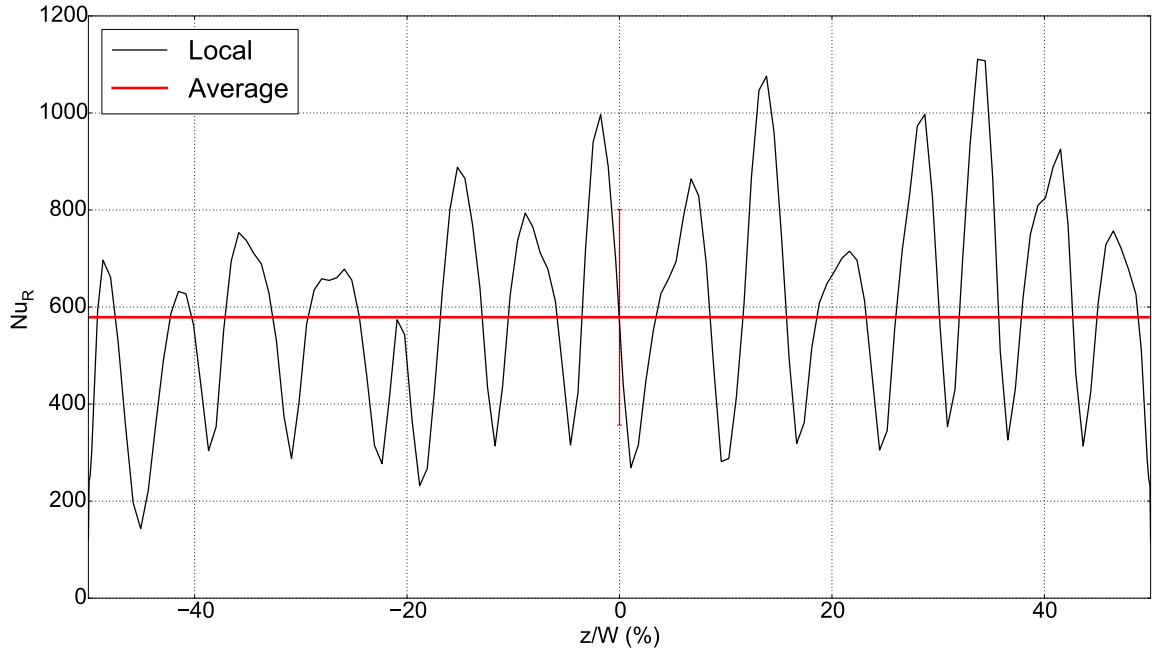




**Figure 6.8:** Schematic of heat transfer enhancement caused by streamwise vortices



**Figure 6.9:** Heat transfer in first  $45^\circ$  section of the outer wall



**Figure 6.10:** Outer wall heat transfer across width of duct on a line  $15^\circ$  from start of heated wall

$Gr_s = 1.92 \times 10^9$ . The spanwise coordinate  $z$  has been non-dimensionalised by the duct width  $W$  and the wall Nusselt Number has been calculated using the outer wall radius  $R$ .

The streamwise vortices can be seen to cause significant local variation in wall heat transfer across the duct width, with peak Nusselt Numbers over 1000 and the lower values below 300. In order to quantify the heat transfer enhancement the average ( $\overline{Nu}$ ) and standard deviation ( $\sigma$ ) across the width have been calculated. The ratio of heat transfer enhancement caused by the vortices is 2.25 at section 6. The ratio is defined as  $(\overline{Nu} + \sigma) / (\overline{Nu} - \sigma)$ .

### 6.3.3 Wall Temperature Study

Previous literature studies on the development of the streamwise vortices have found that the effect of wall temperature on the vortex wavelength ( $\Lambda$ ) is quite weak and follows a  $Gr^{1/4}$  trend. An investigation was conducted to see if this effect was also found using the numerical simulations of the casing rig.

The baseline simulation used an outer wall temperature of  $150^\circ\text{C}$  and this was compared to simulations run with wall temperatures of 50, 100, 200 &  $250^\circ\text{C}$ .

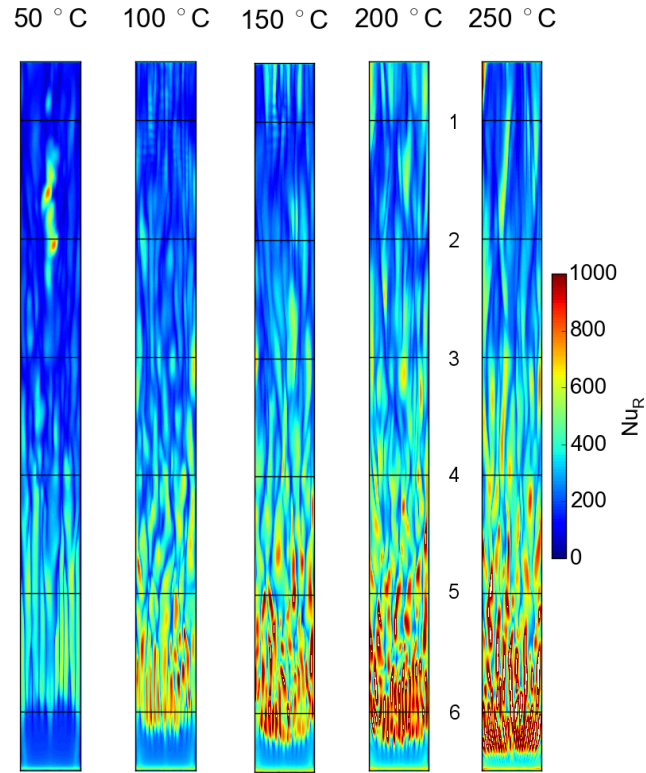
The instantaneous outer wall heat transfer for the different wall temperatures is shown

in Fig.6.11. It can be seen that as the outer wall temperature is increased the location of the onset of the vortices moves upstream and the vortex wavelength decreases. The strength of the vortices also increases with increasing wall temperature, this can be seen by the increased heat transfer caused by the vortices and that the effect of the vortices propagates further downstream. The increase in heat transfer due to the vortices is lowest in the 50°C case, this is due to the relatively small temperature difference between the wall and inlet temperature of 28°C, compared to the other cases. In this case the flow was found to separate from the inner wall of the rig and impinge on the outer wall, this is the cause of the peaks in heat transfer seen near section 2.

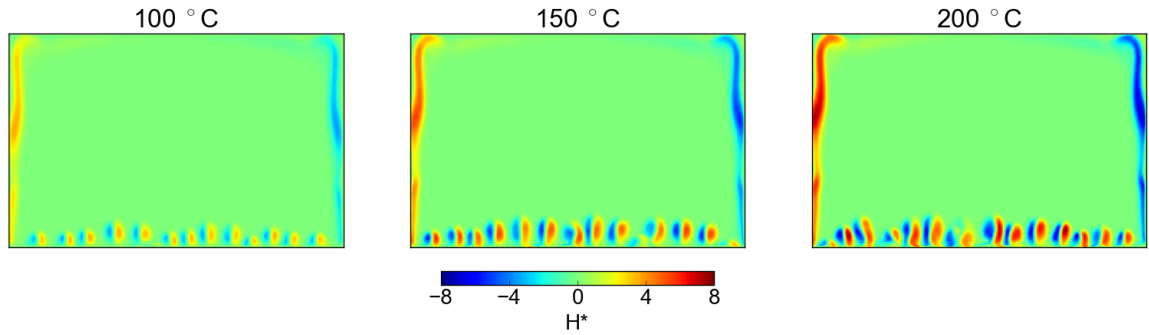
The wavelength of the vortices decreasing with increasing wall temperature can also be seen by looking at contours of helicity, shown at section 6 for three temperatures in Fig.6.12. The non-dimensional helicity shows 13 pairs of vortices for the 100°C case and 14 pairs for the 150°C and 200°C cases. The strength of the vortices in the 100°C case can be seen to be lower at section 6 due to them having had less time to develop compared to the other cases.

The data for the vortex wavelength at the different wall temperatures is given in Table.6.1. The average wavelength was calculated by dividing the width of the casing rig (502 mm) by the number of visible vortex pairs. Also given in Table.6.1 is the critical length  $x_c$ , this is the arc length from the start of the heated wall to the point where the vortices started to effect the wall heat transfer. This critical length can then be used to calculate the critical Grashof Number ( $Gr_c$ ). The fluid properties used to calculate the Grashof Number were based on the film temperature, the average of the outer wall and inlet temperatures.

The vortex wavelength for the different wall temperatures is plotted against the critical Grashof Number in Fig.6.13, and the data is compared to experimental correlations from Cheng and Kim [12]. It can be seen that the results from the LES simulations follow the same  $Gr_c^{1/4}$  trend as the correlation. The results are very similar considering they come from different rigs with different geometries. The different rigs would also have had different inlet boundary layers, which could also have effected the transition location.



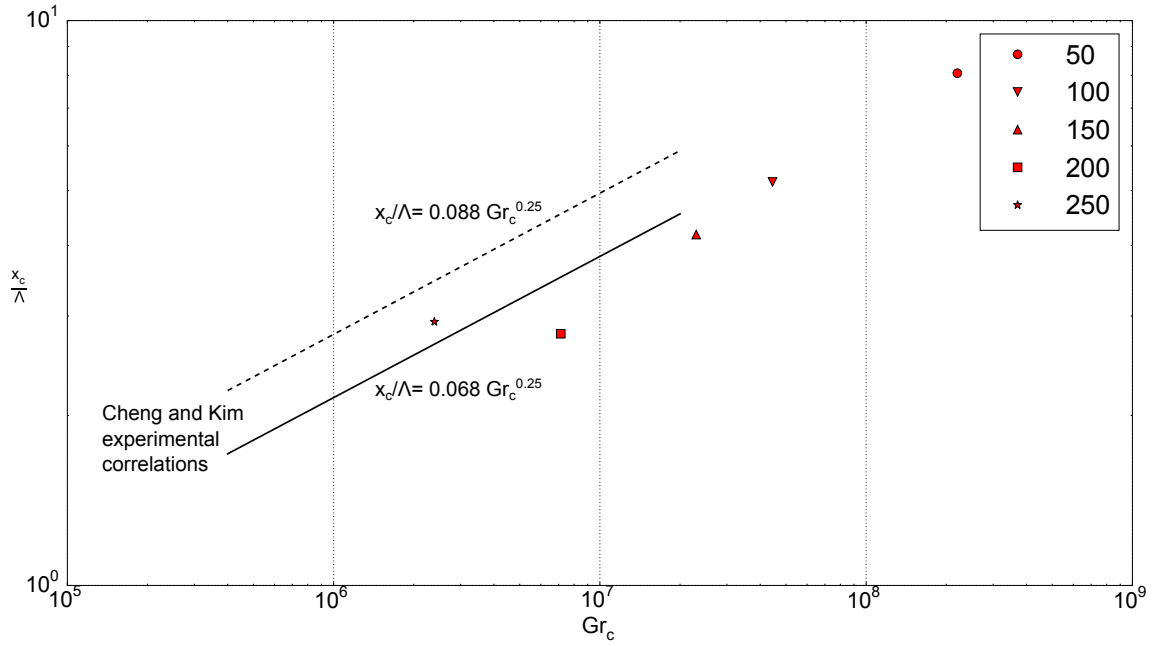
**Figure 6.11:** Contours of instantaneous non-dimensional outer wall heat transfer at different wall temperatures



**Figure 6.12:** Instantaneous contours of helicity at section 6 at different wall temperatures

| Wall Temp (°C) | Vortex Pairs | $\Lambda$ (mm) | $x_c$ (mm) | $Gr_c$             |
|----------------|--------------|----------------|------------|--------------------|
| 50             | 9            | 55.8           | 450        | $2.20 \times 10^8$ |
| 100            | 13           | 38.6           | 200        | $4.45 \times 10^7$ |
| 150            | 14           | 35.9           | 150        | $2.30 \times 10^7$ |
| 200            | 14           | 35.9           | 100        | $7.13 \times 10^6$ |
| 250            | 21           | 23.9           | 70         | $2.39 \times 10^6$ |

**Table 6.1:** Wavelength and critical distance for streamwise vortices at different wall temperatures



**Figure 6.13:** Vortex wavelength against critical Grashof Number for different wall temperatures

### 6.3.4 Prandtl Number study

It was found that the effect of outer wall temperature on the vortex wavelength was quite weak and followed a  $Gr_c^{1/4}$  trend. It was thought that this change in wavelength could be due to variations in fluid properties such as the fluid viscosity or conductivity with temperature. These changes of fluid properties with temperature will effect the Prandtl number of the fluid.

An investigation was conducted running simulations with a wall temperature of 150°C but with fluid properties that had been scaled to alter the Prandtl number. Four cases were run, one with the dynamic viscosity ( $\mu$ ) double that of air and one with half that of air, keeping the other properties constant. The other two cases scaled the thermal conductivity ( $\lambda$ ) to that of double and half that of air. As the Prandtl Number is given by  $Pr = c_p \mu / \lambda$ , these changes will either double or halve the Prandtl Number. Apart from the changes to the fluid properties the simulations were the same as the baseline case and were run in the same way.

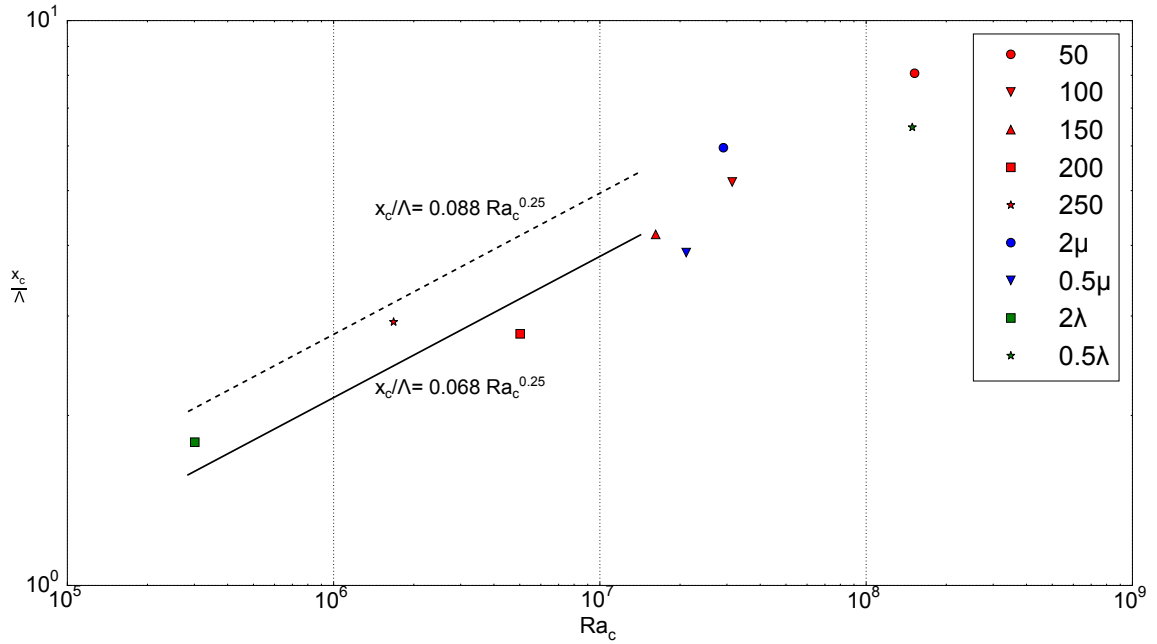
The vortex wavelength and critical length were extracted from the simulations as described previously and the results plotted in Fig.6.14. It was found that there was better col-

lapse of the data when plotted against the critical Rayleigh Number as opposed to Grashof Number. The results from the cases with different wall temperatures were found to follow a trend of  $Ra_c^{1/4}$  and the cases with varying fluid properties was also found to follow this trend.

The outer wall heat transfer with varying fluid properties can be seen in Fig.6.15. The effect of doubling and halving the dynamic viscosity can be seen to have had a relatively small effect on the vortex wavelength. The lower viscosity case gave a shorter wavelength forming 15 pairs of vortices in the rig, compared to the 14 pairs for the baseline case and 13 for the higher viscosity case. A larger effect on the wavelength was found when the thermal conductivity was varied, with a higher conductivity giving a shorter wavelength. In the higher conductivity case 18 vortex pairs were formed compared to the 13 pairs from the lower conductivity case. The effect of varying the fluid properties on the critical distance ( $x_c$ ) can also be seen in Fig.6.15. The viscosity had a small effect on the critical distance, with reducing viscosity reducing the critical distance. The effect from doubling and halving the thermal conductivity was more significant, when the thermal conductivity was doubled the critical distance reduced to 50 mm compared to 150 mm in the baseline case. The distance at which the vortices break down and the flow becomes turbulent can also be seen to have decreased. When the conductivity was halved the critical distance increased and the effect of the vortices on the wall heat transfer can be seen to propagate all of the way to the exit of the rig. The data from varying the fluid properties can be seen in Table.6.2.

These results showed that the viscosity had less of an effect on the vortices than the thermal conductivity. This suggest that the initial spanwise vortex formation, which would be affected by the viscosity and the momentum boundary layer, play a less important role in the formation of the streamwise vortices. This is in comparison to the buoyancy effects, which are driven by the wall heat transfer and affected by the thermal conductivity.

The overall effect of doubling and halving the fluid Prandtl Number by changing the viscosity and conductivity was lower than expected. The changes in Prandtl number caused by the wall temperature change were much smaller than the factor of 2 changes that were investigated, going from  $Pr = 0.712$  at  $50^\circ\text{C}$  to  $Pr = 0.703$  at  $250^\circ\text{C}$ . But the magnitude of the effect from changing the wall temperature was similar to the effect from the large



**Figure 6.14:** Vortex wavelength against critical Rayleigh Number varying fluid properties

| Case         | Vortex Pairs | $\Lambda$ (mm) | $x_c$ (mm) | $Gr_c$             |
|--------------|--------------|----------------|------------|--------------------|
| $2\mu$       | 13           | 38.6           | 230        | $2.08 \times 10^7$ |
| $0.5\mu$     | 15           | 33.5           | 130        | $6.00 \times 10^7$ |
| $2\lambda$   | 18           | 27.9           | 50         | $8.53 \times 10^5$ |
| $0.5\lambda$ | 13           | 38.6           | 250        | $1.07 \times 10^8$ |

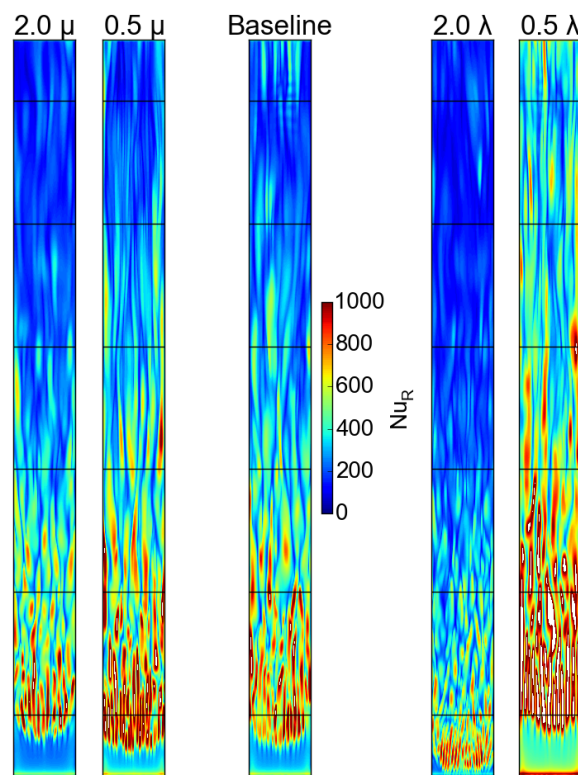
**Table 6.2:** Wavelength and critical distance for streamwise vortices with different fluid properties

changes in Prandtl number. This shows that the effect of the vortex wavelength changing with temperature cannot be put down to changes in fluid properties alone.

### 6.3.5 Heated Inner Wall

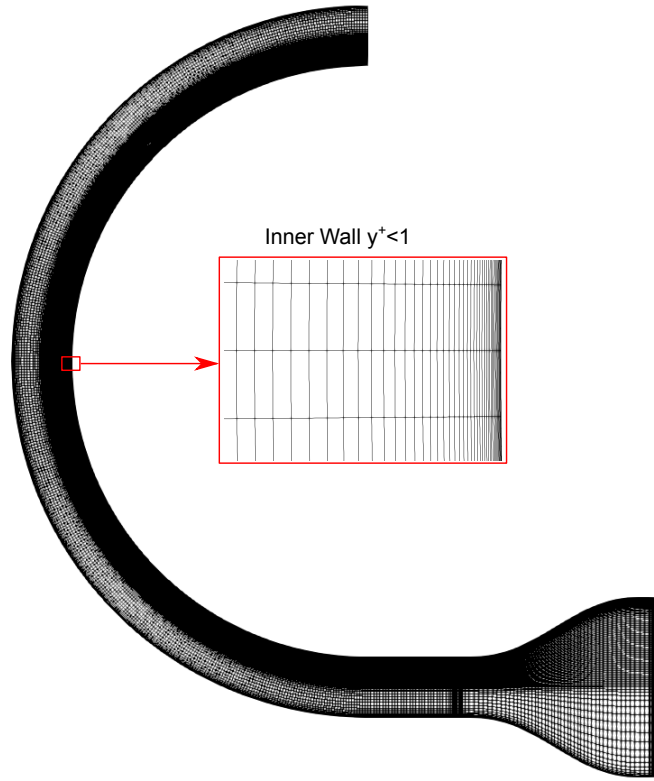
The vortices formed when the flow was on top of the heated outer surface, so the case of when the flow is underneath a heated surface was then investigated. A simulation was conducted using the casing rig geometry but with the inner wall heated and the outer wall unheated.

The boundary conditions for the simulation set the inner wall to a uniform 150°C and the outer wall to an adiabatic boundary condition. The computational mesh used for the simulation was modified from the previous test cases, the mesh is shown in Fig.6.16. The



**Figure 6.15:** Contours of instantaneous non-dimensional outer wall heat transfer with varying fluid properties





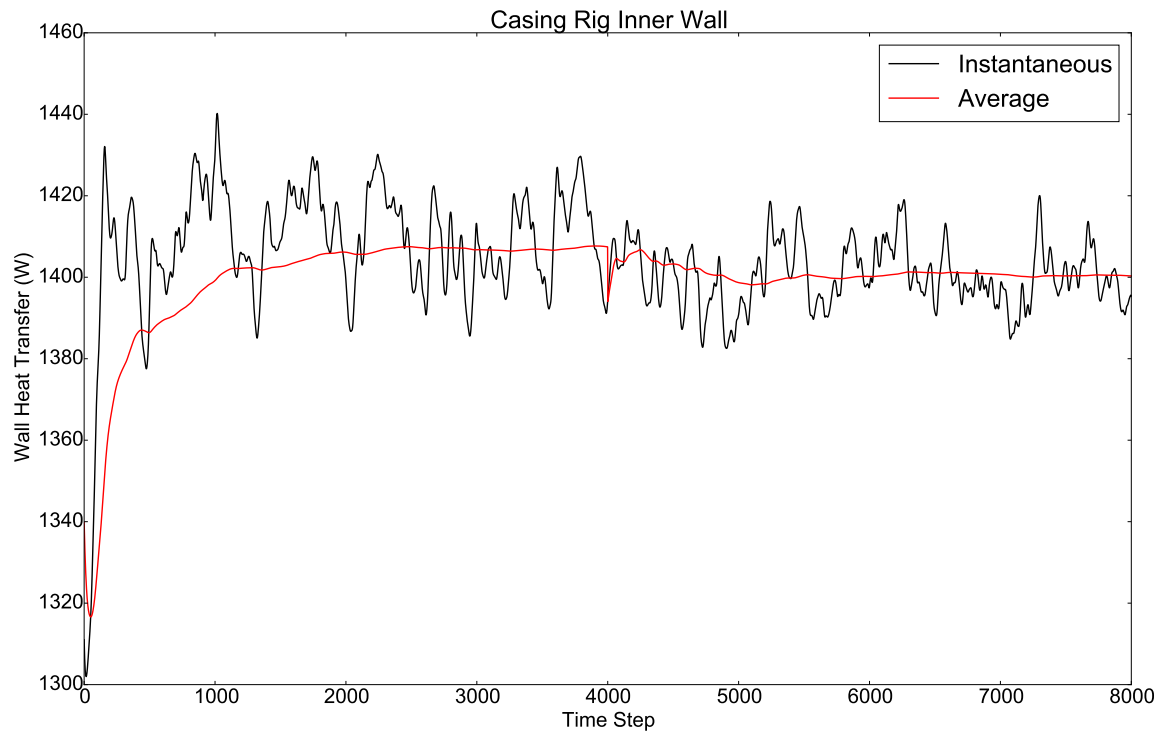
**Figure 6.16:** Cross section of mesh used for heated inner wall case

mesh still used  $7.0 \times 10^6$  cells but the cells were now clustered towards the inner wall instead of the outer wall, this kept the inner wall  $y^+ < 1$ .

