

1D and 3D simulation of a water-cooled reactor cavity cooling system experimental facility

P.F. Niemand



orcid.org / 0000-0002-0193-8004

Thesis accepted in fulfillment of the requirements for the degree *Doctor of Philosophy in Nuclear Engineering* at the North-West University

Promoter: Prof CGDK du Toit

Graduation: August 2021

Student number: 22134530

PREFACE

Acknowledgement: This work is based on the research supported by the South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa (Grant No 61059).

Disclaimer: Any opinion, finding and conclusion or recommendation expressed in this material is that of the author(s) and the NRF does not accept any liability in this regard.

I would like to thank my supervisor Prof. C.G. du Toit for his continued guidance throughout the duration of the study. The k-factors used to represent the secondary losses, as well as the associated dimensions were calculated by Prof. du Toit. The article shown in Appendix E was also written by Prof. du Toit.

I would also like to thank Mr. Vincent Brits from M-Tech Industrial for his advice on the Flownex/Fluent coupling. Mr. Brits is the original creator of the coupled Flownex/Fluent file that was used as a blueprint for the subsequent simulations.

A poem about simulations

Simulations simulations

big or small

simulations simulations

I have broken them all

Floating point exceptions

Failures to converge

Machine check exceptions

and blue screens of death

Not the best vitamin B
Nor all the filter coffee
can help you win
Its broken again
Start over
discard files
and empty recycle bin

The residuals don't want to go low
but they did a while ago
what has changed
I don't know

What keeps a person going
surrounded by folders filled with failures
You will feel silly, about your negativity
If you knew the answer was around the corner
But the deadline is tomorrow

Keep on grinding
keep on with the trial and error.
You go in as one person
and emerge a little better.
The simulations might never succeed,
but it will teach you patience;
a quality we all need.

Soli Deo gloria

DECLARATION

I, P.F. Niemand declare herewith that the thesis entitled, 1D and 3D simulation of a water-cooled reactor cavity cooling system experimental facility which I herewith submit to the North-West University as partial completion of the requirements set for the PhD Nuclear Engineering degree, is my own work, has been text edited and has not already been submitted to another university.

I understand and accept that the copies that are submitted for examination are the property of the university.

Signature of student  University-number 22134530

Signed at Klerksdorp this 27th day of August 2020

ABSTRACT

The fourth-generation nuclear reactors have many safety features, including the reactor cavity cooling system (RCCS). This system is intended to remove decay heat from the reactor cavity. Designs exist for both air-cooled and water-cooled configurations. It operates completely passively and is designed to operate without external inputs.

The accurate simulation of the phenomena in a reactor cavity cooling system is important for design calculations, to ensure reactor designs that are passively safe. The codes that are used to simulate the phenomena must therefore be verified and validated. One method to validate both a particular design and simulation code is to build an experimental facility based on a prototype. In many cases scale models are constructed of prototype designs. The facility constructed at the University of Wisconsin (Madison) is an example of a scale model of an RCCS. The facility represents a 5° radial slice of the RCCS of the modular high temperature gas-cooled reactor (MHTGR) design.

The prime focus of the study is the heated section/heater box of the facility. A study was done by Lisowski (2013) which encompassed single and two-phase experiments as well as a simplified version that was numerically simulated. This study aims to expand on the single-phase simulations done by Lisowski, by making fewer geometry simplifications.

The study focussed on simulating the heated section of the UW RCCS facility by using 3D computational fluid dynamics (CFD) and simulating the full loop by using 1D system CFD. The 3D simulations explored different geometric configurations, namely circular riser tubes with and without fins, square tubes with and without fins and rectangular tubes with and without fins. The heat fluxes and temperatures on the surfaces were examined and compared between the different geometry configurations.

The 3D CFD code was also coupled with a 1D systems CFD code. The systems were coupled such that the 1D and 3D code would transfer boundary conditions. The codes were linked such that each respective code's strength could be exploited for the best combination of accuracy and effective use of the computational resources. Two variations were simulated and similarly to the 3D cases, the heat fluxes and temperatures on the various surfaces were examined for comparison.

The limitations of the study mean that the results can't be used as a direct comparison with the experiments but can nonetheless be used to philosophically understand the phenomena that occur within the facility.