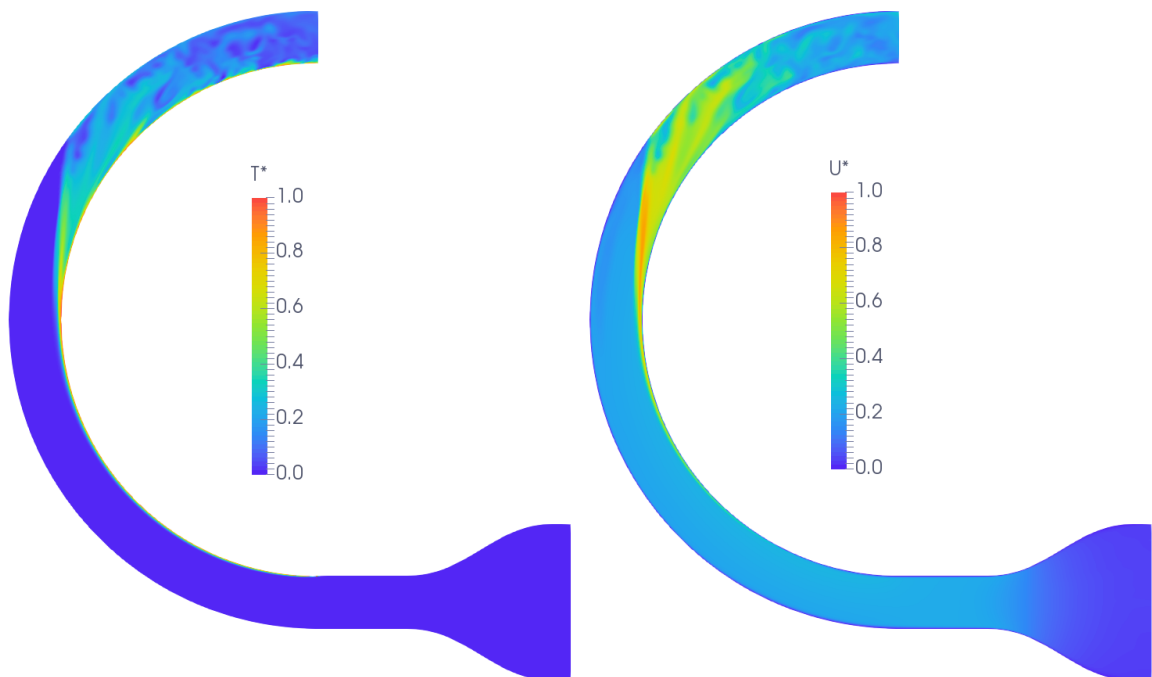
The heated inner wall simulation required a smaller time step in order to remain stable, compared to the heated outer wall cases. The heated inner wall simulation used a time step of 0.01 s compared to the 0.1 s used previously. The simulation was run for 4000 time steps to allow for the flowfield to develop with averages then taken over a further 4000 time steps. The inner wall heat transfer was monitored throughout the simulation to check for convergence, this history monitor is shown in Fig.6.17.

The temperature and velocity field from the heated inner wall simulation are shown in Fig.6.18. It was found that the flowfield was very different from that in the heated outer wall case. Whereas the flow close to the heated outer wall stayed attached, in the heated inner wall case the flow separates from the inner wall, giving a large separated region in the top half of the rig.

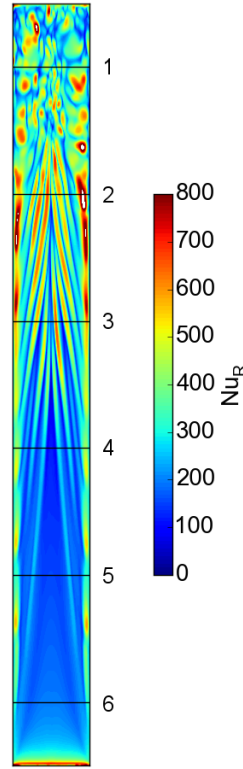
This separated region at the top of the rig can be seen by looking at the inner wall heat



**Figure 6.17:** History monitor of inner wall heat transfer



**Figure 6.18:** Cross section of non-dimensional temperature and velocity from heated inner wall case

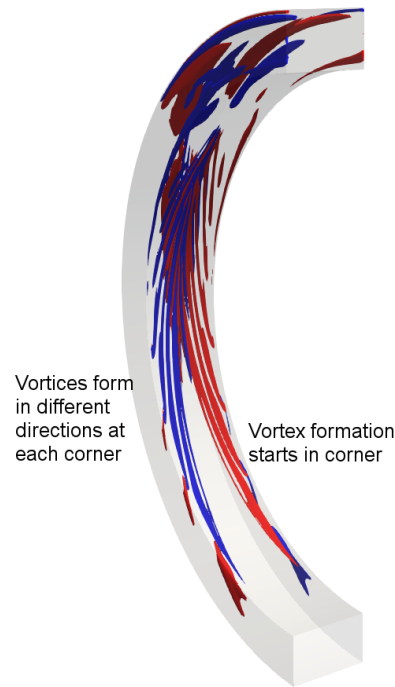


**Figure 6.19:** Inner wall non-dimensional heat transfer

transfer. It is visible in Fig.6.19 as the region with very non-uniform heat transfer near section 1.

Also visible in the heat transfer contour further upstream in the rig is a distinctive herringbone pattern in the heat transfer. This pattern is caused by a vortex phenomenon which originates at the side-walls of the rig and migrates towards the centreline. These vortices are a form of secondary flow which form in curved channels. These Dean vortices [52] were investigated for the flow in a curved rectangular duct by Thangam and Hur [90]. The important non-dimensional parameter for predicting the formation of the vortices is the Dean Number, given by  $De_H = Re_H C_H^{-0.5}$  where the Reynolds Number is given by  $Re_H = UH/\nu$  and  $C_H$  is the curvature ratio  $R_{inner}/H$ . For this heated inner wall case the curvature ratio is  $1.575/0.32 = 4.9$  and the Reynolds Number is  $1.1 \times 10^4$  this gives a Dean number of 5000 and puts the flow into an unstable region where the vortices are expected to occur.

These vortices can be visualised using iso-surfaces of helicity, shown in Fig.6.20. It



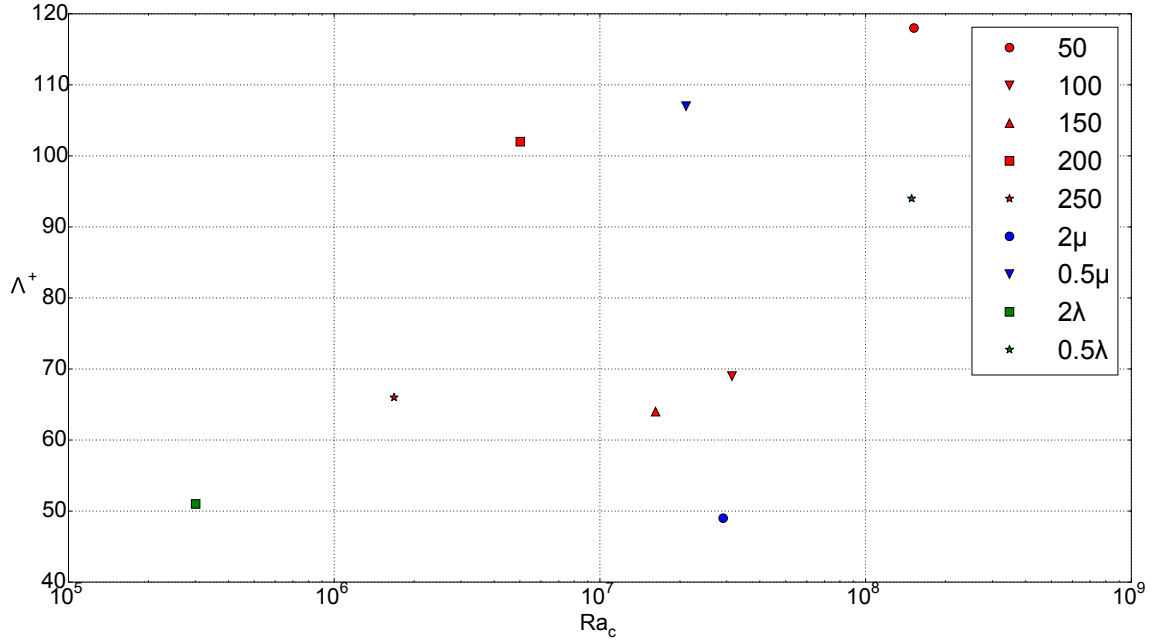
**Figure 6.20:** Iso-surfaces of helicity caused by secondary flows from heated inner wall case

can be seen that the vortices originate in the corner between the heated inner wall and the side walls, this is in contrast to the streamwise vortices which form throughout the width of the rig. As one vortex forms and moves away from the side wall and towards the centreline another vortex forms in the corner downstream of the first, this in turn leads to the herringbone pattern. The direction of rotation of the vortices formed at the two corners are different, shown by the different colours of the iso-surfaces, this is different from the streamwise vortices which form as a counter-rotating pair.

It has therefore been shown that the streamwise vortices did not occur in the heated inner wall case. Instead a form of secondary flow occurred in the curved duct which also had an effect on the wall heat transfer.

## 6.4 Comparison with Hairpin Vortices

The streamwise vortices show some similarity to hairpin or horseshoe vortices that form in the viscous sublayer of a boundary layer [22]. The formation mechanism for the hairpin vortices is due to stretching of a transverse vortex element as described by Kline et al. [41].



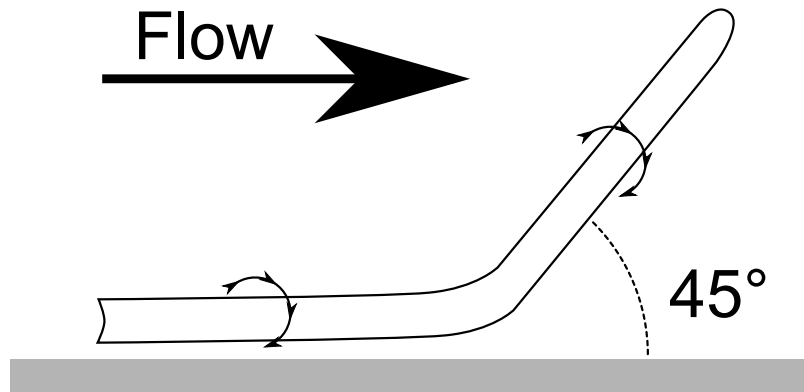
**Figure 6.21:** Vortex wavelength in wall units against critical Rayleigh number

This is very similar to the proposed mechanism for the formation of the streamwise vortices found in the casing rig. It is therefore thought that the streamwise vortices could be a form of hairpin vortices.

Firstly the wavelength of the hairpin vortices will be compared to that of the streamwise vortices. The wavelength of the hairpin vortices in wall units is typically  $\Lambda^+ \approx 100$ , [41, 32] where  $\Lambda^+ = \Lambda U_\tau / \nu$ . The wavelength of the streamwise vortices in the casing rig was calculated in terms of wall units, using the wall shear stress at the point where the vortices became visible.

The results are plotted against the critical Rayleigh number in Fig.6.21. It can be seen that the  $\Lambda^+$  values are of the same order as the hairpin vortices, with the average value being 80, but with a large amount of scatter. A large variation in wavelength from  $\Lambda^+ = 20 - 200$  was observed however by Oldaker and Tiederman [62], from experiments in a water channel. The similarity of the wavelength to that formed by the hairpin vortices suggests the two mechanisms may be linked.

A difference in the behaviour of the hairpin vortices to the streamwise vortices occurs as the vortices propagate downstream. The streamwise vortices stay close to and parallel to



**Figure 6.22:** Schematic of hairpin vortex

the wall, whereas the hairpin vortices lift up and make an angle of around  $45^\circ$  to the wall. This lifting of the hairpin vortex was demonstrated by Head and Bandyopadhyay [26] and is shown schematically in Fig.6.22.

The theory that hairpin vortices are a type of Görtler instability has been disputed by Jang et al. [32]. The different downstream behaviour of the streamwise vortices and hairpin vortices would also suggest this. It was also shown with the heated inner wall case that a buoyancy force away from the wall is required for the vortices to form, this is not the case for hairpin vortices. The fact that the wavelength in wall units of the streamwise vortices is similar to that of the hairpin vortices does suggest some link between the two. It could be the case that the initial wavelength in the spanwise vortex is set by the same mechanism as the hairpin vortices, before the buoyancy forces start to dominate the vortex development.

## 6.5 Conclusions

An investigation into the formation of streamwise vortices was conducted in order to better understand the physical mechanism of how the vortices form. Large eddy simulations were conducted using the casing rig geometry which allowed for different boundary conditions and flow parameters to be modified.

A proposed physical mechanism for the formation of the streamwise vortices was described. The spanwise boundary layer vortex is raised up at certain points by the action of buoyancy forces. The raised up sections of the vortex are then in faster moving fluid and move ahead of the lower sections of the vortex. This action causes the spanwise vortex to

become stretched and turn streamwise, forming the counter rotating vortex pairs.

The effect of the heat transfer enhancement due to the effect of the vortices was shown. There was significant variation in heat transfer across the duct width, with the peaks in heat transfer being over twice that of the lower regions of heat transfer.

The effect of wall temperature on the wavelength of the vortices was investigated. Previous studies in the literature found a relationship of wavelength to  $Gr^{1/4}$ , an investigation was conducted to see if this relationship also occurred in the numerical simulations of the casing rig. Simulations with wall temperatures varying from 50°C to 250°C were run and compared with the experimental correlation. It was found that there was good agreement between the two datasets, giving confidence in the simulations.

It was thought that the variation in vortex wavelength with temperature could be down to the effect of changing fluid properties with temperature. A study was then conducted varying the Prandtl number of the fluid in the simulation. This was achieved by doubling or halving the fluid viscosity or thermal conductivity. The effect on the vortex wavelength obtained by changing the viscosity was relatively small but changing the conductivity had a more significant effect. The changes in wavelength were lower than expected which shows that the wavelength changes with temperature cannot be put down to changes in fluid properties alone.

A final investigation was to investigate if vortices would form when flowing underneath a heated surface. A case was run with the inner wall of the rig heated to 150°C and the outer wall unheated. It was found that the streamwise vortices did not form but instead a form of secondary flow occurred. This secondary flow vortex gave a distinctive herringbone pattern in the contour of wall heat transfer. This investigation shows that the flow does need to be on top of a heated surface in order for the streamwise vortices to form.

The streamwise vortices were then compared to the similar hairpin vortices found in boundary layers. Whilst the wavelength of the two types of vortex was similar, the vortex behaviour was different, meaning that the streamwise vortices are unlikely to be a form of hairpin vortices.

## Chapter 7

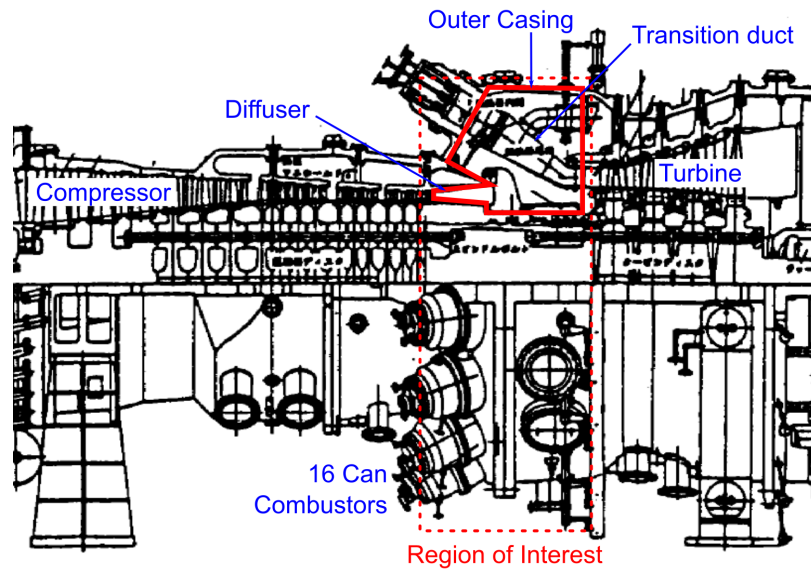
# Shutdown Flow Control in Gas Turbines

*During the shutdown of a gas turbine, natural convection flows occur inside of the engine casing. These natural convection flows cause the top of the engine casing to cool slower than the bottom of the casing. This non-uniform cooling can cause the casing to distort, known as “cat back” distortion. This distortion leads to reduced tip and seal clearances which could cause rubbing and binding. In this study different methods for reducing the non-uniform casing heat transfer in an industrial gas turbine during shutdown have been investigated. A baseline case with no active method for reducing the casing heat transfer has been compared with purge and flow extraction methods. It has been found that the new proposed extraction method improves the uniformity of the casing heat transfer during the initial shutdown period.*

### 7.1 Introduction

Natural convection flows occur inside of gas turbine casings during shutdown. These flows cause non-uniform cooling of the casing and cause the casing to distort, known as “cat-back” distortion. This distortion of the casing is undesirable as it can cause rubbing and binding of seals and rotor blades, reducing the operational flexibility of the engine. In order to reduce the casing distortion during shutdown, methods to make the casing heat transfer more uniform have been analysed. If the shutdown heat transfer can be made more uniform and the casing distortion reduced, rotor tip and seal clearances could potentially be reduced.





**Figure 7.1:** Cross Section of Gas Turbine Casing, modified from [59]

This could lead to improved engine efficiency and increased compressor stability.

A cross section of a large industrial gas turbine is shown in Fig.7.1. The region of interest is the large cavity inside of the outer casing, which surrounds the combustor cans. Immediately after a gas turbine is shut down the outer casing is at a temperature of around 350 °C. The outer casing is a relatively thick cast component which takes days to cool down [53]. This is compared to the combustor cans which are made of much thinner metal and cool more quickly, giving temperature differences of over 100 °C between these components. The different temperatures inside of the engine casing cause buoyancy driven flows to occur. There is also a flow of air through the engine driven by the draught from the chimney downstream of the engine. These flows then determine the convective heat transfer inside the engine casing.

The flowfield inside of the engine is shown schematically in Fig.7.2. In areas where the flow inside of the engine casing impinges on the outer wall, the heat transfer from the hot outer casing is increased. At the top of the engine a region of thermally stratified flow occurs below the outer casing; in this stratified flow region the heat transfer from the casing is reduced. The low heat transfer in the stratified region and higher heat transfer in the impingement regions causes the outer casing to cool non-uniformly. This non-uniform

cooling then leads to “cat-back” distortion of the casing.

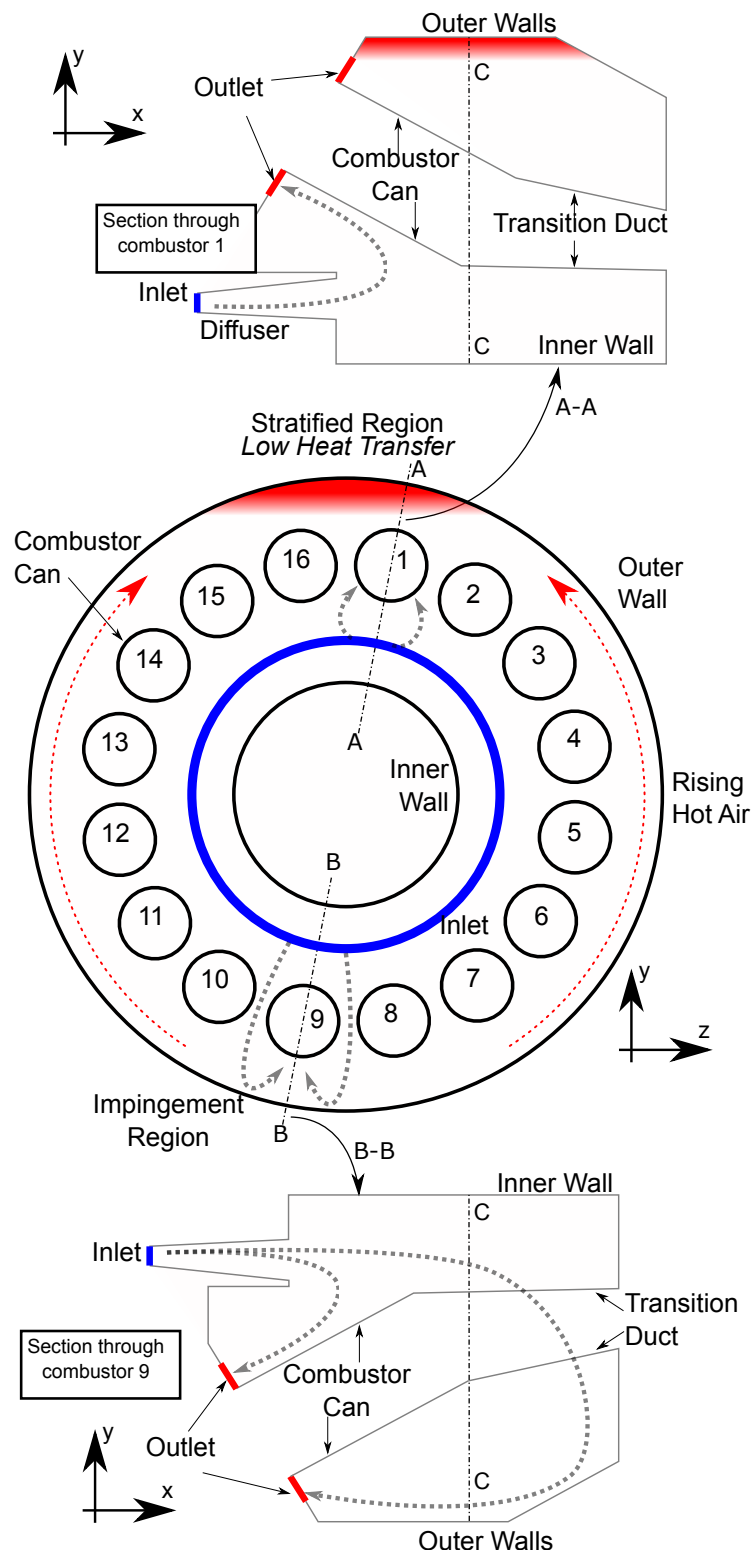
## 7.2 Investigation

This study involved conducting simulations looking at the effectiveness of different methods for reducing the casing distortion in an engine during shutdown. The simulations were used to predict the flow and heat transfer inside of a 3D geometry representative of a large industrial gas turbine casing. The different methods for reducing distortion were evaluated based on the uniformity of the heat transfer around the casing, as a larger non-uniformity would give rise to greater casing distortion. This method of conducting 3D simulations of natural convection shutdown flows is not common in the literature or industry. Instead correlations are often used to obtain the casing heat transfer [24] however correlations cannot easily predict changes in heat transfer due to changes in design.

The baseline case for this study had no active method of controlling the natural convection flows occurring inside of the gas turbine casing. Without any active controlling method it would be expected that a region of thermally stratified flow would occur at the top of the casing. This stratified layer would give low heat transfer from the outer casing.

In order to reduce the non-uniform cooling during shutdown a purge system may be employed. The purge system injects cool air into the top of the engine through a hole in the outer casing. The aim of this system is to break up the stratified layer of air at the top of the casing, hence increasing the heat transfer in this region. Experience from engines has found that the purge system does not completely eliminate the casing distortion and it would be desirable to reduce this further. A purge case simulation was conducted in order to observe the heat transfer distribution using this system.

A proposed method for controlling the outer casing heat transfer is an extraction based method. Instead of injecting air into the casing, flow is extracted from the casing, with the aim of removing the thermally stratified layer of air. One feature of the extraction method is that it could potentially be retrofitted to existing engines if they are using the purge system. The flow would be extracted through the hole in the casing used for the purge flow, with modifications limited to the external pipework.