The finned configurations shielded the back wall from direct radiation heat transfer from the heater wall, which is preferable in terms of preserving the structural integrity of the reactor cavity wall. The cases without fins resulted in higher temperatures on the back walls, but more uniform temperature distributions around the circumference of the riser tubes. This in turn resulted in lower stresses due to uniform thermal expansion.

Radiation heat transfer was the dominating mechanism, but the convection heat transfer nonetheless contributed a non-negligible fraction of the heat transfer.

The convection heat transfer coefficients were calculated on the surfaces in the air-filled cavity and for the inner surfaces of the water-cooled riser tubes.

Two variations of the coupled methodology were tested, and the wall temperatures compared well between the coupled methods as well as with the 3D method. The convection heat transfer coefficients were also compared for the inner surfaces of the middle riser tube for the three approaches. The coefficients compared reasonably well in terms of the trends that were exhibited by the different approaches on the four inside surfaces.

Keywords: Natural circulation, reactor cavity cooling system, 1D systems CFD, 3D CFD, 1D/3D coupling.

OPSOMMING

Die vierde generasie kernkrag reaktore beskik oor talle veiligheids stelsels, ondermeer die reaktor ruimte verkoeling stelsel. Hierdie stelsel se doel is om die hitte wat vrygestel word deur die proses van radioaktiewe verval uit die reaktor ruimte te verwyder. Daar bestaan ontwerpe vir lug- sowel as water verkoelde konfigurasies van die stelsel. Die stelsel kan funksioneer sonder eksterne krag of insette van 'n operateur.

Die akkurate simulاسie van die verskynsels in die reaktor ruimte verkoelingstelsel is belangrik vir die ontwerp berekeninge en om te verseker dat die reaktor ontwerpe inherent veilig is. Die programkodes wat ingespan word om die verskynsels te simuleer moet dus geverifieer en gevalideer word. Een metode om validasie te bewerkstellig van beide 'n ontwerp of 'n kode, is om 'n eksperimentele fasiliteit te bou van 'n prototipe ontwerp. Deur toetse op die fasiliteit kan validasie bewerkstellig word. Die fasiliteit wat gebou is deur die Universiteit van Wisconsin (Madison) is 'n voorbeeld van 'n skaalmodel van 'n reaktor ruimte verkoelingstelsel. Die fasiliteit verteenwoordig 'n 5° radiale snit van die ontwerp van die stelsel vir 'n modulêre hoë temperatuur gas-verkoelde reaktor.

Die hooffokus van die studie is die verhitte ruimte van die fasiliteit. 'n Studie is gedoen deur Lisowski (2013) wat enkel en twee fase vloeï eksperimente behels het, sowel as vereenvoudigde numeriese modelle. Hierdie studie beoog om uit te brei op die enkel fase simulاسies deur Lisowski deur minder geometriese vereenvoudigings te maak aan die geometrie wat in die numeriese model in ag geneem word.

Die studie fokus op die simulاسie van die verhitte ruimte van die bogenoemde fasiliteit deur gebruik te maak van 3D berekenings vloeï dinamika as alleenstaande metode en as gekoppel met 1D sisteem berekenings vloeï dinamika. Deur gebruik te maak van die 3D metode, is verskillende geometrieë vir die stygbuise ondersoek, naamlik ronde buise met en sonder vinne, vierkantige buise, met en sonder vinne en reghoekige buise met en sonder vinne. Die hitte vloed en temperatuur verspreidings op die belangrike oppervlaktes is vergelyk.

Die 3D kode was ook gekoppel met 'n 1D kode. Die stelsel was gekoppel sodanig dat die verskillende kodes die randwaardes sou uitruil. Die kodes was gekoppel om die sterkpunte van elke kode te gebruik om sodoende 'n optimum kombinasie te kry tussen akkuraatheid en effektiewe gebruik van die berekeningshulpbronne. Twee variasies van hierdie metode is gedoen en soortgelyk aan die 3D metode, was die hittevloed en temperatuur verspreidings ondersoek en vergelyk.

Die beperkings van die studie beteken dat die resultate nie ten volle as 'n direkte vergelyking met die eksperimente gebruik kan word nie. Dit kan nietemin gebruik word om in 'n filosofiese konteks die verskynsels te verduidelik wat in die fasiliteit voor kom.

Die hitte oordrag in die lug gevulde ruimte word oorheers deur straling hitte oordrag, maar die invloed van die konveksie hitte oordrag was nie weglaatbaar nie.