**Figure 7.2:** Schematic Showing Axial and Radial Sections of a Gas Turbine Casing

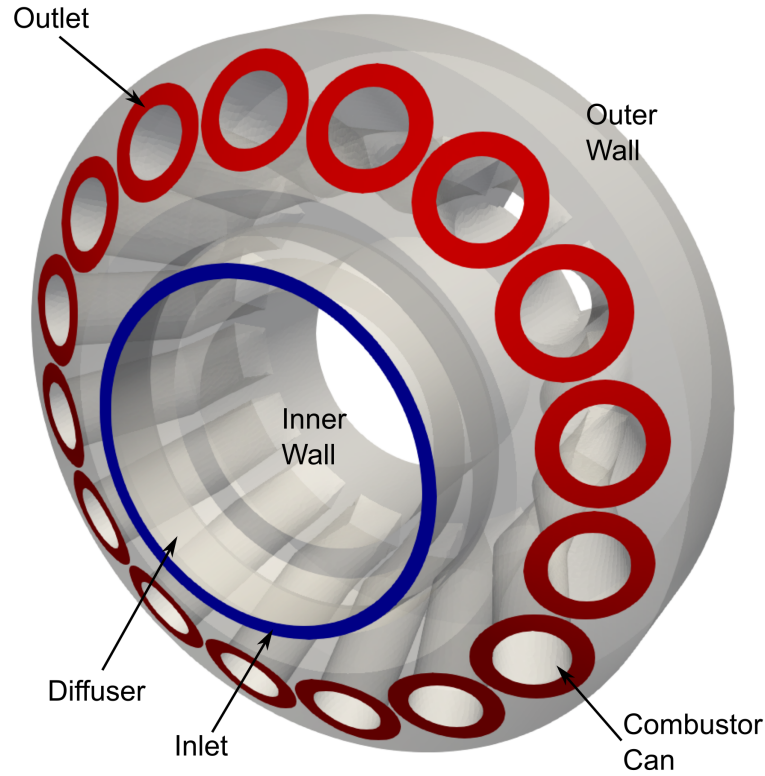
## 7.3 Test Case

The geometry used for this study, shown in Fig.7.3, is representative of a large industrial gas turbine. Previous studies of natural convection flows tend to use very simple geometries such as the heated concentric cylinder [43], the heated cavity [6] or the heated wall [78, 97]. The inclusion of complexities into the geometry, such as the combustor cans and transition ducts, can cause effects such as the blockage and local acceleration of the flow. This can cause a significant difference to the flowfield over the simple test cases.

The geometry consists of a casing outer wall of 3.7 m diameter and an inner wall of 1.32 m diameter, between these is the main flow cavity. The flow enters the cavity through an inlet representing the flow from the compressor exit, followed by a diffuser into the main cavity. From the main cavity the flow would then go into the 16 combustor cans, which are represented by the 16 circular exits in the front wall of the casing.

The boundary conditions used in the transient simulations are representative of those from an industrial gas turbine just after shutdown. Over the simulation time period of 50 seconds the wall and inlet temperatures are assumed to be constant, as this is much shorter than the shutdown period of several days. The heat fluxes calculated during this period can be used to determine if distortion is likely to occur, by observing the uniformity of the heat flux profile around the casing. As the gas turbine slowly cools the temperatures are likely to become circumferentially non-uniform. It is believed that the inlet temperature coming from the compressor exit would develop a temperature profile, with hotter fluid towards the top of the inlet. This temperature profile could well exacerbate any thermal stratification inside of the outer casing.

The walls all had constant temperature boundary conditions applied, the outer walls used 350 °C and all of the other walls used 200 °C. The inlet flow from the compressor was also at a constant 200 °C, with the inlet being modelled as a constant pressure inlet. The effect of the draught through the engine from the chimney was also included in the simulations, this flow was modelled by a 1 kg/s target mass flow rate applied at the combustor outlet. For the cases with a purge flow, this was modelled using a mass flow inlet boundary condition with an inlet temperature of 20 °C. For cases using flow extraction, this



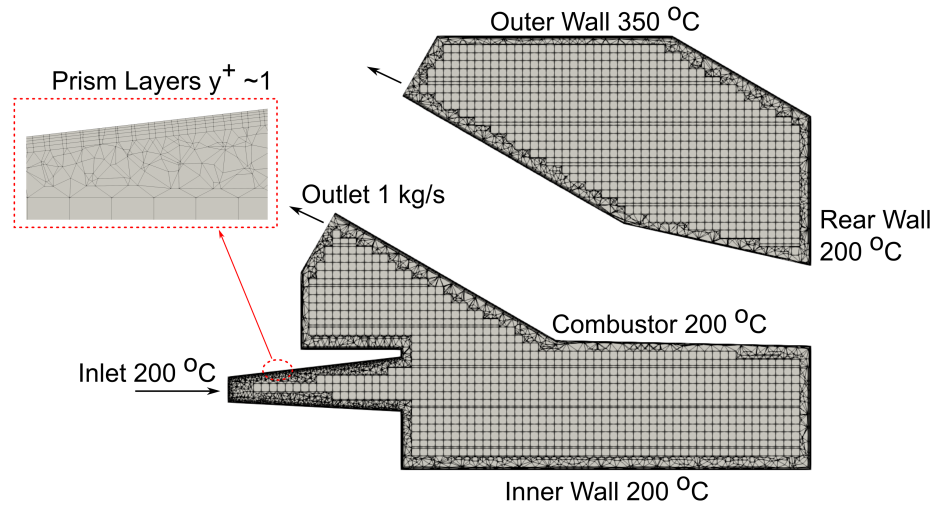
**Figure 7.3:** Computational Domain

was modelled using a pressure outlet boundary condition with a target mass flow rate.

The Rayleigh Number ( $\frac{g\beta\Delta TR^3}{\nu\alpha}$ ) for this case is  $4.8 \times 10^{10}$ , based on the outer wall radius of 1.85 m, a  $\Delta T$  of 150 °C and a reference temperature of 275 °C. This puts the flow into the turbulent natural convection regime. As the case also features a forced flow through the outlet, representing the draught from the chimney, then the heat transfer may be a mixed convection case. To determine if the case is in mixed or natural convection the Richardson number is calculated. Based on the outlet mass flow rate of 1 kg/s and the outlet area of 2.51 m<sup>2</sup> the outlet velocity is 0.617 m/s, the Richardson number is therefore 13.0, which puts the case just into the mixed convection regime.

## 7.4 Numerical Method

The simulations were performed using ANSYS Fluent v15, using the pressure based solver with the  $k-\omega$  SST turbulence model [4]. The SGDh turbulent heat flux model was used for most of the simulations, with the GGDH and GGDH+ models also being tested, see section



**Figure 7.4:** Cross Section of Mesh

| Case       | Mesh Cell Count | Total Outer Wall Heat Transfer (W) |
|------------|-----------------|------------------------------------|
| Baseline   | 4.6 M           | 12 045                             |
| Refined    | 8.8 M           | 12 161                             |
| Difference |                 | −0.95%                             |

**Table 7.1:** Mesh independence study.

7.6. A second order upwind scheme was used for spatial discretization and a second order implicit time stepping scheme was used. The simulations were initially run in a steady manner, the results of which were used as the initial conditions for an unsteady simulation. A time-step of 50 ms was used for the simulations, which were run for 1000 time-steps to allow for calculation transients to decay, before averages were calculated over a further 1000 time-steps. The meshes used in the study consisted of 4.6 million cells and were an unstructured tetrahedral type with prism layers in the boundary layer and a hexahedral core, Fig.7.4. The prism layers kept the wall  $y^+$  around 1 or lower. Simulations were also performed using an 8.8 million cell mesh and the results were found to agree well with the baseline 4.6 million cell mesh, Table.7.1.

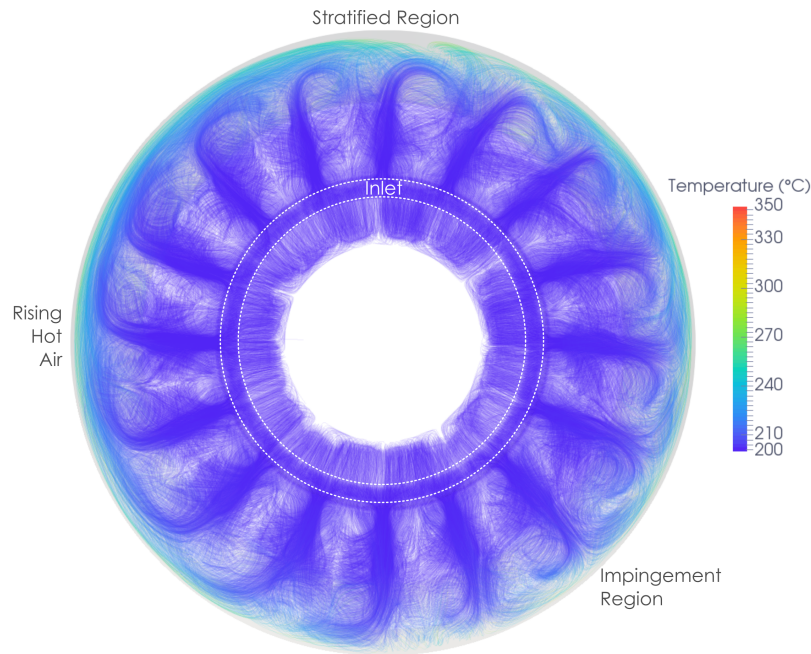
## 7.5 Baseline Investigation

The baseline case for the study used the gas turbine geometry with no active method for controlling the shutdown casing heat transfer. The only inlet into the domain was from the compressor exit and the only outlet from the domain was the combustor outlet.

The flowfield in the baseline case is three-dimensional and is driven by a combination of buoyancy forces and the draught through the engine. The flow can be visualised by looking at time averaged stream-traces originating from the inlet and coloured by temperature, Fig.7.5. The flow enters the domain through the inlet and diffuser and flows into the main cavity. The flow travels radially outward from the diffuser exit and is drawn towards the combustor outlets. The flow also reverses direction axially as it moves towards the outlet, this flow reversal in the axial direction can be visualised by velocity vectors in Fig.7.6, in a section showing the top of the cavity. As the flow moves radially outwards some of the flow is accelerated through the small gaps between combustor cans. This can be seen in Fig.7.7, showing the velocity vectors in the plane of axial section C-C. These accelerated jets impinge on the outer wall causing locally enhanced heat transfer, as can be seen in Fig.7.8.

Also visible in Fig.7.5 is that the flow is not symmetric about the vertical axis but the fluid has a preferred direction around the combustors. As the fluid takes one direction around the combustor this influences the fluid in the adjacent combustors to also flow in this direction. This non-symmetry in the natural convection flowfield for flows with symmetrical geometry and boundary conditions has also been observed by Holtzman et. al. [28] and Yuan et. al. [107]. It was noted by Holtzman et. al. that the symmetrical flowfield is not stable above a critical Grashof Number. There is only a small difference in heat transfer distribution between the left and right hand sides of the casing due to the flow asymmetry, this can be seen in Fig.7.9, with the difference in the total heat transfer between the left and right sides being under 0.6%. In both sides of the casing the trend of low heat transfer at the top of the casing compared to the bottom of the casing is clear.

As well as the draught through the casing the flow is also driven by buoyancy forces, in particular near the hot outer casing. The rising fluid driven by buoyancy forces can be seen



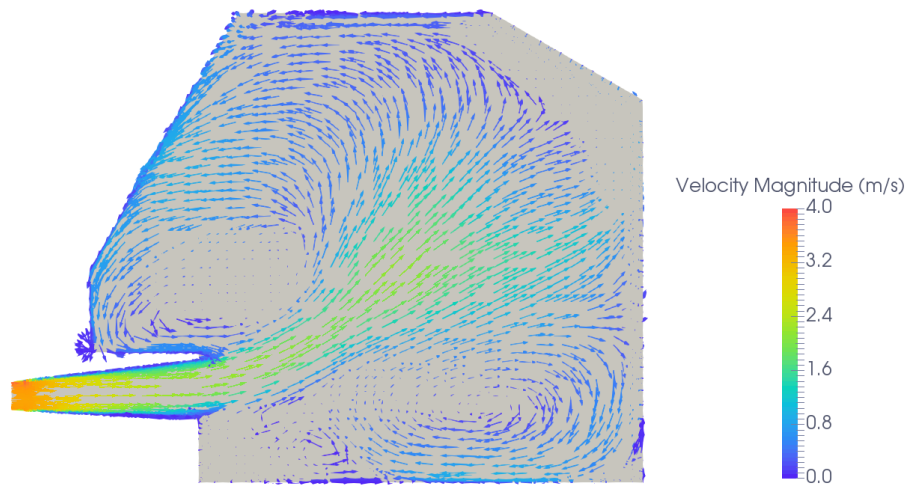
**Figure 7.5:** Baseline Case, Time Averaged Stream-Traces, looking rearward into cavity

in Fig.7.7. It can be seen looking at the surface streamlines in Fig.7.8 that the fluid tends to rise close to the cylindrical part of the outer wall, it is diverted away from the conical part of the outer wall by the impinging jet flow. At the top of the casing a thermally stratified region occurs, where the buoyancy forces are reducing the flow velocity, this can be seen by the low velocity region with few stream-traces in Fig.7.5. This stratified region reduces the heat transfer at the top of the casing by reducing the wall temperature gradient.

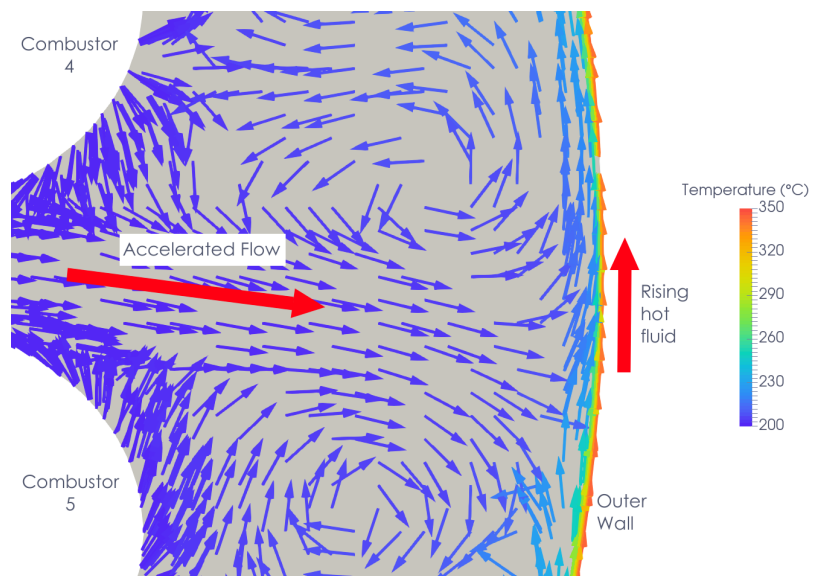
The effect of the three-dimensional flowfield on the outer wall heat transfer can be seen in Fig.7.8. The effect of the combustors causing local flow acceleration and impingement on the outer wall can be seen in the local peaks of heat transfer. The lower heat transfer at the top of the casing due to the stratified region is also evident.

The time averaged heat transfer on the cylindrical and conical sections of the casing outer wall was area averaged axially at circumferential locations from the bottom to the top of the casing. This axially averaged outer wall heat flux is plotted around the casing in Fig.7.10, with  $0^\circ$  being the bottom of the casing and  $180^\circ$  at the top of the casing. It can be seen that the general trend of the data has a lower heat flux at the top of the casing, this trend is also evident from the linear best fit through the data.

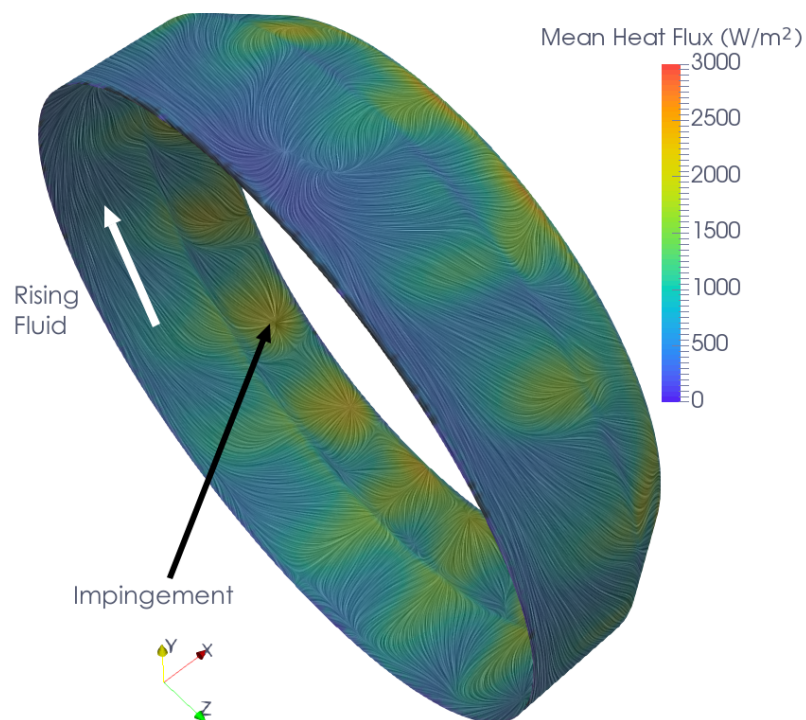




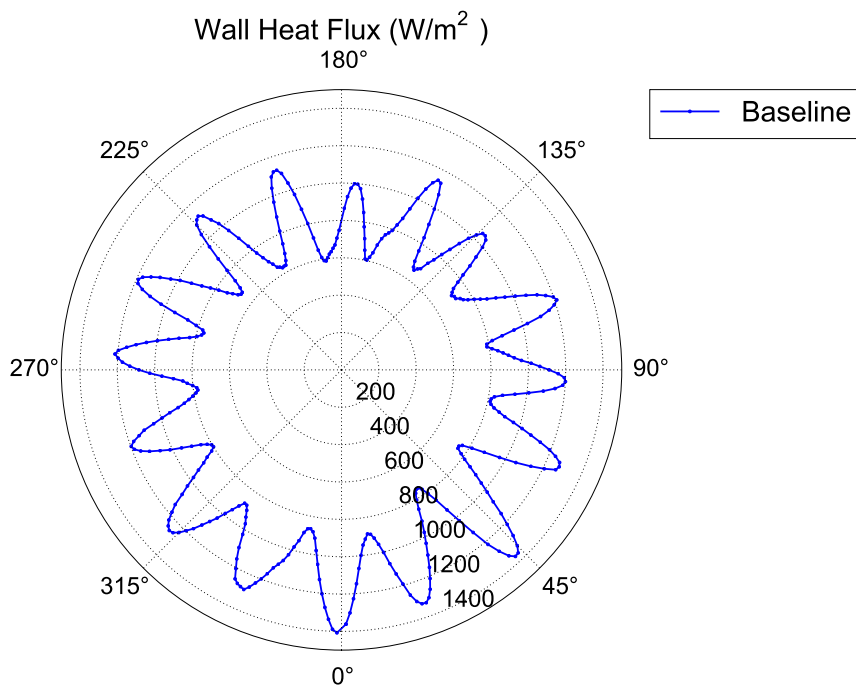
**Figure 7.6:** Baseline Case, Velocity Vectors in Section Through Combustors 16 & 1



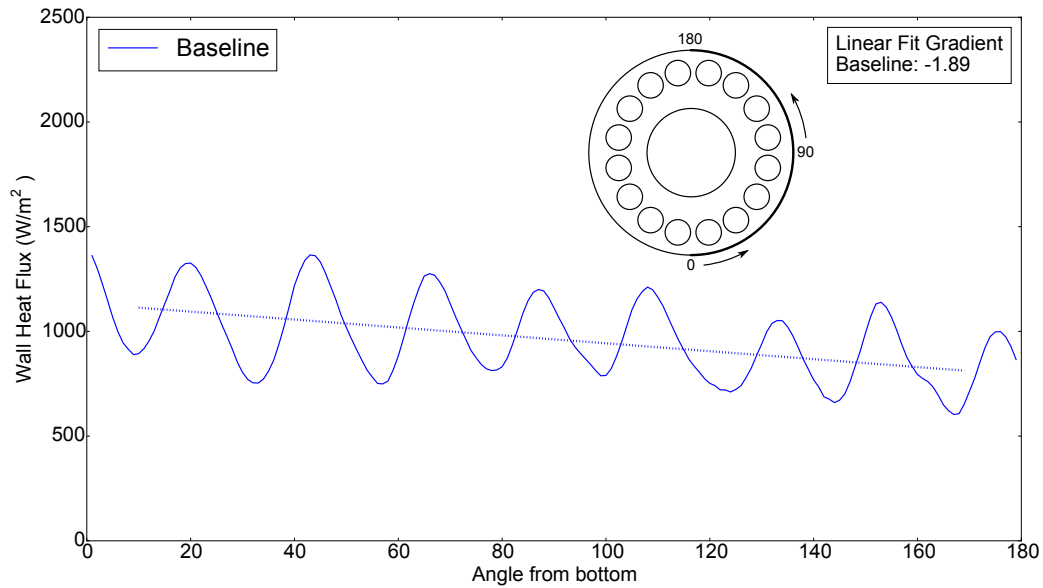
**Figure 7.7:** Baseline Case, Velocity Vectors in Axial Section C-C



**Figure 7.8:** Baseline Case, Outer Wall Heat Flux With Surface Streamlines



**Figure 7.9:** Circumferential distribution of axially averaged outer wall heat transfer



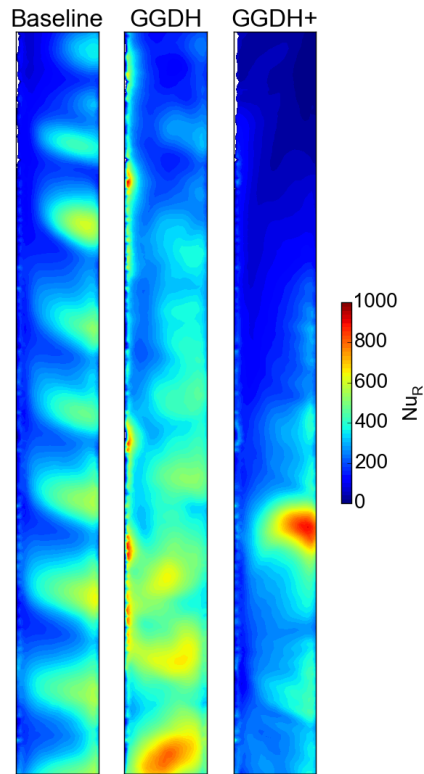
**Figure 7.10:** Baseline Case, Circumferential Distribution of Axially Averaged Outer Wall Heat Flux

This non-uniform heat transfer on the outer casing is responsible for the non-uniform cooling and casing distortion that occurs in industrial gas turbines.

## 7.6 Turbulent Heat Flux Model

The initial simulations in this investigation used the SGDh turbulent heat flux model. This was chosen as it is the simplest and most robust model and the results are suitable for comparisons of cases and finding trends. To determine the effect of using a more advanced turbulent heat flux model, simulations were run using the baseline case with the GGDH and GGDH+ turbulent heat flux models.

It was found that when using the GGDH and GGDH+ models the simulation was less stable than the baseline case. The time-step of 50 ms used for the baseline case had to be reduced to 1 ms to keep the simulations stable. The reduction in time-step meant that the flow time that averages were calculated over had to be reduced to keep an acceptable computational cost, a flow time of 50 seconds would have taken approximately 45 days on a 160 core machine. The simulation was therefore run for 2000 time-steps before averages were obtained over a further 5000 time-steps.



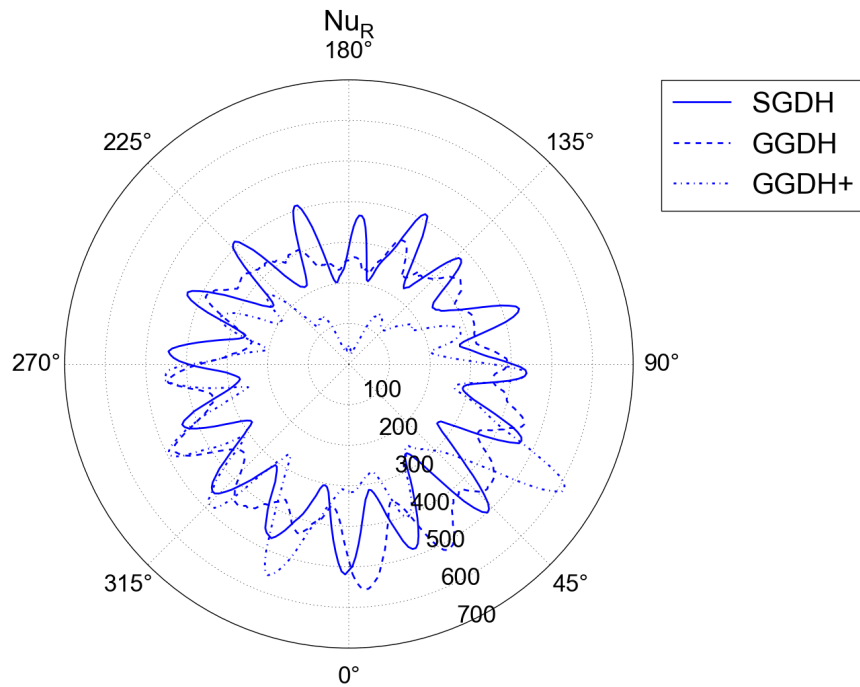
**Figure 7.11:** Outer wall Nusselt number contours using different turbulent heat flux models

The outer wall heat transfer profile from the cases run the GGDH and GGDH+ turbulent heat flux models is compared to the baseline case in Fig.7.11. Initially it can be seen that in all three cases there is still a heat transfer gradient from the bottom to the top of the casing.

In the GGDH case the heat transfer has been enhanced at the bottom of the casing and the values are lower compared to the SGDh model at the top of the casing. This can also be seen more clearly if the Nusselt number is axially averaged over the cylindrical and conical sections of the outer casing, the circumferential distribution shown in Fig.7.12.

It can be seen in Fig.7.12 that the baseline case using the SGDh model has a profile that is almost symmetrical about the vertical axis. This is due to the sufficient averaging time of 50 seconds used for this simulation. The cases using the GGDH and GGDH+ models can be seen to show more asymmetry, showing that the averaging time of 5 seconds was not sufficient for the averages to fully converge.

The distribution of heat transfer in Fig.7.12 using the GGDH+ model can be seen to have a spot of high heat transfer. This spot is caused by impingement of cold flow from

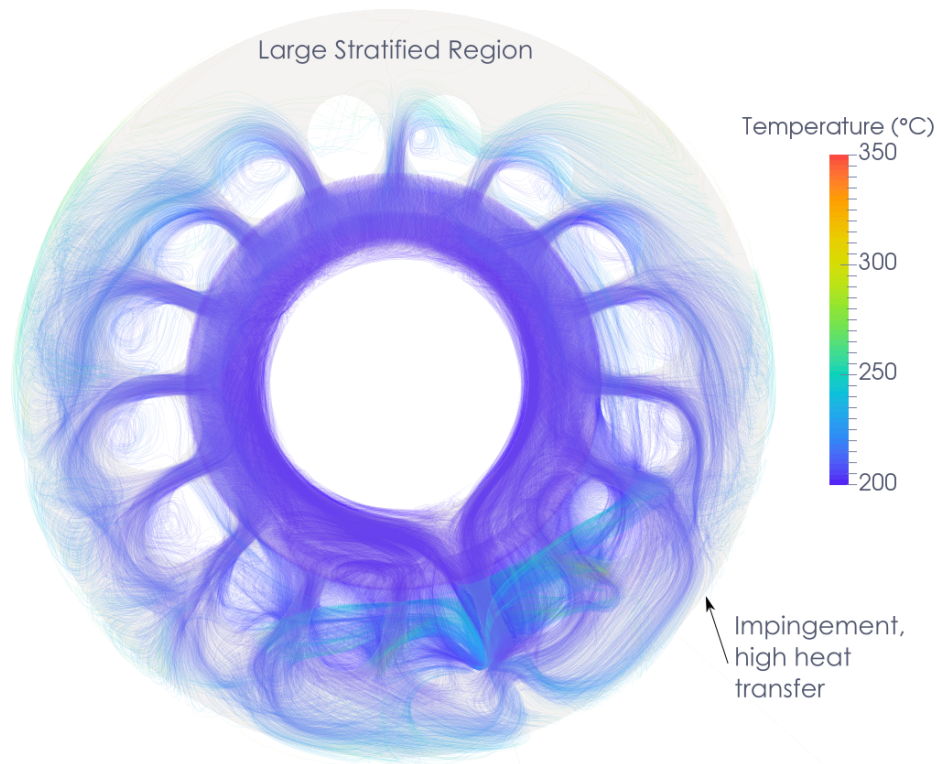


**Figure 7.12:** Circumferential distribution of axially averaged outer wall heat transfer with different turbulent heat flux models

the inlet onto the outer wall and can be seen in the time averaged stream-traces shown in Fig.7.13. This high heat transfer in the average profile is likely due to the averaging period being too short, if averaged over a longer period the stream-traces would become more symmetrical and the impingement region would likely be spread over a larger area.

What is clear from the stream-traces and heat flux profile of the GGDH+ case is that the heat transfer at the top of the casing has been significantly reduced compared to the baseline and GGDH cases. The GGDH+ model includes a term that reduces the turbulent heat flux for the case of a fluid underneath a hot surface, this has the effect of reducing the wall heat transfer which has been observed in this case. The large stratified region with low heat transfer will give a larger vertical gradient in heat transfer compared to the baseline case and would make the distortion problems more pronounced than the baseline case would predict.

The use of more advanced turbulent heat flux models has been found to have an impact on the the predicted natural convection flow and heat transfer in the gas turbine case, but the models come with an increased computational cost. For engineering design cases the base-



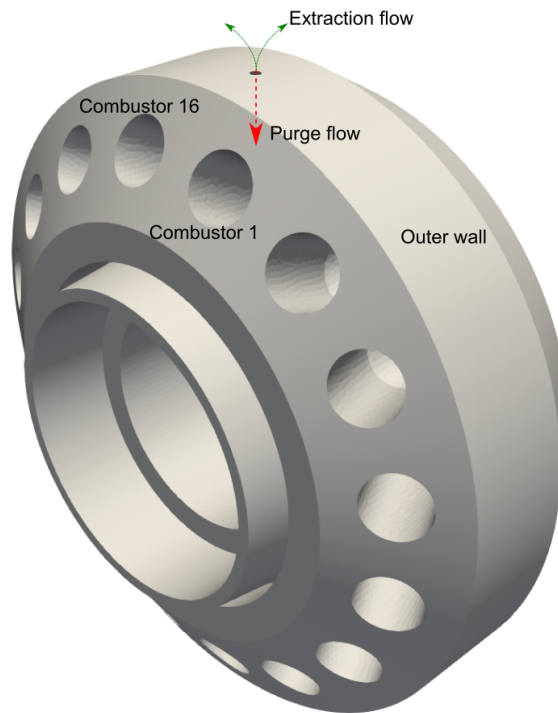
**Figure 7.13:** GGDH+ case, time averaged stream-traces looking rearwards into cavity

line SGD model may be sufficient for predicting trends and deltas between designs. The more advanced models could therefore be used later in the design process or for analysis work for cases where a large eddy simulation is not possible.

The further simulations in this investigation will use the SGD turbulent heat flux model.

## 7.7 Heat Transfer Control Investigation

It has been found that the baseline case—with no methods for controlling the casing heat transfer during shutdown—gave a non uniform casing heat flux, which would lead to casing distortion. Two methods have been investigated in order to control the casing heat transfer during shutdown and reduce the casing heat flux non-uniformity. The two methods are a purge method where cold air is injected into the casing and an extraction method where air is extracted from the casing. The two methods are shown schematically in Fig.7.14. The main aim of both of these methods is to reduce the stratified region and increase the heat



**Figure 7.14:** Schematic of Purge & Extraction Cases

transfer at the top of the casing.

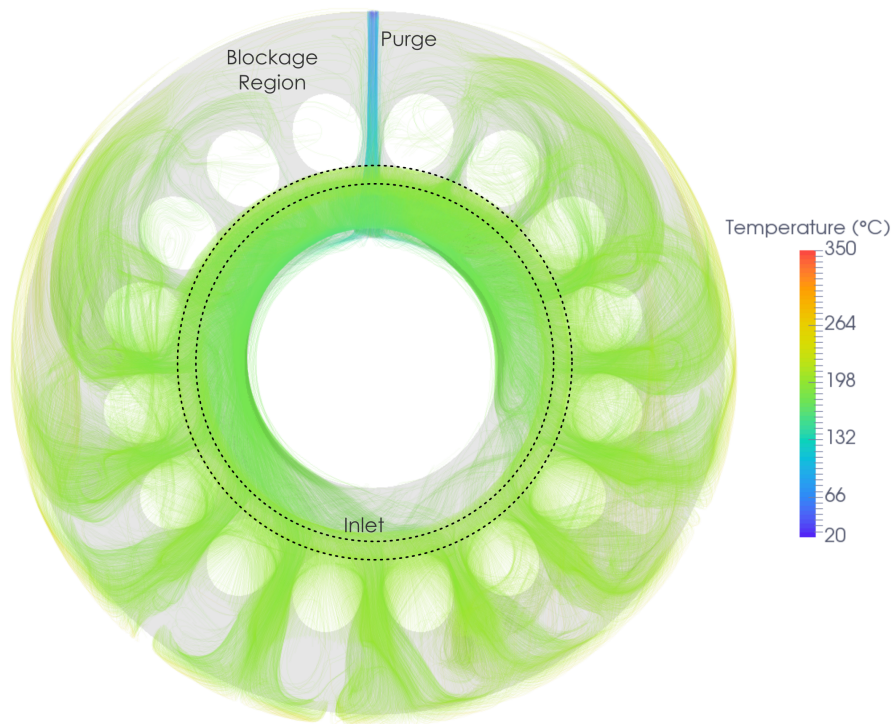
### 7.7.1 Purge Case

The first method to control the casing heat transfer was the purge flow case. In this case cold fluid is injected into the top of the casing in order to increase the heat transfer at the top of the casing. This technique of using a purge flow is used by large industrial gas turbines currently in service.

The mesh for the purge case was modified from the baseline by adding a circular inlet of 60 mm diameter to the top of casing, with mesh refinement around the inlet. The purge flow boundary condition was 0.13 kg/s flow at 20 °C.

The aim of the purge flow was to increase the heat transfer at the top of the gas turbine casing. The outer wall heat transfer is shown for this case in Fig.7.16. The purge flow has increased the heat transfer in small regions very close to the top of the casing, one region directly around the purge hole and the other further rearward. The high heat transfer directly around the purge hole is caused by the high HTC from the high velocity jet of the



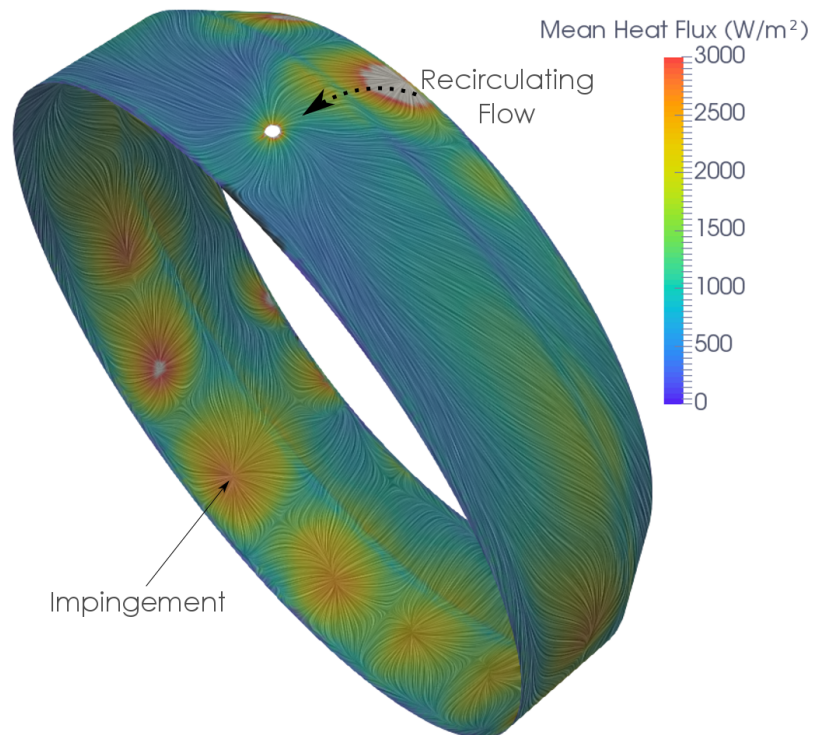


**Figure 7.15:** Purge Case, Time Averaged Stream-Traces, looking rearward into cavity

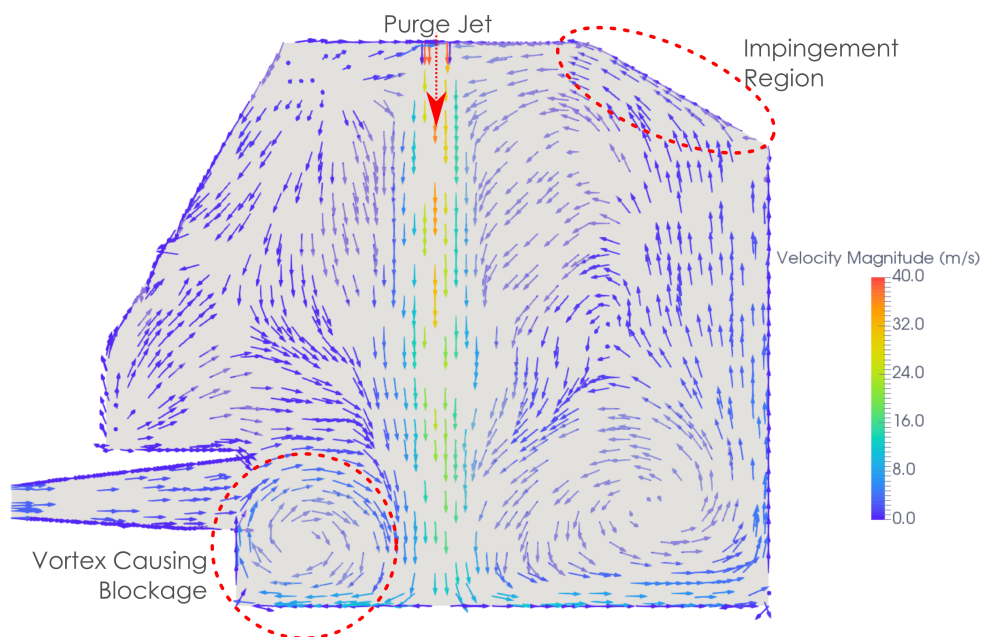
purge air. The high HTC region further rearward is due to a recirculating flow impinging on the sloping rear casing, this can be seen by looking at the velocity vectors on a cross section through the casing, Fig.7.17.

There is a region from  $110^\circ$  to close to the top of the casing of relatively low heat transfer, in which the effect of local increases in heat transfer caused by the flow acceleration around the combustors is not visible in the heat transfer distribution. This can also be seen by looking at stream-traces in the cavity, Fig.7.15. There are fewer stream-traces in the top half of the cavity, indicating a low flow region. This effect is due to the purge flow interacting with the inlet flow. In the baseline case the flow into the cavity from the compressor is circumferentially uniform, but in the purge case the distribution is skewed with less flow going into the top half of the cavity. This can be seen by looking at the circumferential velocity distribution at the compressor inlet, Fig.7.18. The low velocity into the top half of the cavity gives a large region of stagnant flow at the top of the casing, causing the low heat transfer region. The root cause of the low velocity into the top half of the cavity is a vortex which forms at the top half of the diffuser exit, this can be seen by looking at the

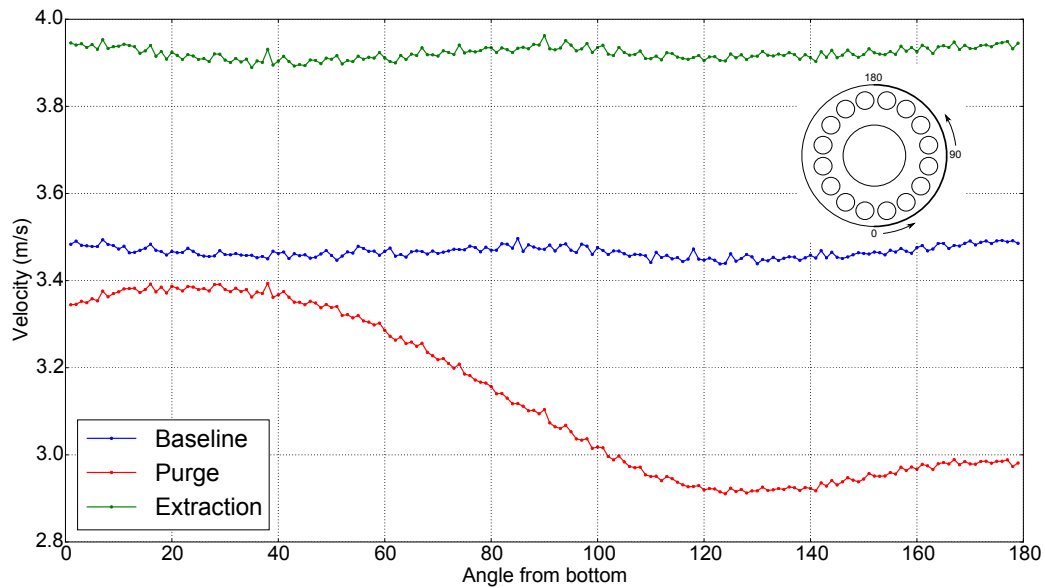




**Figure 7.16:** Purge Case, Outer Wall Heat Flux With Surface Streamlines



**Figure 7.17:** Velocity Vectors in Purge Case, Section Between Combustors 1 & 16



**Figure 7.18:** Circumferential Distribution of Velocity at Inlet

velocity vectors in Fig.7.17 and an iso-surface of Q-criterion at the diffuser exit in Fig.7.19. In summary the purge flow is causing a blockage to the flow in the top half of the cavity giving the low outer wall heat transfer.

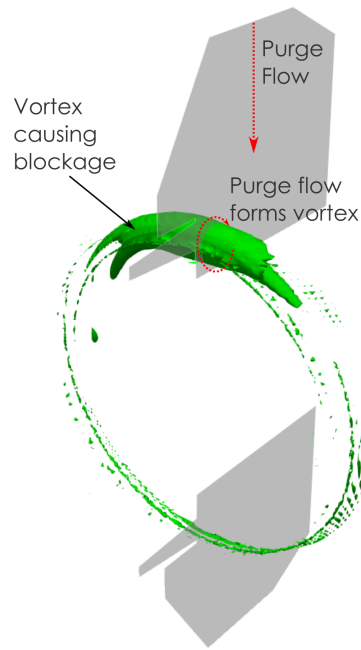
Looking at a circumferential distribution of axially averaged heat flux around the casing, Fig.7.23, the results from the purge flow analysis show that this method also produces a gradient in heat transfer from the bottom to the top of the casing.

In order to reduce the blockage effect caused by the purge, a case with a lower purge mass flow rate of 0.05 kg/s was investigated. The blockage effects however are still present in the lower mass flow rate case. This means the casing heat transfer profile cannot easily be modified by changing the purge mass flow rate.

Due to the blockage effect, the purge method of controlling the casing heat transfer is not providing a uniform distribution of casing heat flux. This non-uniform cooling could then lead to casing distortion during shutdown.

### 7.7.2 Extraction Case

Another method for reducing the non-uniformity of casing heat transfer is flow extraction from the gas turbine casing. Instead of injecting air into the casing as in the purge case, flow is extracted in order to reduce the stratified flow at the top of the casing.



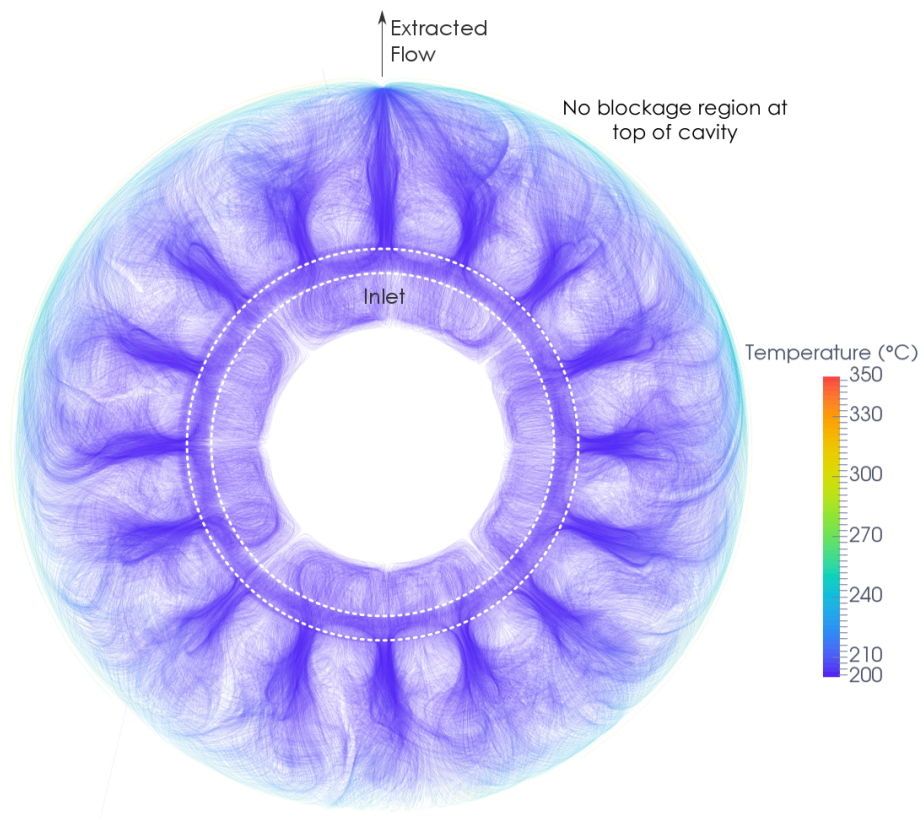
**Figure 7.19:** Purge Case, Iso-Surface of Q-Criterion at Diffuser Exit

The first extraction method tested was a system using the same 60 mm diameter hole used in the purge case, now used for extracting flow. This was to allow for the extraction system to potentially be retrofitted into existing engines. This test case used the same mesh as the purge case but the inlet at the top of the casing was changed to an outlet with a target mass flow rate boundary condition. An initial mass flow rate of 0.13 kg/s was used in order to be consistent with the purge case.

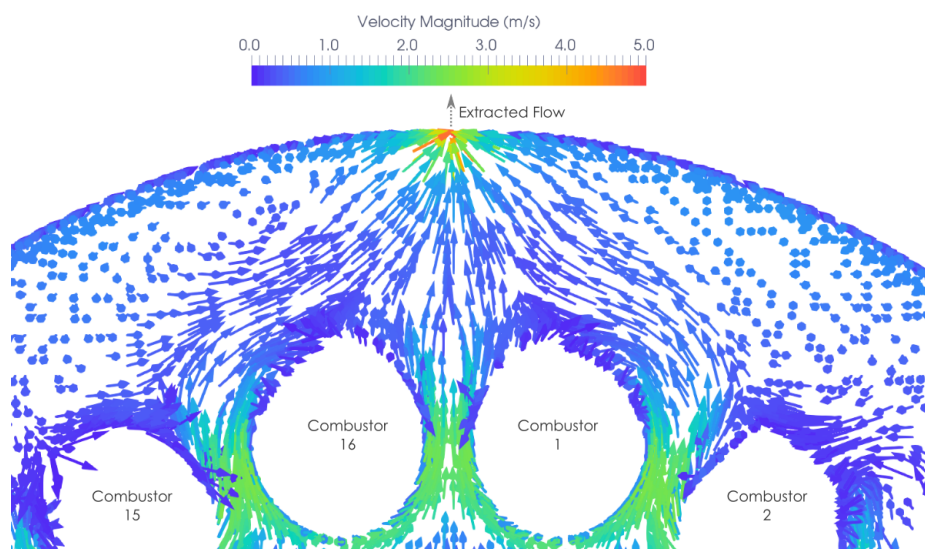
The flowfield in the extraction case can be visualised by stream-traces released from the inlet, shown in Fig.7.20. It can be seen that the flow is more uniform than in the purge case with no large blockage region in the top half of the cavity. The low velocity stratified region has also been reduced by the extraction flow. This can be seen by in the velocity vectors at the top of the casing, Fig.7.21.

The time averaged outer wall heat flux is shown in Fig.7.22. It can be seen that the profile is now more uniform, with the heat transfer being enhanced at the top of the casing.

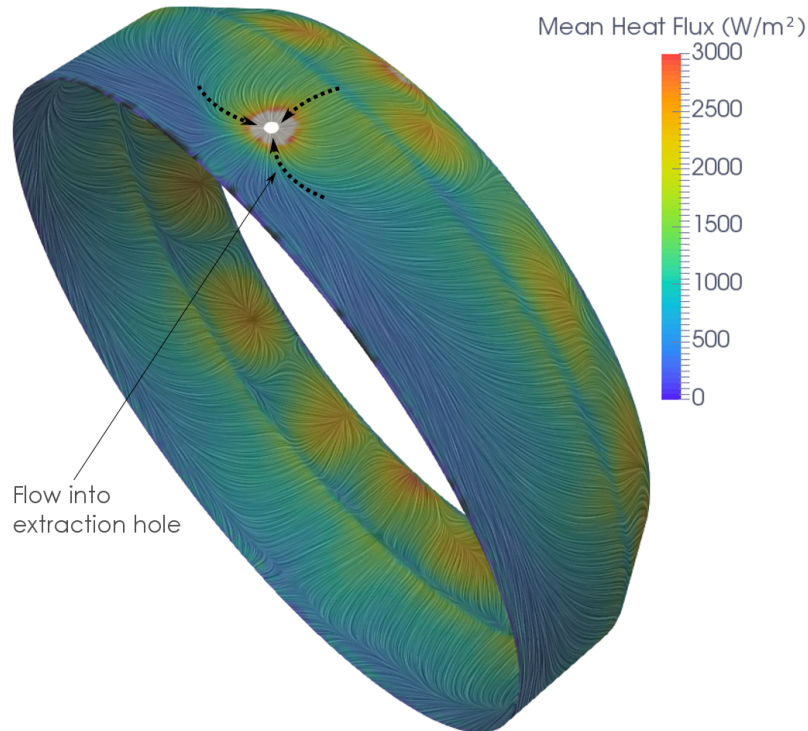
Results of the axially averaged heat flux around the casing outer wall is plotted around the circumference for the three cases in Fig.7.23. The heat flux profile around the casing is more uniform in the extraction case than both the baseline and purge methods. This is true



**Figure 7.20:** Extraction Case, Time Averaged Stream-Traces, looking rearward into cavity



**Figure 7.21:** Extraction Case, Velocity Vectors in Axial Section C-C

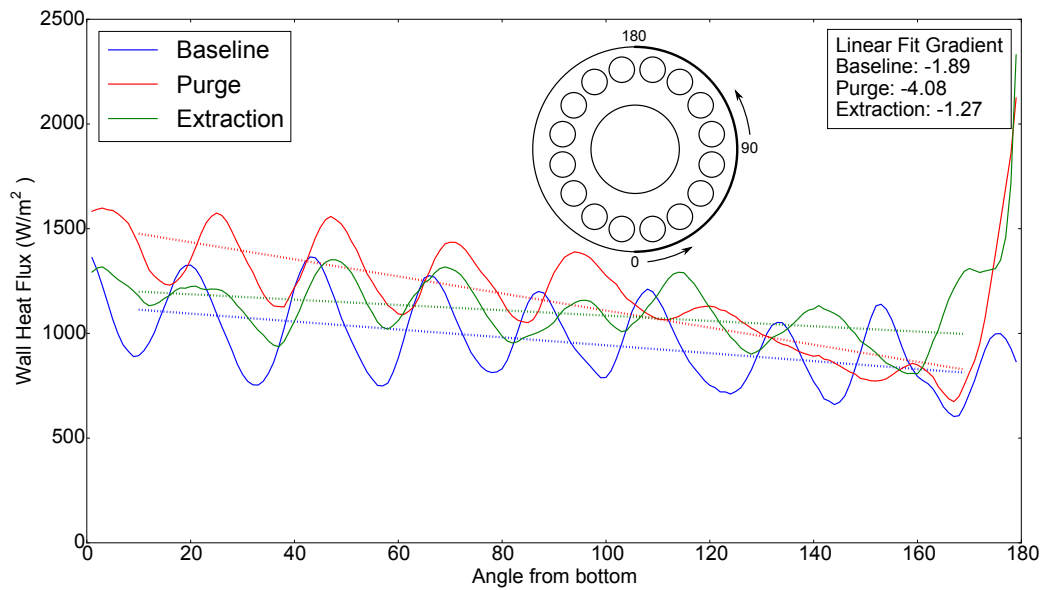


**Figure 7.22:** Extraction Case, Outer Wall Heat Flux With Surface Streamlines

for both a reduction in the local heat transfer peaks from the combustors and the general trend around the circumference. There is a peak in the heat transfer near to the top of the casing for the extraction case, this heat transfer enhancement is due to flow acceleration close to the extraction hole.

In order to obtain a quantitative comparison of the three cases the gradient of a linear fit through the data in Fig.7.23 has been calculated. The gradient shows the trend of heat transfer from the bottom to the top of the casing. Ideally the gradient of the linear fit would be zero, giving no change in the distribution of heat flux around the circumference. The linear fit for the extraction case shows a 33% reduction in the gradient compared to the baseline case, showing the more uniform heat transfer distribution with the extraction case. The linear fit has been calculated using data from  $10^\circ$  to  $170^\circ$ , this avoids the large peak in heat transfer around the hole which can skew the gradient calculation.

The extracted mass flow in this case was 0.13 kg/s to be consistent with the purge flow case, however lower mass flow rates could potentially be used. A lower mass flow rate would have a lower velocity through the extraction hole and less heat transfer enhancement



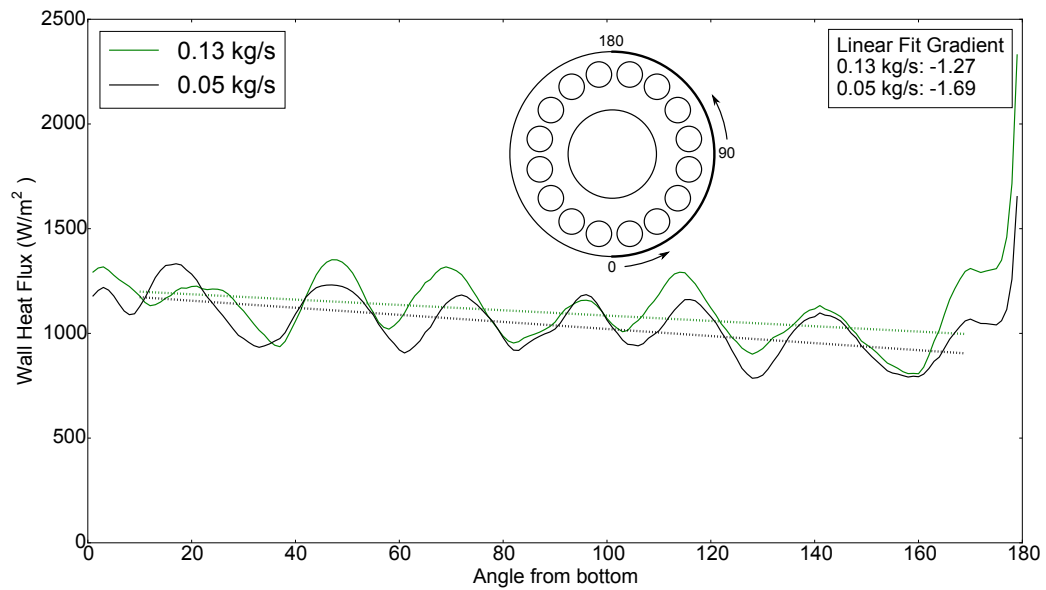
**Figure 7.23:** Circumferential Distribution of Axially Averaged Outer Wall Heat Flux

around the hole, which could give a heat flux profile that is more circumferentially uniform. A lower extraction mass flow of 0.05 kg/s was therefore tested. The circumferential distribution of heat flux with two extraction mass flows is shown in Fig.7.24, it was found that the lower mass flow case gives a heat flux profile that is similar to the 0.13 kg/s case, but the region of enhanced heat flux near to the extraction hole is now smaller. This lower mass flow rate may not be optimal however as the general trend of heat flux is still lower at the top of the casing compared to the bottom.

### 7.7.3 Two Hole Extraction

The first extraction method tested used the same 60 mm diameter hole for extraction, as was used in the purge configuration. This was to allow potential retrofitting of the extraction system to existing engines fitted with the purge system. For new engine designs a better system may be possible if the constraint of using the single 60 mm hole is removed. An alternate method would therefore be extraction from two ports. It can be seen in Fig.7.10 that the lowest heat transfer does not occur right at the top of the casing but at around 170°, measured from the bottom of the casing. These locations of lowest heat transfer line up with the point of the combustor being closest to the casing. A modification was tested that added two holes, of 60 mm diameter, at the locations of lowest heat transfer. The total





**Figure 7.24:** Extraction case, circumferential outer wall heat flux with different extraction mass flows

mass flow rate extracted through both holes was initially kept at 0.13 kg/s to allow for a comparison with the other methods.

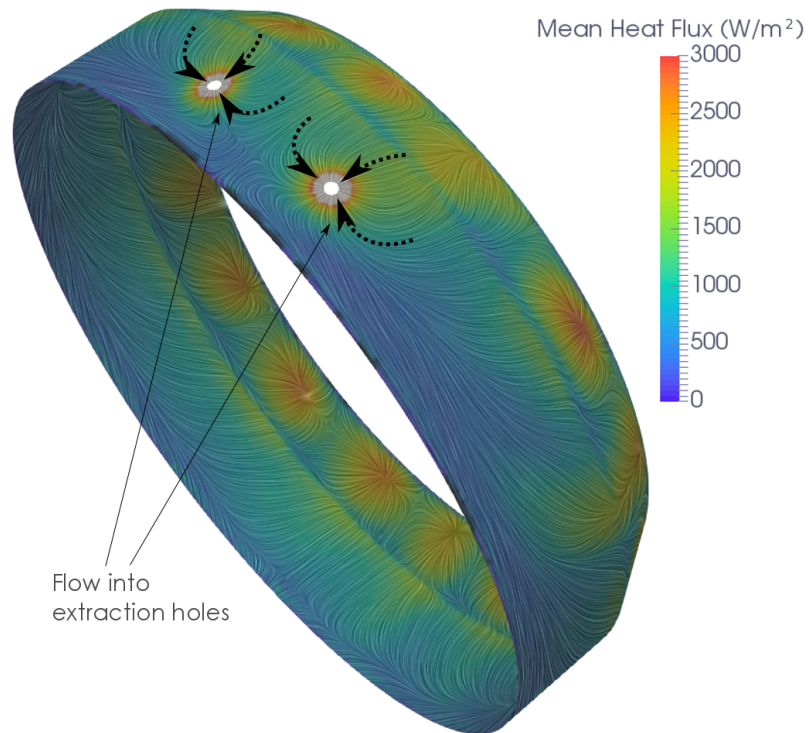
The results of the two hole extraction are shown in Fig.7.25. The distribution of heat flux looks more uniform than for the single extraction hole but the local regions of high heat transfer around the extraction holes are still present.

In order to reduce the local high heat transfer around the two extraction holes a case with a 0.05 kg/s extraction mass flow was tested. It can be seen in Fig.7.26 that the lower mass flow rate reduced the region of high heat transfer at the top of the casing but the general trend of heat flux is now lower at the top of the casing than the bottom.

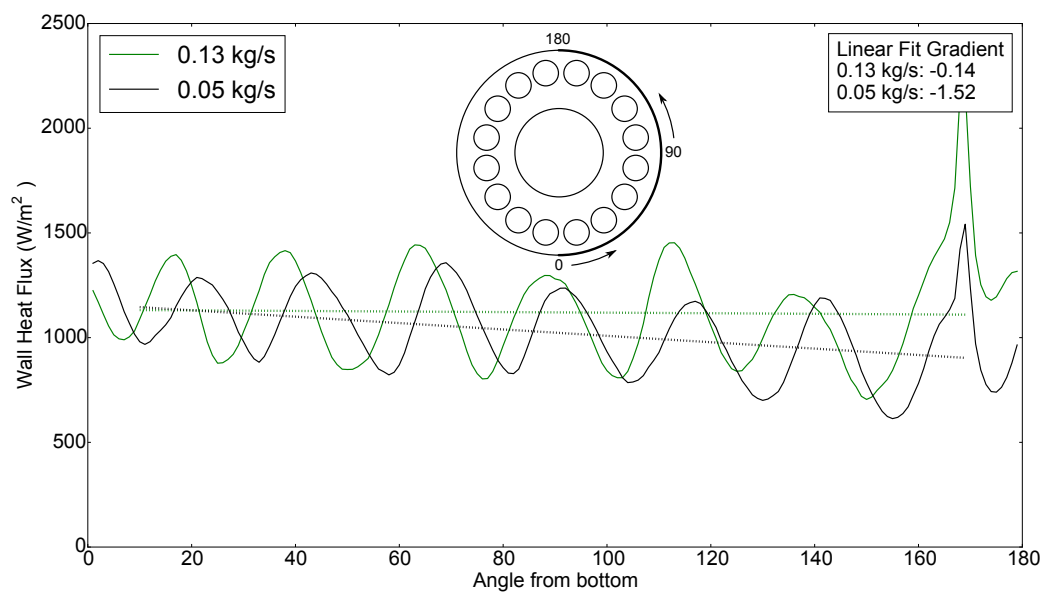
The two hole extraction case does give a more uniform heat transfer profile than the single hole however the two hole method would be difficult to retrofit and may not be worth the extra complexity of the single hole.

## 7.8 Comparison

A comparison of the contours of heat flux on the cylindrical section of the outer casing for the different methods is shown in Fig.7.27. It can be seen that the extraction methods give a

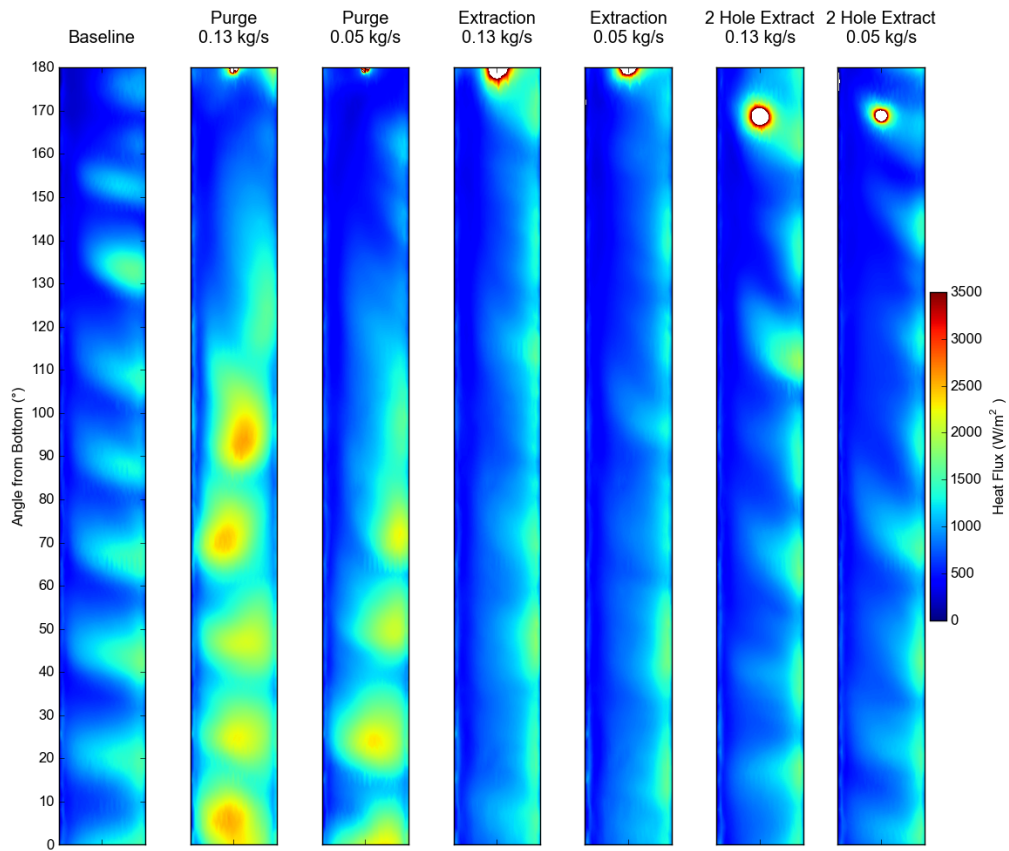


**Figure 7.25:** Two hole extraction case, outer wall heat flux with surface streamlines



**Figure 7.26:** Two hole extraction case, circumferential outer wall heat flux with different extraction mass flows





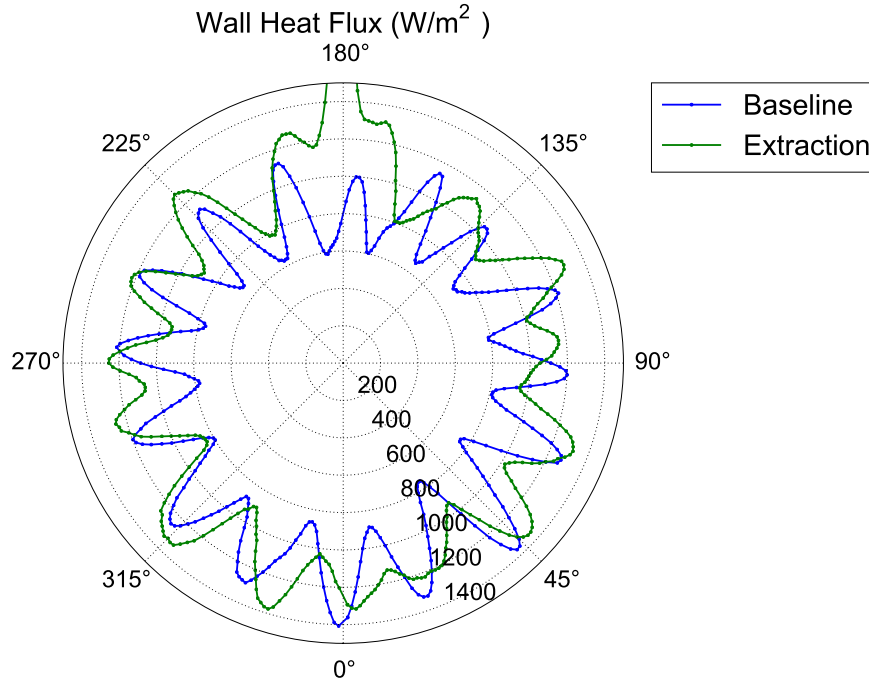
**Figure 7.27:** Comparison of heat flux on cylindrical section of outer casing

more uniform heat flux profile than the baseline and purge cases. The differences between the one and two hole extraction are relatively small so the benefits of the two hole system may not be worth the increased complexity. The extracted mass flow rate can be varied to tune the heat flux profile depending on the results found in service.

The circumferential distribution of wall heat transfer from baseline case is compared to the 0.13 kg/s extraction case, in Fig.7.28. It can be seen that the profile of outer wall heat transfer is more uniform using the extraction, with a reduction in the difference in heat transfer from the top to the bottom of the casing.

## 7.9 High Mass Flow Case

The previous methods for controlling the shutdown flow tried to modify the natural convection flowfield to improve the uniformity of the casing heat transfer. An alternative tech-



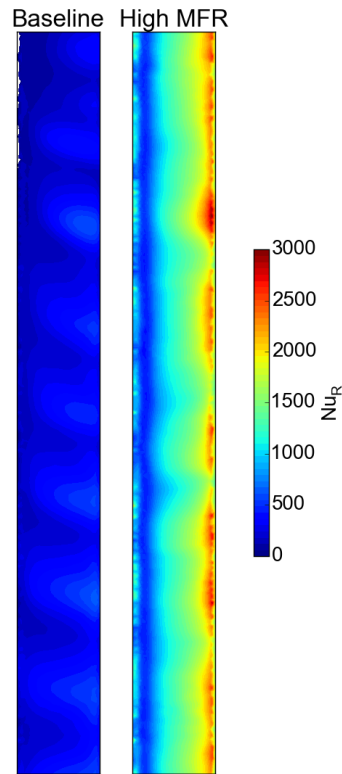
**Figure 7.28:** Circumferential distribution of axially averaged outer wall heat transfer for baseline and extraction cases

nique would be to increase the forced mass flow through the engine to the point at which the natural convection heat transfer would become small, this would occur at a Richardson number of  $Ri < 0.3$ . For the baseline case the forced mass flows was 1 kg/s which gave mass averaged outlet velocity of 0.9 m/s, a forced mass flow of 5 kg/s would give an outlet velocity  $\approx 4.5$  m/s. Using the outer casing radius of 1.85 m, the outer casing temperature of 350°C and the inlet temperature of 200°C this would give  $Ri = 0.28$ , putting the flow into the forced convection regime.

A simulation was run using the baseline case but with a target mass flow rate increased from 1 kg/s to 5 kg/s. For consistency with the baseline the case was run with a 50 ms time-step for 1000 time-steps, averages were then computed over a further 1000 time-steps.

For this case the inlet flow was assumed to stay at a constant 200°C. If the increased airflow was drawn from cool ambient conditions then this temperature could decrease, however the source of the air could be hotter air from downstream of the turbine and be recirculated back into the inlet.

Looking at the results of the outer wall heat transfer from the high mass flow case

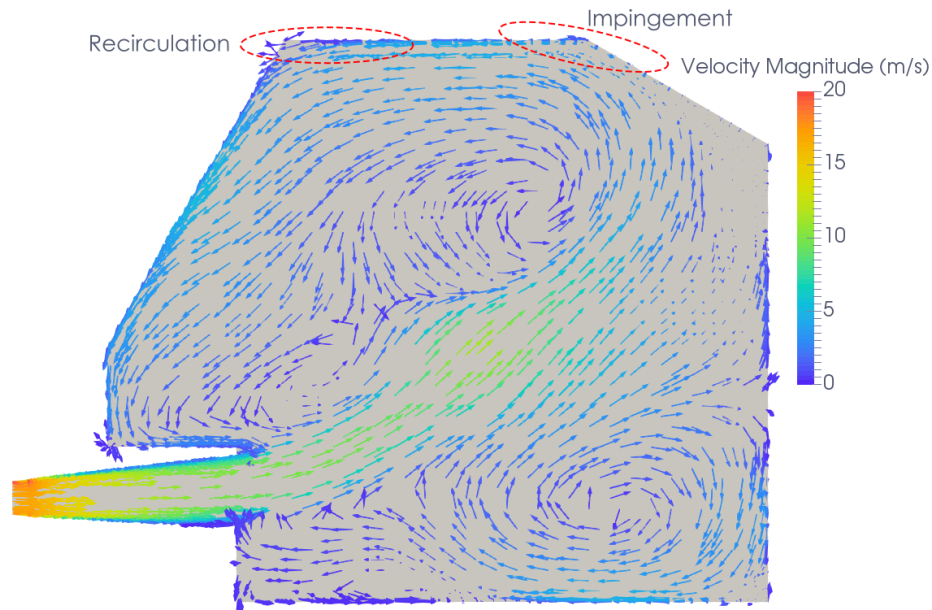


**Figure 7.29:** Outer wall Nusselt number contours for baseline and high mass flow rate cases

compared to the baseline case in Fig.7.29, it can be seen that the levels of heat transfer are much higher in the high mass flow rate case. This would have the effect of cooling the gas turbine more quickly during shutdown, which may not be desirable if a quick start-up is required. The high mass flow rate has performed as desired in creating a heat transfer distribution that is circumferentially very uniform .

It can be seen that while the heat transfer is circumferentially uniform there is a large axial gradient in the heat transfer, this may cause issues of increased stress in the outer casing. The cause of the large axial gradient can be seen by looking at velocity vectors on the rig centreline, shown in Fig.7.30. The high heat transfer region further rearward is due to flow from the inlet impinging onto the outer wall. As this flow moves radially inwards towards the outlet, a recirculation region occurs towards the front section of the outer casing. This recirculation region is the cause of the lower heat transfer region at the forward section of the outer wall.

The high mass flow rate case was able to provide a circumferentially uniform outer



**Figure 7.30:** Velocity vectors in high mass flow case, Section between combustors 1 & 16

wall heat transfer but also caused large axial gradients in heat transfer. This method of controlling the shutdown flow, by making the natural convection heat transfer small compared to the forced convection, can therefore not be recommended as a method for reducing the casing distortion.

## 7.10 Post Shutdown Case

The previous test cases were run using boundary conditions that were representative of an engine just after shutdown, the wall temperatures were therefore circumferentially uniform. As the engine cools a vertical temperature gradient occurs, with cooler temperatures occurring at the bottom of the engine in both the metal and inlet air. This temperature gradient will affect the natural convection flowfield which in turn will affect the wall heat transfer.

The effect of this temperature gradient will be investigated, one method would be to perform a conjugate simulation starting from the initial shutdown and proceeding for several hours of flow time. The issue arises that the flow time-scale is much shorter than the time-scale of the large metal components. If a simulation was run with a 50 ms time-step then hundreds of thousands of time-steps would be required to run for several hours of flow time.

An alternative method is to prescribe temperature boundary conditions at several times after shutdown. This method has a much lower computational cost as only tens of seconds of flow time have to be simulated to calculate the average wall heat transfer. The problem occurs in that the temperature boundary conditions are not well known, due to the limited shutdown temperature data. Simulations will therefore be run using boundary conditions that are representative of those at two points after the initial shutdown. The results will show the trend of how the temperature gradient effects the heat transfer.

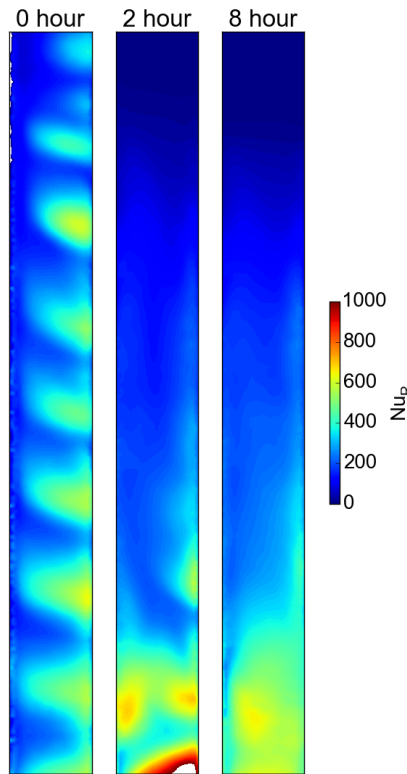
Two cases were run representing an engine at +2 hours and +8 hours after shutdown. The geometry was the same as the baseline case with no active method of controlling the flow. A vertical temperature profile was applied to the boundary conditions for the outer casing and the inlet. The +2 hour case used an inlet profile that varied linearly from 50 to 100°C, and an outer wall profile that varied from 275 to 325°C. The +8 hour case used an inlet profile that varied from 25 to 75°C, and an outer wall profile that varied from 175 to 275°C. The forced draught was assumed to keep a constant 1 kg/s for all of the cases. The numerical method for these cases was the same as the previous baseline case.

### **7.10.1 Results and Discussion**

The outer wall Nusselt number from the baseline +2 hour and +8 hour cases are given in Fig.7.31. The Nusselt number was calculated using the local outer wall temperature, with the reference temperature being the average inlet temperature. It can be seen that the heat transfer profile is very different post shutdown, implying a different flowfield.

The top of the cavity does not show the effect of the accelerated flow through the combustors impinging on the outer wall, and there is a region of high heat transfer at the bottom of the outer wall. This heat transfer profile can be explained by looking at stream-traces in the cavity, shown for the +2 hour case in Fig.7.32, the +8 hour case is qualitatively similar to this. It can be seen that the cold flow from the lower part of the inlet sinks as it flows into the main cavity, where it then impinges onto the outer wall. This cold flow impinging on the hot outer casing causes the region of high heat transfer at the bottom of the casing.

The effect of the flow accelerated through the combustors and impinging on the outer wall is not visible in the contours of the post shutdown cases. This can also be seen in

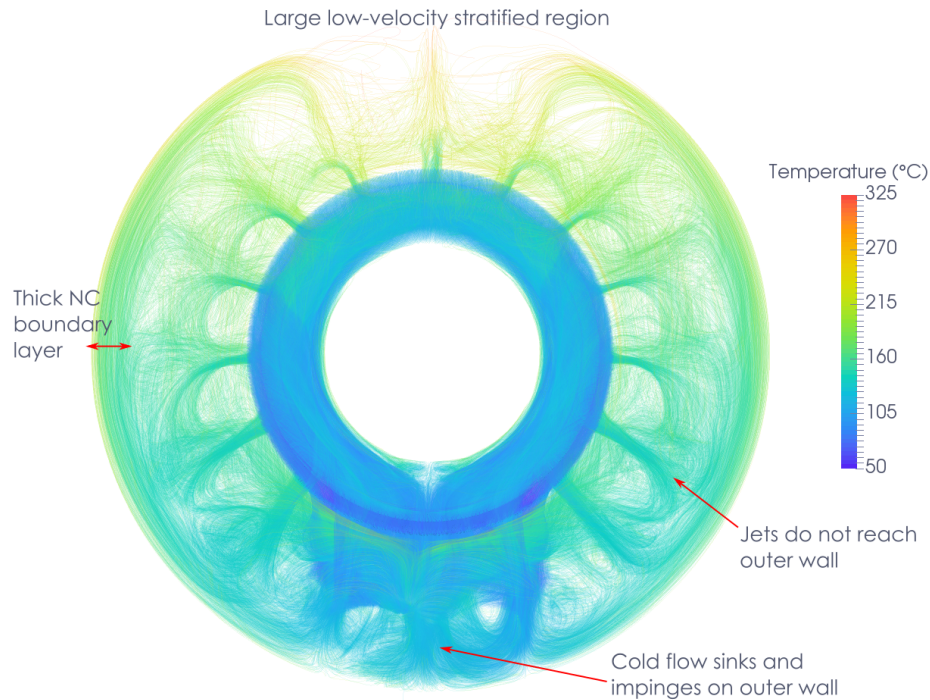


**Figure 7.31:** Outer wall Nusselt number contours at different shutdown times

Fig.7.32, in that the jet from the combustor does not reach the outer wall. Instead it is stopped by the natural convection boundary layer rising next to the outer wall, which is much thicker than in the baseline case. The natural convection effects are having a more significant effect than the jetting caused by the forced flow through the combustors in the post shutdown cases. This is confirmed by calculating the Richardson number which is 25.3 and 24.8 for the +2 hour and +8 hour cases respectively. These numbers put the flow into the natural convection regime compared to the baseline in the mixed convection regime with a Richardson number of 8.0.

The region of low velocity thermally stratified fluid can also be seen to be larger in the post shutdown case compared to the baseline case. This is causing the region of very low heat transfer towards the top of the casing.

The effect of the post shutdown temperature gradient on the gradient of outer wall heat transfer is shown in Fig.7.33. As suspected the vertical temperature gradient has exacerbated the vertical heat transfer gradient which has significantly increased over the baseline



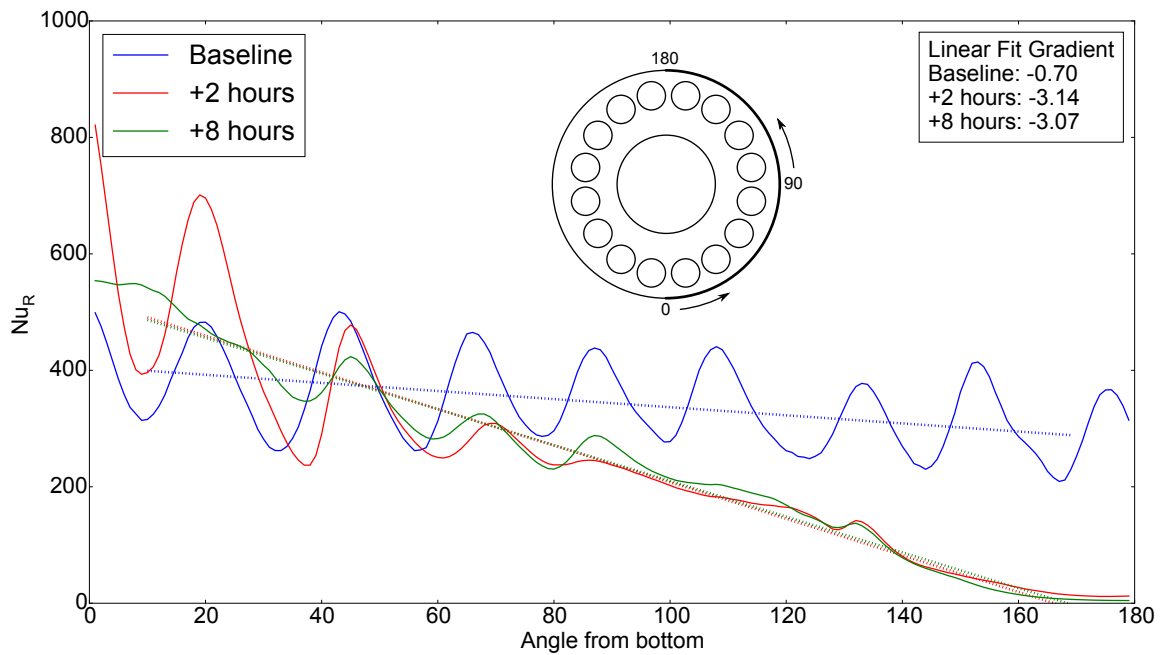
**Figure 7.32:** +2 hour case, time averaged stream-traces looking rearwards into cavity

case. This figure illustrates the need for an active method of controlling the shutdown flowfield inside of the engine. If the thermally stratified layer is allowed to develop this decreases the heat transfer in the top of the casing, causing the stratified layer to increase further. This acts as a positive feedback mechanism which will ultimately cause casing distortion. Therefore a shutdown flow control method would ideally stop the stratified layer from forming.

## 7.11 Conclusions

An investigation was conducted into methods for reducing the non-uniformity of heat transfer from industrial gas turbine casings during shutdown. Numerical simulations were performed on a geometry that is representative of a section of a large industrial gas turbine. The simulations were run in an unsteady manner and the results were taken from the time average of the simulations.

A baseline case with no active method for controlling the casing heat flux distribution was found to have lower heat transfer at the top of the casing, this region of low heat transfer



**Figure 7.33:** Circumferential Distribution of Axially Averaged Nusselt Number

would be responsible for casing distortion.

The use of more advanced turbulent heat flux models was investigated. When the GGDH+ model was used the vertical heat transfer gradient was found to increase. This was due to a larger region of stratified fluid forming at the top of the cavity. The penalty of using the more advanced models was an increased the computational cost so the SGD H model was used for the rest of the study.

A case with a purge system, which injected cool air into the top of the engine casing was then investigated. It was found that the casing heat transfer in the purge case was also non-uniform, with low heat transfer at the top of the casing caused by a blockage from the purge flow.

A new extraction method for reducing the casing heat flux non-uniformity was then investigated. The casing heat flux distribution in this case was found to be more uniform than both the baseline and purge cases. The initial extraction mass flow rate was 0.13 kg/s, in order to be consistent with the purge case, but other mass flow rates were investigated. This allows some scope for the heat flux profile in the engine to be tuned by adjusting the extracted mass flow rate.



A two hole extraction method was then tested with the aim of further improving the uniformity of heat flux. It was found that the two hole method gave results similar to the one hole extraction.

The extraction method was found to give an improved heat flux distribution over the baseline and purge cases for the initial shutdown period and could give reduced casing distortion compared to the baseline and purge methods. The extraction method therefore has potential to improve the shutdown performance for both new engine designs and current engines if retrofitted.

An alternative method for controlling the shutdown flow is to make the forced convection large compared to the natural convection, this was achieved with a large mass flow rate through the engine. The distribution of heat transfer with a large through-flow was circumferentially uniform but gave a large axial heat flux gradient which could cause thermal stresses.

The casing heat transfer distribution was then investigated at different times after the initial shutdown. It was found that the vertical gradient in heat transfer was increased in the post shutdown cases. This was due to an increased stratified region at the top of the cavity. The Richardson number of the flow also increased as the natural convection flows became dominant over the forced draught. The need for a method to control the shutdown flowfield was shown in order to stop the large stratified region from forming.

In this study the natural convection flow inside of a gas turbine casing during shutdown was simulated. Different methods of controlling the shutdown flow were investigated and potential improvements have been identified. It has been shown that the proposed extraction method could give an improved shutdown performance.

# Chapter 8

## Conclusions

### 8.1 Summary

During the shutdown of a gas turbine natural convection flows occur inside of the engine casing. These natural convection flows cause non-uniform heat transfer inside of the engine and can cause the casing to become distorted. The first step in alleviating the shutdown casing distortion is to understand the flowfield inside of the engine.

Engineers need a computationally low cost method for predicting the natural convection flow and heat transfer occurring inside of the engine, this means the use of RANS simulations. It has been found that commercial solvers give errors in wall heat transfer greater than 20% for high Rayleigh number natural convection flows. Methods to improve the prediction of heat transfer in gas turbines during shutdown have therefore been investigated.

Experimental data is required in order to validate any new numerical methods. There are limited datasets in the literature and those available tend to use simple geometries. Ideally validation data would come from a real engine during shutdown but this is very difficult to obtain. Instead a test rig was constructed by MHI that represented a section of gas turbine casing and operated at the same Rayleigh number as a gas turbine during shutdown.

Validation data from the casing rig gave mean temperature profiles and wall heat transfer measurements. In order to better understand the flowfield inside of the casing rig Large Eddy Simulations (LES) were conducted. The simulations gave insight into the vortex

transition mechanism as well as providing turbulent statistics that could not be obtained directly from the experiment. The combination of experimental and LES data could then be used to validate RANS simulations.

In order to provide further validation for new numerical methods a literature test case was used. The differentially heated tall cavity was chosen as a test case due to the large validation dataset available. Both temperature and velocity data is available and data includes both mean and rms measurements, however the turbulent heat flux terms  $(\overline{u'T'})$  are not available. In order to get a better understanding of the flowfield and to calculate turbulent statistics a Direct Numerical Simulation (DNS) was performed, this allowed the turbulent heat fluxes to be resolved. The combination of experimental and DNS data could then be used to validate the new numerical models.

In order to improve predictions of natural convection flow and heat transfer using RANS simulations, different models for the turbulent heat flux term were implemented. The baseline model, the Simple Gradient Diffusion Hypothesis (SGDH), was found to be unsuitable for high Rayleigh number natural convection flows and gave errors in heat transfer greater than 20%. More advanced models for the turbulent heat flux such as the Generalised Gradient diffusion Hypothesis (GGDH) and the Algebraic Flux Model (AFM) were then implemented into the commercial solver. The performance of the models was then evaluated using the validation datasets.

During the investigation into the flowfield in the gas turbine casing rig, it was found that streamwise vortices appeared in the rig and acted as the flow transition mechanism. The vortices were of interest as they caused a significant enhancement in heat transfer. A numerical investigation was conducted into the vortices in order to try and understand how the vortices form and which factors can affect the vortex wavelength.

An investigation was conducted looking at different methods for controlling the natural convection flow inside of a gas turbine casing during shutdown. Simulations were conducted looking at a baseline case, with no active method of controlling the flow. This was compared to cases with purge flows injected into the casing and with flows extracted from the casing. The uniformity of the heat transfer distributions around the casing produced by the different flow control methods were then compared.

## 8.2 Conclusions

A review of the literature found that there are very few studies looking at natural convection flows in gas turbines during shutdown, with no cases for large industrial gas turbines. Experimental test cases were found looking at natural convection at high Rayleigh numbers in simple geometries, such as a vertical plate or cavity flows, which could be used as CFD validation cases. There was a gap in the literature for an experimental test rig that represents an industrial gas turbine and operates at engine realistic Rayleigh numbers.

Large eddy simulations were performed on the gas turbine casing rig, it was found that the vortex transition mechanism that was observed in the experiment was also predicted by the simulations. The transition also occurred at a lower Rayleigh number than would be found on a heated vertical plate, This was due to the buoyancy forces from the flow on top of the heated surface enhancing the boundary layer growth. At the top of the heated rig, when the flow was underneath a heated surface, the wall heat transfer was found to be reduced due to the thermal stratification reducing the turbulent fluctuations.

The DNS simulation performed on the tall cavity geometry gave a dataset that complemented the experimental data. The mean values of temperature and velocity were in good agreement with the experiment but it was found that the DNS predicted higher temperature fluctuations. This was likely due to the higher frequency temperature fluctuations that were resolved in the simulation but not measured in the experiment. The turbulent heat flux terms that were resolved by the DNS were then used for validation of turbulent heat flux models.

A method was developed to implement the more advanced GGDH and AFM turbulent heat flux models into the commercial solver ANSYS Fluent. The more advanced models outperformed the baseline SGDH model when predicting wall heat transfer and turbulent heat fluxes in the vertical plate and tall cavity test cases. The wall heat transfer was then within the 20% target, making the simulations suitable for engineering design. It was found that the AFM model suffered stability issues and would not converge when used on the casing rig geometry. The RANS simulations had a computational cost ranging from a few minutes on an engineering PC to a day on a cluster, this would put the simulations

within the reach of engineers.

When comparing the wall heat transfer in the casing rig predicted by the LES to the RANS simulation using the GGDH model, it was found that the RANS simulation over-predicted the heat transfer at the top of the casing. In this region the flow is stably stratified which reduces the heat transfer, however this effect is not included in the GGDH model. A modification to the GGDH model, known as GGDH+, added a factor that accounted for if the flow was underneath or on top of a heated surface, and modified the turbulent heat flux accordingly. This modification was found to improve the heat transfer and turbulent heat flux predictions over the baseline GGDH model, with the computational cost of the GGDH+ model being the same as the standard GGDH model.

The SGDH model has been shown to generally give errors in wall heat flux greater than 20%, but the model is useful to engineers due to its fast runtime. It may be useful for studies where trends are more important than absolute values such as a large design of experiments. It has been shown that the GGDH and GGDH+ models offer improved accuracy relative to the baseline SGDH model but with around a sixfold increase in computational time. The increase in computational time over the baseline SGDH model is due to the use of a UDF to implement the models. If the models were implemented directly into the solver a significant speed-up would be possible. A final option available to engineers is to use a LES rather than a RANS simulation. The LES simulations were found to give the best agreement with experimental data, but have a increased computational cost over RANS simulations. The LES has to be run in an unsteady manner and requires a finer mesh than the RANS simulations. LES may be useful to engineers for certain analysis cases where the improved accuracy justifies the increased computational cost.

A mechanism was proposed to describe the formation of the streamwise vortices that were found in the casing rig. Simulations were performed to investigate what sets the vortex wavelength. It was found that the variation of wavelength with wall temperature agreed with experimental correlations found in the literature. It was thought that the wavelength changes could be due to changes in material properties however investigations showed that was not the case. Finally the effect of flow underneath a heated wall was investigated and it was found that in this case the vortices did not occur, showing the need for a buoyancy

force away from the wall.

The investigation into controlling the flow in a gas turbine during shutdown showed the low heat transfer stratified region in the baseline case. It was found that the purge flow case caused a blockage at the top of the cavity and therefore did not increase the heat transfer in this region. The extraction flow case showed a more uniform distribution of heat transfer with the extraction flow removing the stratified region. The extraction method could therefore improve the shutdown flowfield and reduce distortion if implemented into gas turbines.

### **8.3 Future Work**

The validation dataset produced by the gas turbine casing rig is unique but time was limited for collecting data. The experimental campaign for the second build of the rig was less than three weeks. It would be useful to obtain detailed near wall temperature measurements in the top half of the rig, where the flow is underneath of the heated wall. This would allow for the wall heat flux to be calculated and would allow for a more detailed validation throughout the whole of the casing rig.

To make the new turbulent heat flux models easier to use they could be integrated into the commercial solver. Currently with the User Defined Function (UDF) implementation the internal turbulent heat flux model has to be suppressed and the new model added as a source term, several new variables have to be saved using User Defined Memory (UDM) allocations. If the models were fully integrated then choosing a turbulent heat flux model could be as simple as choosing a turbulence model. The runtime may also be reduced by avoiding duplication of calculations by both the solver and the UDF.

The buoyancy modification similar to that in the GGDH+ model could also be applied to the SGDH model to give an SGDH+ model. This could give improved predictions when the flow is underneath a heated surface relative to the baseline SGDH model.

The calculations performed on the gas turbine used constant temperature boundary conditions. A conjugate simulation however would allow the wall temperature to vary depending on the metal temperature, which in turn would depend on the heat transfer. A conjugate

simulation is very computationally expensive as it requires simulations with small time-steps to be run for long periods of time. This cost could potentially be reduced using the method developed by Fadl et al. [19], allowing for larger fluid time-steps. The simulation could also be improved by adding a representation of the compressor upstream of the main cavity, this would remove the need to specify an air temperature gradient at the entrance to the main cavity.

## **8.4 Final Remarks**

The main contribution of this thesis is the improvement of predictions of natural convection heat transfer through the implementation of more advanced turbulent heat flux models. The models were validated with datasets created through high resolution simulations in combination with experimental data. The models have a relatively low computational cost and are now in use by engineers performing simulations of high Rayleigh number flows.

The work in this thesis can ultimately be used to improve the shutdown performance of gas turbines, leading to improved operational efficiency.

# Bibliography

- [1] F Ampofo and TG Karayiannis. Experimental benchmark data for turbulent natural convection in an air filled square cavity. *International Journal of Heat and Mass Transfer*, 46(19):3551–3572, 2003.
- [2] Amid Ansari, Jim Forsythe, and William Strang. Three dimensional simulations of buoyancy driven flows within high pressure compressor bore cavities. In *45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, page 5376, 2009.
- [3] ANSYS, Inc., Canonsburg, PA, USA. *ANSYS Fluent Theory Guide*, 15.0 edition, November 2013.
- [4] ANSYS, Inc., Canonsburg, PA, USA. *ANSYS Fluent User’s Guide*, 15.0 edition, November 2013.
- [5] ANSYS, Inc., Canonsburg, PA, USA. *Fluent 15.0 Beta Features Manual*, 15.0 edition, November 2013.
- [6] P. L. Betts and I. H. Bokhari. Experiments on turbulent natural convection in an enclosed tall cavity. *International Journal of Heat and Fluid Flow*, 21(6):675–683, 2000.
- [7] R Bolgiano Jr. Turbulent spectra in a stably stratified atmosphere. *Journal of Geophysical Research*, 64(12):2226–2229, 1959.
- [8] Dominik Born, Peter Stein, Gabriel Marinescu, Stefan Koch, and Daniel Schumacher. Thermal modeling of an intermediate pressure steam turbine by means of conjugate heat transfersimulation and validation. *Journal of Engineering for Gas Turbines and Power*, 139(3):031903, 2017.



- [9] R Boudjemadi, V Maupu, D Laurence, and P Le Quéré. Budgets of turbulent stresses and fluxes in a vertical slot natural convection flow at rayleigh  $Ra = 105$  and  $5.4 \cdot 10^5$ . *International journal of heat and fluid flow*, 18(1):70–79, 1997.
- [10] Dean R Chapman. Computational aerodynamics development and outlook. *AIAA journal*, 17(12):1293–1313, 1979.
- [11] TS Chen and Keh-Lih Tzuoo. Vortex instability of free convection flow over horizontal and inclined surfaces. *Journal of Heat Transfer*, 104(4):637–643, 1982.
- [12] K. C. Cheng and Y. W. Kim. Flow visualization studies on vortex instability of natural convection flow over horizontal and slightly inclined constant-temperature plates. *Journal of Heat Transfer*, 110:608–615, 1988.
- [13] Seok-Ki Choi and Seong-O Kim. Turbulence modeling of natural convection in enclosures: A review. *Journal of Mechanical Science and Technology*, 26:286–297, 2012.
- [14] Adil A Dafa’Alla and Philip L Betts. Experimental study of turbulent natural convection in a tall air cavity. *Experimental Heat Transfer An International Journal*, 9(2):165–194, 1996.
- [15] Bart J. Daly and Francis H. Harlow. Transport equations in turbulence. *The Physics of Fluids*, 13:2634–2649, 1970.
- [16] DANTEC Dynamics. *Probes for Hot-wire Anemometry*, 2017.
- [17] Paul A Durbin. Some recent developments in turbulence closure modeling. *Annual Review of Fluid Mechanics*, 50:77–103, 2018.
- [18] Alireza Ebadi, Faraz Mehdi, and Christopher M. White. An exact integral method to evaluate wall heat flux in spatially developing two-dimensional wall-bounded flows. *International Journal of Heat and Mass Transfer*, 84:856 – 861, 2015.

- [19] M Fadl, L He, P Stein, and G Marinescu. Assessment of unsteadiness modeling for transient natural convection. *Journal of Engineering for Gas Turbines and Power*, 140(1):012605, 2018.
- [20] Daniel D. Fahy, Peter T. Ireland, Leo V. Lewis, and Emile Raya. A novel experimental technique for investigating natural convective heat transfer in a gas turbine annulus. In *Proceedings of ASME Turbo Expo 2018*, pages 1–10. Oslo, Norway, 2018.
- [21] Fuh-min Fang. A Design Method for Contractions With Square End Sections. *Journal of Fluids Engineering*, 119:454–458, 1997.
- [22] HE Fiedler. Coherent structures in turbulent flows. *Progress in Aerospace Sciences*, 25(3):231–269, 1988.
- [23] Niranjana S Ghaisas, Dinesh A Shetty, and Steven H Frankel. Large eddy simulation of thermal driven cavity: Evaluation of sub-grid scale models and flow physics. *International Journal of Heat and Mass Transfer*, 56(1-2):606–624, 2013.
- [24] Yanan Guo and Danmei Xie. Thermal stress and deformation analysis of HP casing during shutdown of CPR1000 nuclear steam turbine. In *Proceedings of ASME Turbo Expo 2017*, pages 1–8. Charlotte, NC, USA, 2017.
- [25] Kemal Hanjalić and Brian Launder. *Modelling turbulence in engineering and the environment: second-moment routes to closure*. Cambridge university press, 2011.
- [26] MR Head and P Bandyopadhyay. New aspects of turbulent boundary-layer structure. *Journal of fluid mechanics*, 107:297–338, 1981.
- [27] R. A. W. M. Henkes and C. J. Hoogendoorn. Numerical determination of wall functions for the turbulent natural convection boundary layer. *International Journal of Heat and Mass Transfer*, 31:1087–1097, 1988.

- [28] GA Holtzman, RW Hill, and KS Ball. Laminar natural convection in isosceles triangular enclosures heated from below and symmetrically cooled from above. *Journal of Heat Transfer*, 122(3):485–491, 2000.
- [29] J. C. R. Hunt, A. A. Wray, and P. Moin. Eddies, streams, and convergence zones in turbulent flows. In *Studying Turbulence Using Numerical Simulation Databases*, 2, 1988.
- [30] N. Z. Ince and B. E. Launder. On the computation of buoyancy-driven turbulent flows in rectangular enclosures. *International Journal of Heat and Fluid Flow*, 10:110–117, 1989.
- [31] N. Z. Ince and B. E. Launder. Three-dimensional and heat-loss effects on turbulent flow in a nominally two-dimensional cavity. *International Journal of Heat and Fluid Flow*, 16:171–177, 1995.
- [32] PS Jang, DJ Benney, and RL Gran. On the origin of streamwise vortices in a turbulent boundary layer. *Journal of Fluid Mechanics*, 169:109–123, 1986.
- [33] Peter Jeschke and Hans Beer. Longitudinal vortices in a laminar natural convection boundary layer flow on an inclined flat plate and their influence on heat transfer. *Journal of Fluid Mechanics*, 432:313–339, 2001.
- [34] Peter Jeschke, Ralf Biertümpfel, and Hans Beer. Liquid-crystal thermography for heat-transfer measurements in the presence of longitudinal vortices in a natural convection flow. *Measurement Science and Technology*, 11(5):447–453, 2000.
- [35] William K. George Jr. and Steven P. Capp. A theory for natural convection turbulent boundary layers next to heated vertical surfaces. *International Journal of Heat and Mass Transfer*, 22:813–826, 1979.
- [36] BA Kader. Temperature and concentration profiles in fully turbulent boundary layers. *International journal of heat and mass transfer*, 24(9):1541–1544, 1981.

- [37] Nobuhide Kasagi and Naoki Shikazono. Contribution of direct numerical simulation to understanding and modelling turbulent transport. *Proc. R. Soc. Lond. A*, 451(1941):257–292, 1995.
- [38] S. Kenjeres and K. Hanjalic. Prediction of turbulent thermal convection in concentric and eccentric horizontal annuli. *International Journal of Heat and Fluid Flow*, 16:429–439, 1995.
- [39] John Kim, Parviz Moin, and Robert Moser. Turbulence statistics in fully developed channel flow at low reynolds number. *Journal of fluid mechanics*, 177:133–166, 1987.
- [40] Deniz Kizildag, FX Trias, I Rodríguez, and A Oliva. Large eddy and direct numerical simulations of a turbulent water-filled differentially heated cavity of aspect ratio 5. *International Journal of Heat and Mass Transfer*, 77:1084–1094, 2014.
- [41] Stephen J Kline, William C Reynolds, FA Schraub, and PW Runstadler. The structure of turbulent boundary layers. *Journal of Fluid Mechanics*, 30(4):741–773, 1967.
- [42] Takuma Kogawa, Junnosuke Okajima, Atsuki Komiya, Steven Armfield, and Shigenao Maruyama. Large eddy simulation of turbulent natural convection between symmetrically heated vertical parallel plates for water. *International Journal of Heat and Mass Transfer*, 101:870–877, 2016.
- [43] T. H. Kuehn and R. J. Goldstein. An experimental and theoretical study of natural convection in the annulus between horizontal concentric cylinders. *Journal of Fluid Mechanics*, 74(4):695 – 719, 1976.
- [44] Rajesh Kumar and Anupam Dewan. A study of les–sgs closure models applied to a square buoyant cavity. *International Journal of Heat and Mass Transfer*, 98:164–175, 2016.
- [45] GE Lau, GH Yeoh, V Timchenko, and JA Reizes. Large-eddy simulation of natural convection in an asymmetrically-heated vertical parallel-plate channel: Assessment of subgrid-scale models. *Computers & Fluids*, 59:101–116, 2012.

- [46] GE Lau, GH Yeoh, V Timchenko, and JA Reizes. Large-eddy simulation of turbulent buoyancy-driven flow in a rectangular cavity. *International Journal of Heat and Fluid Flow*, 39:28–41, 2013.
- [47] Brian Edward Launder and Dudley Brian Spalding. The numerical computation of turbulent flows. In *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pages 96–116. Elsevier, 1983.
- [48] D. K. Lilly. A proposed modification of the germano subgrid-scale closure method. *Physics of Fluids A: Fluid Dynamics*, 4(3):633–635, 1992.
- [49] J. R. Lloyd and E. M. Sparrow. On the instability of natural convection flow on inclined plates. *Journal of Fluid Mechanics*, 42(3):465–470, 1970.
- [50] LUNA user’s guide ODiSI-B. [lunainc.com/wp-content/uploads/2014/05/ODiSI-B-Users-Guide.pdf](http://lunainc.com/wp-content/uploads/2014/05/ODiSI-B-Users-Guide.pdf). Accessed: 2018-03-19.
- [51] William M. Kays. Turbulent prandtl number-where are we? *ASME Transactions Journal of Heat Transfer*, 116:284–295, 04 1994.
- [52] W.R. Dean M.A. Xvi. note on the motion of fluid in a curved pipe. *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science*, 4(20):208–223, 1927.
- [53] Gabriel Marinescu, Wolfgang F. Mohr, Andreas Ehresam, Paolo Ruffino, and Michael Sell. Experimental investigation into thermal behavior of steam turbine components - temperature measurements with optical probes and natural cooling analysis. *Journal of Engineering for Gas Turbines and Power*, 136:021602, 11 2013.
- [54] Andrew E McLeod and Eugene H Bishop. Turbulent natural convection of gases in horizontal cylindrical annuli at cryogenic temperatures. *International Journal of Heat and Mass Transfer*, 32(10):1967–1978, 1989.
- [55] Florian R Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 32(8):1598–1605, 1994.

- [56] MHPS industrial gas turbine. [https://www.mhps.com/products/gasturbines/img/index\\_im02.jpg](https://www.mhps.com/products/gasturbines/img/index_im02.jpg). Accessed: 2018-01-25.
- [57] Parviz Moin. Advances in large eddy simulation methodology for complex flows. *International journal of heat and fluid flow*, 23(5):710–720, 2002.
- [58] Dejan Morar. *Subgrid-scale Heat Flux Modeling for Large Eddy Simulation of Turbulent Mixed Convection*. PhD thesis, KIT-Bibliothek, 2014.
- [59] Nobuo Mori, Hideo Matsuda, Yoshihisa Ishii, and Mikio Midori. Design and operation experience of combined cycle plant using gas turbine inlet temperature of 1350°C - no.5 unit of himeji no.1 power station of the kansai electric power co., inc-. Technical Report Vol.33 No3, Mitsubishi Heavy Industries, Ltd., Oct 1996.
- [60] PXI analog input module. <http://www.ni.com/en-gb/shop/select/pxi-analog-input-module>. Accessed: 2018-05-09.
- [61] Ayodeji Ojofeitimi and Yasuo Hattori. Wall-resolved large eddy simulation of turbulent mixed-convection heat transfer along a heated vertical flat plate. *International Journal of Heat and Mass Transfer*, 109:428–439, 2017.
- [62] D. K. Oldaker and W. G. Tiederman. Spatial structure of the viscous sublayer in drag-reducing channel flows. *The Physics of Fluids*, 20(10):S133–S144, 1977.
- [63] James W. Lottes Paul F. Fischer and Stefan G. Kerkemeier. nek5000 web page. <http://nek5000.mcs.anl.gov>.
- [64] Shia-Hui Peng and Lars Davidson. Large eddy simulation for turbulent buoyant flow in a confined cavity. *International Journal of Heat and Fluid Flow*, 22(3):323–331, 2001.
- [65] Shia-Hui Peng and Lars Davidson. On a subgrid-scale heat flux model for large eddy simulation of turbulent thermal flow. *International Journal of Heat and Mass Transfer*, 45(7):1393–1405, 2002.

- [66] JR Phillips. Direct simulations of turbulent unstratified natural convection in a vertical slot for  $Pr = 0.71$ . *International journal of heat and mass transfer*, 39(12):2485–2494, 1996.
- [67] DB Pitz and JW Chew. Numerical simulation of natural convection in a differentially heated tall enclosure using a spectral element method. In *23rd ABCM International Congress of Mechanical Engineering - COBEM2015*, December 2015.
- [68] S.B. Pope. *Turbulent Flows*. Cambridge University Press, 2000.
- [69] V Reddy, K Selvam, and R De Prosperis. Gas turbine shutdown thermal analysis and results compared with experimental data. In *Proceedings of ASME Turbo Expo 2016*, pages 1–9. Seoul, South Korea, 2016.
- [70] Osborne Reynolds. On the dynamical theory of incompressible viscous fluids and the determination of the criterion. *Philosophical Transactions of the Royal Society of London. A*, 186:123–164, 1895.
- [71] Konstantinos Ritos, Ioannis W. Kokkinakis, and Dimitris Drikakis. Performance of high-order implicit large eddy simulations. *Computers & Fluids*, 2018.
- [72] PE Roach. The generation of nearly isotropic turbulence downstream of streamwise tube bundles. *International journal of heat and fluid flow*, 7(2):117–125, 1986.
- [73] M Salinas-Vazquez, W Vicente, E Martinez, and E Barrios. Large eddy simulation of a confined square cavity with natural convection based on compressible flow equations. *International Journal of Heat and Fluid Flow*, 32(5):876–888, 2011.
- [74] DSP fundamentals: Sampling rates, bandwidth, spectral lines, and more... - Siemens PLM community. <https://community.plm.automation.siemens.com/t5/Testing-Knowledge-Base/Digital-Signal-Processing-Sampling-Rates-Bandwidth-Spectral/ta-p/402991>. Accessed: 2018-02-19.

- [75] W S Saric. Görtler vortices. *Annual Review of Fluid Mechanics*, 26(1):379–409, 1994.
- [76] Kathiravan Selvam, Roberto De Prosperis, and Nageswara Rao Vanga. Transient thermal analysis of gas turbine shut down physics; normal and forced cooling. In *ASME 2014 International Mechanical Engineering Congress and Exposition*, pages 1–9. Montreal, Quebec, Canada, 2014.
- [77] Jong Keun Shin, Jeong Soo An, and Young Don Choi. Elliptic relaxation second moment closure for turbulent heat flux. In *Proceed. 4th Int. Symp. Turbulence and Shear Flow Phenomenon*, pages 271–277, Williamsburg, U. S. A, 2005.
- [78] D. L. Siebers, R. F. Moffatt, and R. G. Schwind. Experimental, variable properties natural convection from a large, vertical, flat surface. *Journal of Heat Transfer*, 107:124–132, 1985.
- [79] Jeffrey Slotnick, Abdollah Khodadoust, Juan Alonso, David Darmofal, William Gropp, Elizabeth Lurie, and Dimitri Mavriplis. CFD vision 2030 study: a path to revolutionary computational aerosciences. Technical Report NASA/CR2014-218178, NASA, 2014.
- [80] Pavel E Smirnov, Semir Kapetanovic, Mark E Braaten, Yuri Egorov, and Florian R Menter. Application of the sas turbulence model to buoyancy driven cavity flows. In *ASME Turbo Expo 2009: Power for Land, Sea, and Air*, pages 1207–1216. American Society of Mechanical Engineers, 2009.
- [81] EO Smith, AJ Neely, and AP Butcher. Experimental observations of thermal bow due to natural convection on a gas turbine compressor rotor shaft analogue. In *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*. American Society of Mechanical Engineers, 2016.
- [82] Evan Oscar Smith and Andrew J Neely. The effect of aircraft integration design on gas turbine shaft thermal bow and the newkirk effect’. In *Proceedings of ASME Turbo Expo 2014*, pages 1–12. Düsseldorf, Germany, June 2014.



- [83] P R Spalart and S R Allmaras. A one-equation turbulence model for aerodynamic flows. In *30th aerospace sciences meeting and exhibit*, page 439, 1992.
- [84] Philippe R Spalart. Philosophies and fallacies in turbulence modeling. *Progress in Aerospace Sciences*, 74:1–15, 2015.
- [85] E. M. Sparrow, R. Eichhorn, and J. L. Gregg. Combined forced and free convection in a boundary layer flow. *The Physics of Fluids*, 2(3):319–328, 1959.
- [86] E. M. Sparrow and R. B. Husar. Longitudinal vortices in natural convection flow on inclined plates. *Journal Fluid Mechanics*, 37:251–255, 1969.
- [87] Renqia Sun and Xing Zhang. An experimental study of the natural convection within the horizontal cylindrical annular enclosures. In *Proceedings of the Eighth International Heat Transfer Conference*, volume 4, pages 1569–1574, 1986.
- [88] Zixiang Sun, Klas Lindblad, John W Chew, and Colin Young. Les and rans investigations into buoyancy-affected convection in a rotating cavity with a central axial throughflow. *Journal of engineering for gas turbines and power*, 129(2):318–325, 2007.
- [89] Atsushi Tateishi, Toshinori Watanabe, and Takehiro Himeno. Unsteady flow simulation of buoyancy-driven flows in high-pressure compressor disk cavities. In *Proceedings of ASME Turbo Expo 2018*, pages 1–10. Oslo, Norway, 2018.
- [90] S. Thangam and N. Hur. Laminar secondary flows in curved rectangular ducts. *Journal of Fluid Mechanics*, 217:421–440, 1990.
- [91] YS Tian and TG Karayiannis. Low turbulence natural convection in an air filled square cavity: part i: the thermal and fluid flow fields. *International Journal of Heat and Mass Transfer*, 43(6):849–866, 2000.
- [92] YS Tian and TG Karayiannis. Low turbulence natural convection in an air filled square cavity: Part ii: the turbulence quantities. *International Journal of Heat and Mass Transfer*, 43(6):867–884, 2000.

- [93] Andrei Tisarev, Sergey Falaleev, Christian Koch, Matthias Nagorski, and Stephan Staudacher. Natural cooling affecting the restart of micro gas turbine. In *Proceedings of ASME Turbo Expo 2016*, pages 1–10. Seoul, South Korea, 2016.
- [94] FX Trias, Andrei Gorobets, A Oliva, and Carlos David Pérez-Segarra. Dns and regularization modeling of a turbulent differentially heated cavity of aspect ratio 5. *International journal of heat and mass transfer*, 57(1):171–182, 2013.
- [95] FX Trias, Andrei Gorobets, M Soria, and A Oliva. Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with rayleigh numbers up to 1011–part i: Numerical methods and time-averaged flow. *International Journal of Heat and Mass Transfer*, 53(4):665–673, 2010.
- [96] FX Trias, Andrei Gorobets, M Soria, and A Oliva. Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with rayleigh numbers up to 1011–part ii: Heat transfer and flow dynamics. *International Journal of Heat and Mass Transfer*, 53(4):674–683, 2010.
- [97] T. Tsuji and Y. Nagano. Characteristics of a turbulent natural convection boundary layer along a vertical flat plate. *International Journal of Heat and Mass Transfer*, 31:1723–1734, 1988.
- [98] T. Tsuji and Y. Nagano. Turbulence measurements in a natural convection boundary layer along a vertical flat plate. *International Journal of Heat and Mass Transfer*, 31:2101–2111, 1988.
- [99] T. Tsuji and Y. Nagano. Velocity and temperature measurements in a natural convection boundary layer along a vertical flat plate. *Experimental Thermal and Fluid Science*, 2:208–215, 1989.
- [100] Anatoli Tumin. The spatial stability of natural convection flow on inclined plates. *Journal of fluids engineering*, 125(3):428–437, 2003.
- [101] James C Tyacke and Paul G Tucker. Future use of large eddy simulation in aero-engines. *Journal of Turbomachinery*, 137(8):081005, 2015.

- [102] E Van de Sande and BJG Hamer. Steady and transient natural convection in enclosures between horizontal circular cylinders (constant heat flux). *International Journal of Heat and Mass Transfer*, 22(3):361–370, 1979.
- [103] Henk Kaarle Versteeg and Weeratunge Malalasekera. *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education, 2007.
- [104] TAM Versteegh and FTM Nieuwstadt. Turbulent budgets of natural convection in an infinite, differentially heated, vertical channel. *International Journal of heat and fluid flow*, 19(2):135–149, 1998.
- [105] David C Wilcox. Formulation of the k-w turbulence model revisited. *AIAA journal*, 46(11):2823–2838, 2008.
- [106] X Yuan, A Moser, and P Suter. Wall functions for numerical simulation of turbulent natural convection along vertical plates. *International journal of heat and mass transfer*, 36(18):4477–4485, 1993.
- [107] Xing Yuan, Fatemeh Tavakkoli, and Kambiz Vafai. Analysis of natural convection in horizontal concentric annuli of varying inner shape. *Numerical Heat Transfer, Part A: Applications*, 68(11):1155–1174, 2015.

# Appendix A

## Turbulence Modelling Study

### A.1 Turbulence Model

The role of the turbulence model is to close the Reynolds stress term  $(\overline{u'_i u'_j})$  that appears in the Navier Stokes momentum equation after the Reynolds averaging process has been performed.

The choice of turbulence model for a RANS simulation can have a significant impact on the solution. A study has therefore been conducted to determine which model performs best for natural convection flows.

### A.2 Eddy-viscosity model

A method for determining the Reynolds stress term is to use the Boussinesq approximation. The Reynolds stress is assumed to be proportional to the mean rates of deformation, Equation A.1 [103]. A model for the eddy-viscosity  $\mu_t$  is then required, leading to a family of eddy-viscosity turbulence models.

$$-\rho \overline{u'_i u'_j} = \tau_{ij} = \mu_t \left( \frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (\text{A.1})$$

#### A.2.1 Spalart-Allmaras Model

A one-equation eddy-viscosity turbulence model was developed by Spalart and Allmaras [83], solving a single equation for the eddy-viscosity. The model also needs a turbulent

length scale to be specified in order to calculate the turbulent dissipation, the length scale is often defined as the distance from a wall.

The Spalart-Allmaras (SA) model has been found to perform well for boundary layer flows but struggles with complex geometries where the turbulent length scale is hard to define.

### **A.2.2 $k - \epsilon$ Model**

The  $k - \epsilon$  model devolved by Launder and Spalding [47] is a two-equation model. Equations are solved for the turbulent kinetic energy  $k$  and the dissipation of turbulent kinetic energy  $\epsilon$ . The problem of having to define a turbulent length scale in the Spalart-Allmaras model is resolved by calculating a turbulent length scale as  $l = k^{\frac{3}{2}}/\epsilon$ . The turbulent viscosity is then calculated as  $\mu_t = \rho c_\mu \frac{k^2}{\epsilon}$ .

The  $k - \epsilon$  model is the most widely used and validated model [103]. It has found to perform well in confined flow cases common in industry but performs less well in unconfined flows.

### **A.2.3 $k - \omega$ Model**

The  $k - \omega$  model of Wilcox [105] is another two-equation eddy viscosity model. In this model equations are solved for the turbulent kinetic energy  $k$  and the specific dissipation rate of turbulent kinetic energy  $\omega$ . The turbulent viscosity is then calculated as  $\mu_t = \rho \frac{k}{\omega}$ . The  $k - \omega$  model has some advantages over the  $k - \epsilon$  model in that  $\omega$  tends to infinity close to the wall and does not require wall damping functions. Problems can occur in the free stream as  $\omega$  tends to zero making the turbulent viscosity indeterminate.

### **A.2.4 $k - \omega$ SST Model**

The  $k - \omega$  SST turbulence model developed by Menter [55] was designed to combine the benefits of the  $k - \epsilon$  model and  $k - \omega$  models. The  $k - \omega$  model is used near to the wall and is blended into the  $k - \epsilon$  model in the free stream. The  $k - \omega$  SST model has since found use a general purpose turbulence model [103].

### A.3 Reynolds Stress Model

An alternative method to using the Boussinesq approximation for obtaining the Reynolds stress is to solve a transport equation for each stress component, known as a Reynolds stress model [25]. In a 3D simulation six equations are solved for the  $\overline{u'u'}$ ,  $\overline{v'v'}$ ,  $\overline{w'w'}$ ,  $\overline{u'v'}$ ,  $\overline{v'w'}$  and  $\overline{u'w'}$  stresses with a seventh equation for the dissipation of turbulent kinetic energy  $\epsilon$ .

The Reynolds stress model has been found to outperform eddy-viscosity models for more complex flows but has also been found to suffer from convergence issues [103].

### A.4 Buoyancy Effects in Turbulence Models

Buoyancy forces can have an effect on the turbulent kinetic energy of a flow. For example the buoyancy forces in a stably stratified flow can act to damp out turbulent kinetic energy. This buoyancy effect can be modelled by including source terms in the equations for the kinetic energy, Equation.A.2 and specific dissipation rate of kinetic energy, Equation.A.3 [5].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k U_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \beta^* k \omega \quad (\text{A.2})$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega U_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\alpha}{\nu_t} G_k + G_{\omega b} - \rho \beta \omega^2 + 2(1-F_1) \rho \sigma_{\omega,2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (\text{A.3})$$

The key terms in these equations are the buoyancy production terms  $G_b$  and  $G_{\omega b}$ .

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \quad [3] \quad (\text{A.4})$$

$$G_{\omega b} = \frac{\omega}{k} C_{1\epsilon} (C_{3\epsilon} G_b) - \frac{\omega}{k} G_b \quad [5] \quad (\text{A.5})$$

Where the terms  $C_{1\epsilon} = 1.44$  and  $C_{3\epsilon} = \tanh \left| \frac{v}{u} \right|$  where  $v$  is the component of flow parallel the the gravity vector and  $u$  is the component perpendicular to the gravity vector [3].

The option is available in ANSYS Fluent v15.0 as a beta feature to enable these buoyancy source terms for the  $k - \omega$  *SST* model.

## A.5 Turbulence Model Study

An investigation into the best turbulence model to use for natural convection simulations was conducted. The test case chosen was the differentially heated tall cavity running four test cases using different turbulence models. The models tested were the  $k - \omega$  *SST*,  $k - \epsilon$ , Reynolds stress model (RSM) and the Spalart-Allmaras (SA) model. In all cases the SGDH turbulent heat flux model was used, with the details of the computational mesh and method the same as discussed in Chapter.5.

The temperature profiles predicted by the RANS simulations are compared to the experimental and DNS datasets in Fig.A.1. At the cavity mid-height it can be seen that the results are all very similar, differences between the simulations are only apparent towards the top and bottom of the cavity. In particular the RSM and the  $k - \epsilon$  models are not predicting the correct temperature profile.

The results of the predicted wall heat flux compared to the experimental data at nine locations in the tall cavity are given in Table.A.1. Also given are the average difference and the standard deviation over all nine locations. It can be seen the the  $k - \epsilon$  model has the lowest average difference but the spread of the results is quite large ranging from 38.6% to -13.1%. The  $k - \omega$  *SST* model has the second lowest average but the spread of results is much tighter, from -13.7% to -36.2%. The RSM is tending to over-predict and give large errors in the wall heat flux and finally the SA model is tending to under-predict the wall heat transfer.

The  $k - \omega$  *SST* model has been chosen as the best model for the natural convection simulations. This is due to the accurate predictions of the general temperature profile and the reasonable predictions of wall heat transfer. The heat transfer results were also consistent over the cavity height, which would make it easier to determine the effects with simulations run with different turbulent heat flux models. Whist the  $k - \epsilon$  model had a lower average difference in heat transfer from the experiment, the large spread of results would

| $y/H$ (%) | $k - \omega$ SST | $k - \epsilon$ | RSM    | SA     |
|-----------|------------------|----------------|--------|--------|
| 95        | -13.7%           | -13.1%         | -6.0%  | -9.8%  |
| 90        | -20.7%           | -0.5%          | 14.6%  | -19.2% |
| 70        | -16.4%           | 24.1%          | 49.8%  | -21.6% |
| 60        | -26.3%           | 14.6%          | 39.8%  | -33.8% |
| 50        | -24.7%           | 20.4%          | 50.2%  | -35.3% |
| 40        | -36.2%           | 4.6%           | 30.3%  | -47.4% |
| 30        | -34.6%           | 11.5%          | 42.4%  | -48.0% |
| 10        | -27.6%           | 38.6%          | 106.1% | -45.8% |
| 5         | -24.0%           | 17.9%          | 87.5%  | -40.3% |
| Average   | -24.9%           | 13.1%          | 46.1%  | -33.5% |
| $\sigma$  | 7.5%             | 15.0%          | 34.2%  | 13.8%  |

**Table A.1:** Difference in wall heat flux from experimental data with different turbulence models

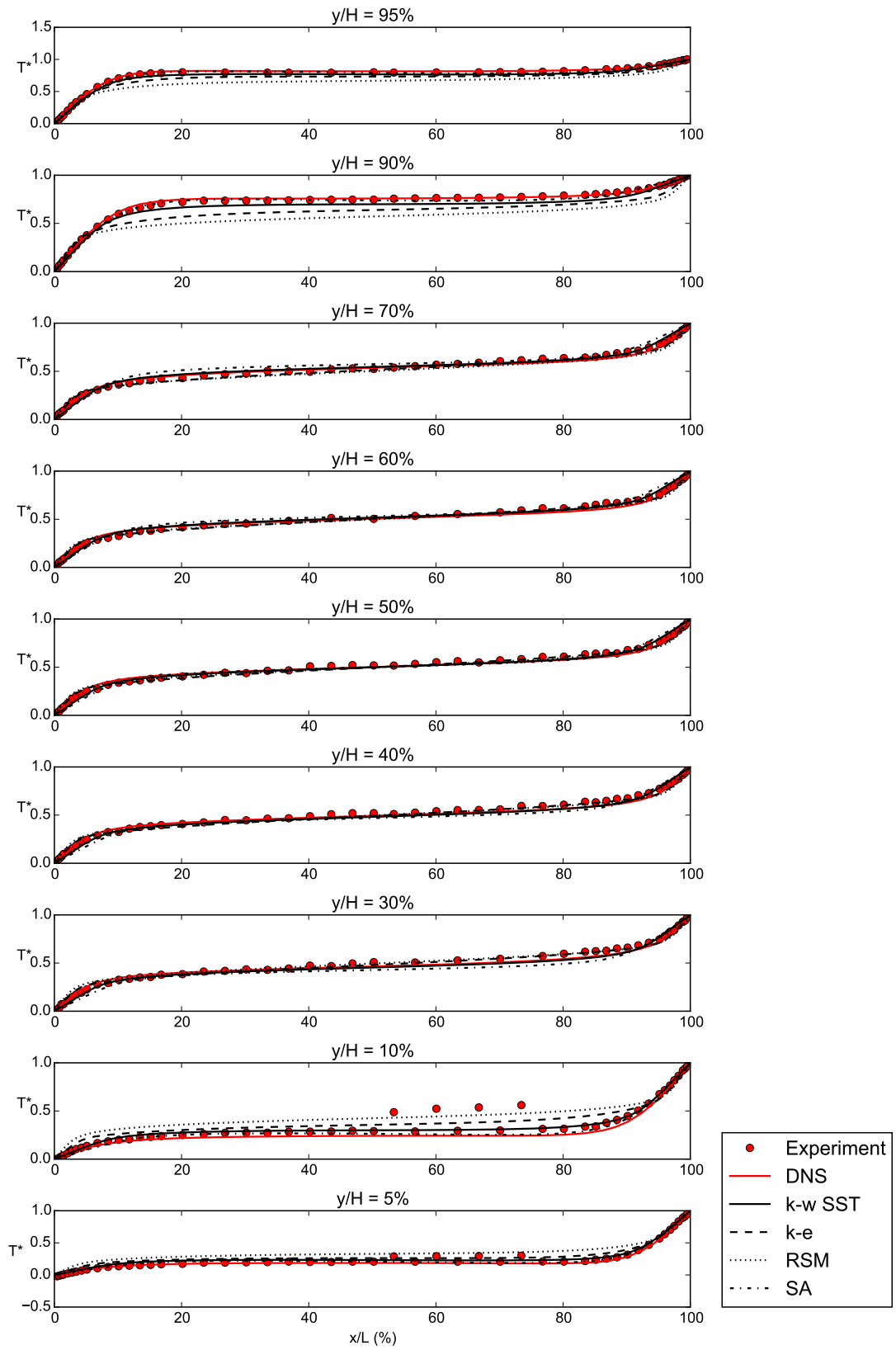
make it harder to determine if a particular turbulent heat flux model was giving improved heat transfer predictions. The  $k - \epsilon$  model also did not give the correct mean temperature profile at some cavity heights.

## A.6 Buoyancy Effect Study

An investigation into the effect of including buoyancy source terms in the turbulence model was conducted. The baseline simulation used the  $k - \omega$  SST model without any buoyancy terms, this was compared to a simulation using the terms in the  $k$  and  $\omega$  equations. The test case was again the tall cavity and the SGDH turbulent heat flux model was used in both cases.

The predictions of turbulent kinetic energy from the baseline RANS case and the case with buoyancy terms are compared to results from the DNS simulation at nine cavity heights, shown in Fig.A.2. The turbulent kinetic energy has been non dimensionalised by the square of the buoyant velocity,  $(\sqrt{g\beta\Delta T})^2 = 0.221^2$ . The key result is that the two RANS simulations are giving very similar predictions for turbulent kinetic energy. It can be seen that the only difference between the two RANS cases occurs close to the top and bottom wall. This is due to the buoyancy source term being proportional to the temperature gradient in the direction of the gravity vector. As there is only a small vertical tempera-





**Figure A.1:** Non-dimensional temperature profiles in tall cavity with different turbulence models

| y/H (%) | Baseline | Buoyancy Term |
|---------|----------|---------------|
| 95      | -13.7%   | -13.6%        |
| 90      | -20.7%   | -20.8%        |
| 70      | -16.4%   | -16.7%        |
| 60      | -26.3%   | -26.6%        |
| 50      | -24.7%   | -24.8%        |
| 40      | -36.2%   | -36.0%        |
| 30      | -34.6%   | -34.5%        |
| 10      | -27.6%   | -29.1%        |
| 5       | -24.0%   | -26.0%        |

**Table A.2:** Difference in wall heat flux from experimental data with buoyancy effects

ture gradient at the cavity mid-height, the buoyancy source term disappears. However the vertical temperature gradient at the top and bottom of the cavity has only resulted in small changes to the turbulent kinetic energy profile.

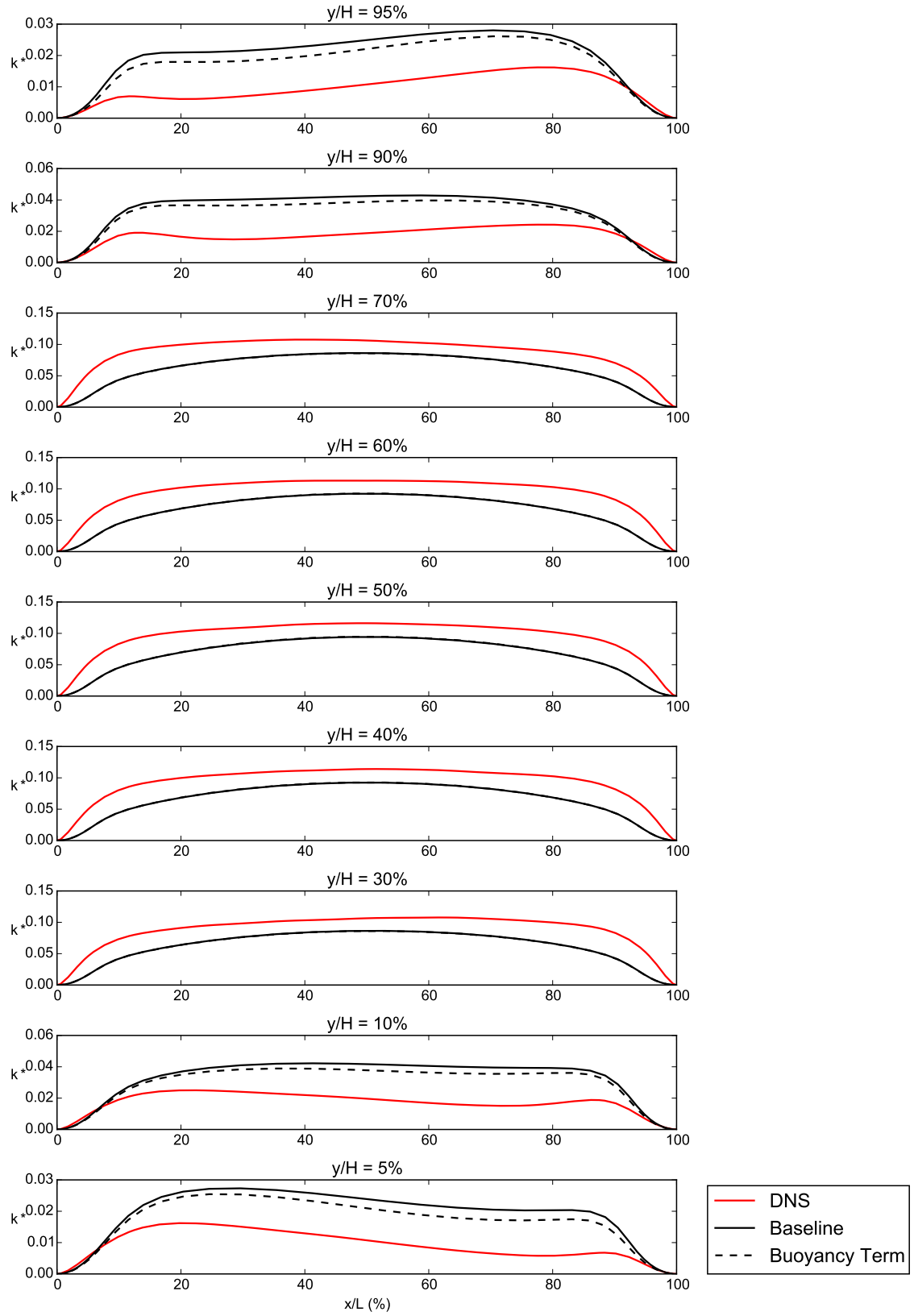
The effect of the buoyancy source term on the wall heat transfer can be seen in Table A.2. There is very little difference between the baseline case and the case with the buoyancy terms included in the turbulence model.

The results of this investigation into buoyancy effects in the turbulence model has found that they do not lead to significant differences in prediction of turbulent kinetic energy or heat transfer. The buoyancy source terms are also a beta feature in ANSYS Fluent v15.0 so may cause numerical stability issues. Furthermore it is a fallacy as described by Spalart [84], that obtaining the correct value of  $k$  and  $\omega$  is the key for success in a two-equation model. The buoyancy terms will therefore not be used in the main investigation.

## A.7 Conclusions

An investigation was conducted into which turbulence model would be best for natural convection simulations. It was found that the  $k - \omega$  *SST* performed the best in terms of predicting the correct temperature profiles in the tall cavity. The  $k - \omega$  *SST* model also gave consistent predictions of the wall heat transfer, which will make improvements due to different turbulent heat flux models easier to identify. The  $k - \omega$  *SST* model was therefore used for the main investigation.

The effect of including buoyancy terms in the turbulence model was also investigated. It



**Figure A.2:** Non-dimensional turbulent kinetic energy in tall cavity

was found that including buoyancy source terms in the  $k$  and  $\omega$  equations had only a small effect on the predicted turbulent kinetic energy and heat transfer. The buoyancy source terms were therefore not used in the main investigation.

# **Appendix B**

## **Casing Rig Inlet Design**

### **B.1 Initial Inlet Design**

The inlet used on the casing rig for the first experimental campaign in 2015 was a straight section of duct 0.32 m length with a bellmouth at the inlet. There were no flow conditioning features such as honeycombs or contractions in the inlet design. It was found that this inlet did not provide a sufficiently uniform inlet flow. Looking at smoke flow visualisation of the inlet, Fig.B.1, it can be seen that large corner vortices are present. The flow is recirculating which has carried the smoke further upstream than where it was injected. The vortices propagated downstream into the heated section of the rig giving a complex inflow into the rig. This flow then made it very hard to compare the measured temperature profiles with CFD results.

### **B.2 Contraction Design**

It was decided that a new inlet was to be designed for the second experimental campaign in 2016. The aim of the new inlet was to obtain a uniform flow, free of large flow structures, at the start of the measurement section of the rig. The new inlet would have several features in order to obtain the desired flow-field, shown schematically in Fig.B.2. These features include:

- Bellmouth, this prevents flow separation from the front of the inlet
- Contraction, this causes flow acceleration into the inlet and causes the thinning of the



**Figure B.1:** Smoke flow visualisation in initial inlet

boundary layers

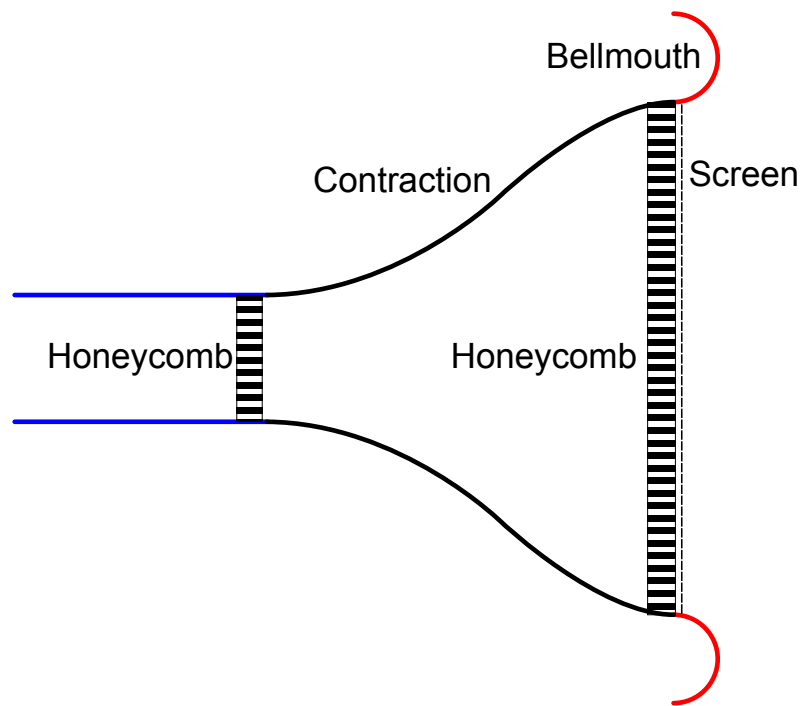
- Honeycomb, to act as a flow straightener
- Screen, to break up large flow structures

The new inlet with the contraction and flow conditioning features can be seen in Fig.B.3.

## B.2.1 Contraction

The most important factor in designing a contraction is the area ratio [21], for this design a ratio of 9 was chosen. The end of the contraction has dimensions 0.32 m by 0.502 m to match the measurement section of the casing rig. The start of the contraction then has dimensions 0.96 m by 1.506 m to give the required contraction ratio. In order to minimise boundary layer growth it was decided to avoid any discontinuities in both the gradient and rate of change of gradient of the contraction walls. This would be achieved using 5th order Bézier curves. The formula for a 5th order Bézier curve is given by:

$$B(X) = \sum_{i=0}^5 \binom{5}{i} (1-X)^{5-i} X^i P_i \quad (\text{B.1})$$



**Figure B.2:** Schematic of Contraction Inlet



**Figure B.3:** Inlet Contraction

Where  $X = x/L$  the  $x$  coordinate normalised by the contraction length  $L$ . The polynomial constants  $P_i$  are (0.16, 0.16, 0.16, 0.48, 0.48, 0.48) for the curve defining the vertical contraction and (0.251, 0.251, 0.251, 0.753, 0.753, 0.753) for the curve defining the horizontal contraction. These values set the start and end locations of the contraction. As well as setting the gradient and rate of change of gradient at the start and end of the contraction to zero.

The length of the contraction was set to 1 m, the length was determined by running steady RANS simulations for a range of lengths and noting the outlet velocity and total pressure profiles.

## **B.2.2 Flow Conditioning Features**

Located around the inlet was a bellmouth, this was designed to avoid a sharp edge at the inlet. This reduced the chance of flow separating at the inlet and causing a non-uniform flow entering the contraction.

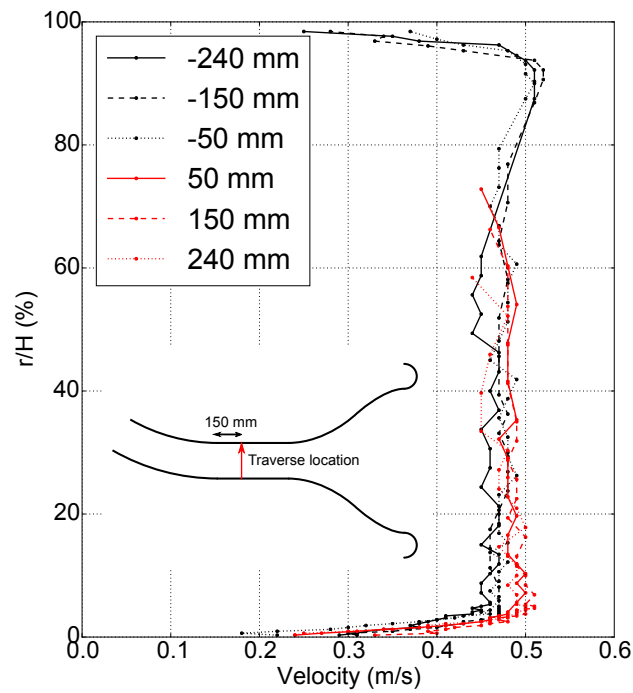
A screen was designed to be added to the very inlet of the rig, the aim of this was to break up large flow structures and prevent them from entering the rig. The screen was made of a fine metal mesh.

Honeycombs were designed to be added before and after the contraction to act as flow straightening devices. The honeycomb was 40 mm in depth and covered the full duct width and height. The honeycomb core diameter was 6.3 mm which gave an aspect ratio of 6.3 for the honeycomb cells.

## **B.3 Inlet Flowfield Validation**

In order to check if the inlet was providing a uniform flowfield at the start of the measurement section, several traverses with a velocity probe were conducted. Six traverses were conducted at a location 150 mm upstream from the start of the heated wall and at locations  $\pm 50$ ,  $\pm 150$  and  $\pm 241$  mm from the duct centreline. As can be seen in Fig.B.4 the profiles are all similar, which shows there is little velocity variation across the duct width. The velocity profiles are also very flat across the duct height, this shows that the inlet section





**Figure B.4:** Velocity profiles at rig inlet

of the rig is performing as required. The new inlet design is producing an inlet condition which allows for simpler analysis of the measurement section flowfield.

## B.4 Conclusions

The inlet used on the gas turbine casing rig for the 2015 experimental campaign was found to not provide a uniform flow into the measurement section of the rig. A new inlet was designed by the author for the 2016 experimental campaign. The new inlet design had several features to provide a uniform flow at the beginning of the measurement section. These features included a 9:1 contraction as well as honeycombs and a screen. Results obtained by velocity traverses showed that the inlet velocity profile was uniform with the new inlet design.

# Appendix C

## Instrumentation Design

### C.1 Introduction

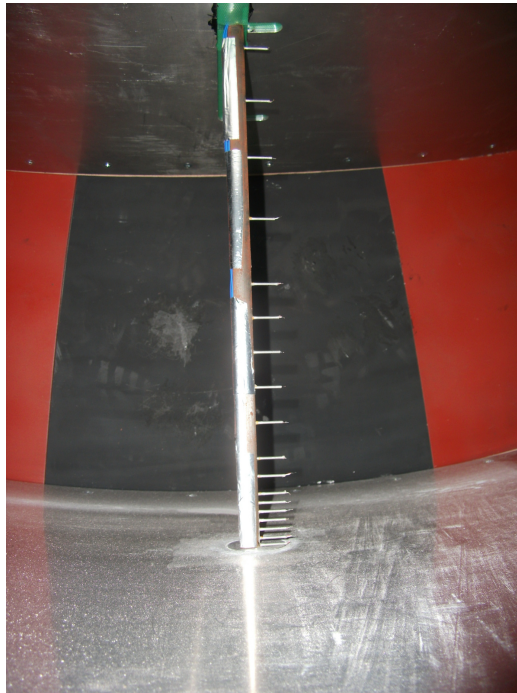
Several different measurement techniques were used in the gas turbine casing rig. The measurements were designed to provide a CFD validation dataset and so measurements were taken to provide the necessary boundary conditions and validation profiles.

### C.2 Thermocouples

Temperature profiles from the casing rig were obtained using rakes of thermocouples at each of the six measurement locations, shown in Fig.C.1. The rake consisted of 18 thermocouples attached to a bar to provide support. The thermocouples used were K-type class 1 with a wire diameter of 0.2 mm and an accuracy of  $\pm 0.9^{\circ}\text{C}$ . Analysis gives the response time of the thermocouples in free air to be 0.49 s, equivalent to 2.0 Hz. The locations of the 18 thermocouples on the rake are given in Table.C.1.

In order to obtain wall temperature boundary conditions 78 thermocouples were used to measure the outer wall temperature and 18 thermocouples were used to measure the inner wall temperature.

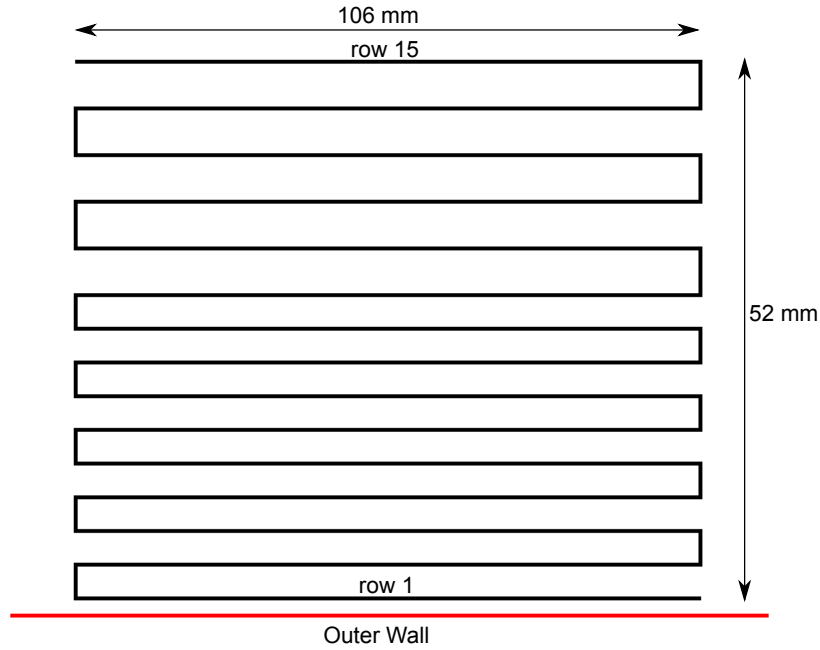
The thermocouples were sampled at a frequency of 1 Hz for 300 seconds. This 5 minute sampling period allowed time for the averages to converge.



**Figure C.1:** Thermocouple Rake

| Thermocouple | Distance from<br>outer wall (mm) | % Duct height |
|--------------|----------------------------------|---------------|
| 1            | 1                                | 0.3           |
| 2            | 5                                | 1.6           |
| 3            | 10                               | 3.1           |
| 4            | 15                               | 4.7           |
| 5            | 20                               | 6.3           |
| 6            | 25                               | 7.8           |
| 7            | 30                               | 9.4           |
| 8            | 40                               | 12.5          |
| 9            | 50                               | 15.6          |
| 10           | 70                               | 21.9          |
| 11           | 90                               | 28.1          |
| 12           | 110                              | 34.4          |
| 13           | 130                              | 40.6          |
| 14           | 150                              | 46.9          |
| 15           | 190                              | 59.4          |
| 16           | 230                              | 71.9          |
| 17           | 270                              | 84.4          |
| 18           | 310                              | 96.9          |

**Table C.1:** Rake thermocouple locations



**Figure C.2:** Optical fibre schematic

### C.3 Fibre Measurement

Temperature data was also obtained using the optical fibre temperature sensor, this allowed a 2D temperature profile to be obtained simultaneously. The fibre sensor used was a LUNA ODiSI-B10 system. This was operated in high resolution mode allowing measurements to be taken with spacings of 0.64 mm along the fibre at a frequency of 11.9 Hz and an accuracy of  $\pm 1.6^{\circ}\text{C}$ . The optical fibre was arranged into a pattern of 15 rows, this was placed normal to the flow to allow a temperature profile across the duct width to be captured, this is shown schematically in Fig.C.2. The location of each row of the fibre is given in Table.C.2. The fibre system was also traversed close to the wall and the first row of fibre used to give detailed temperature measurements that could be used to calculate the wall heat flux.

### C.4 Velocity Traverse

The inlet velocity profile was measured to ensure the inlet section of the rig was providing a uniform flow into the measurement section of the rig. A velocity traverse was performed at a location 150 mm upstream from the start of the heated wall using a hot-wire probe. The probe was traversed vertically through the duct at locations  $\pm 50$ ,  $\pm 150$  and  $\pm 241$  mm

| Row | Distance from<br>outer wall (mm) | % Duct height |
|-----|----------------------------------|---------------|
| 1   | 1                                | 0.3%          |
| 2   | 4                                | 1.3%          |
| 3   | 7                                | 2.2%          |
| 4   | 10                               | 3.1%          |
| 5   | 13                               | 4.1%          |
| 6   | 16                               | 5.0%          |
| 7   | 19                               | 5.9%          |
| 8   | 22                               | 6.9%          |
| 9   | 25                               | 7.8%          |
| 10  | 28                               | 8.8%          |
| 11  | 33                               | 10.3%         |
| 12  | 38                               | 11.9%         |
| 13  | 43                               | 13.4%         |
| 14  | 48                               | 15.0%         |
| 15  | 53                               | 16.6%         |

**Table C.2:** Optical fibre row locations

from the duct centreline. It was found that the velocity profile was very flat and had little variation across the duct width. The probe used was a KANOMAX 0964-02 temperature compensated omni-directional sensor, with a KANOMAX 6332D airflow transducer.

## C.5 Laser Based Velocity Measurement

It is not possible to use hot-wire based velocity measurement techniques in the measurement section of the rig. This is due to the non-isothermal flow causing temperature fluctuations, these temperature fluctuations cannot be distinguished from velocity fluctuations by the hot wire and are difficult to compensate for. The hot-wire could be used in the inlet section due to the flow being isothermal in this region.

Laser based systems do not suffer from this temperature limitation so two different laser based techniques were tried, Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA). Both techniques require the flow to be seeded with particles that can be visualised with the laser but it was found that getting adequate seeding was difficult. For seeding both smoke particles, generated by heating oil, and particles of fine aluminium oxide powder were tried. The low velocity meant that the aluminium particles did not get

carried with the flow. Whereas the smoke was still hot from being vaporised oil, which meant the smoke was buoyant. It was also difficult to get the required smoke density in the rig to allow for reliable measurements. Due to this problem with the flow seeding and limited experiential time it was not possible to obtain velocity data in the measurement section of the rig.

*And now for something completely different. . .*