Die konveksie hitte oordrag koëffisiënte was bereken vir die oppervlaktes in die lug gevulde ruimtes, sowel as die water verkoelde stygbuise.

Twee variasies van die gekoppelde metode is getoets en die verspreidings van die wandtemperatuur vergelyk goed met mekaar, sowel as met die temperatuur verkry deur die 3D metode. Die konveksie hitte oordrag koëffisiënte is ook bereken vir die drie metodes wat vergelyk is. Die koëffisiënte het redelike ooreenstemming getoon tussen die benaderings.

Sleutelwoorde: Natuurlike sirkulasie, reaktor ruimte verkoeling stelsel, 1D stelsel berekenings vloeï dinamika, 3D berekenings vloeï dinamika, 1D/3D koppeling

Table of contents

LIST OF TABLES	XVII
LIST OF FIGURES.....	XX
LIST OF SYMBOLS/ABBREVIATIONS.....	XXXII
CHAPTER 1 - INTRODUCTION.....	1
1.1 Introduction	1
1.2 Background	1
1.2.1 Safety considerations for gas-cooled reactors	1
1.2.2 Description of the RCCS.....	2
1.2.3 Reactor designs employing the RCCS.....	4
1.2.3.1 Primary air-cooling (GT-MHR)	4
1.2.3.2 Primary water cooling, Secondary/tertiary air cooling (HTR-10).....	5
1.2.3.3 Primary water cooling, with forced secondary air cooling (SC-HTGR)	6
1.2.3.4 Primary water cooling (PBMR, VGM).....	6
1.2.3.5 Primary air cooling, with optional primary water cooling.....	9
1.2.3.6 Primary air cooling, secondary water cooling (PMR200).....	10
1.2.3.7 Simultaneous air- and water cooling by using a water pool.....	10
1.2.4 Scaling of experimental facilities	11
1.2.5 Facility overview	11
1.2.6 Summary	13

1.3	Problem statement and aim	13
1.4	Scope	14
1.5	Layout of work	14
CHAPTER 2 - LITERATURE SURVEY		15
2.1	Introduction	15
2.2	Instabilities relating to the RCCS	15
2.3	Simulations of accident scenarios	18
2.4	Experimental and numerical models.....	19
2.5	Methods used for CFD modelling.....	20
2.6	1D and 1D/3D coupled methods	23
2.7	Philosophies in coupling	25
2.8	Mixed convection	25
2.9	Conclusion.....	28
CHAPTER 3 THEORETICAL BACKGROUND		29
3.1	Introduction	29
3.2	Computational Fluid Dynamics (CFD).....	29
3.3	Non-dimensional numbers.....	32
3.4	1D System CFD theory	35
3.5	Conclusion.....	39
CHAPTER 4 FACILITY DESCRIPTION		41
4.1	Introduction	41

4.2	UW facility overview	41
4.3	Conclusion.....	45
CHAPTER 5 - METHODOLOGY VERIFICATION.....		46
5.1	Introduction	46
5.1.1	Simple cubical cavity simulation	46
5.1.2	Natural circulation in an air-cooled cavity.....	48
5.1.3	Isothermal simulation of the riser tubes.....	51
5.1.4	Mesh dependence and verification study	51
5.1.5	k- ϵ turbulence model.....	62
5.1.6	Fluent vs. Star-CCM+ 3D simulation.....	64
5.1.7	Conclusion.....	66
CHAPTER 6 – 3D CFD MODELLING OF THE FACILITY		68
6.1	Introduction	68
6.2	Model descriptions.....	69
6.2.1	Mesh properties.....	69
6.2.2	Boundary conditions and solver properties	70
6.2.3	Conclusion.....	71
CHAPTER 7 – COUPLED 1D/3D MODELLING OF THE FACILITY		72
7.1	Introduction	72
7.2	Model variations	72
7.3	Variation B model (Manifolds modelled using Flownex)	73

7.3.1	Flownex model (Variation B).....	73
7.3.2	Fluent model (Variation B)	77
7.4	Variation C model (natural circulation)	77
7.4.1	Flownex model (Variation C).....	78
7.4.2	Fluent model (Variation C)	82
7.5	Conclusion.....	83
CHAPTER 8 RESULTS		84
8.1	Introduction	84
8.2	Results for the 3D cases	84
8.2.1	Overview	84
8.2.2	Round tubes with fins	86
8.2.2.1	Heat fluxes	87
8.2.2.2	Temperature distributions	88
8.2.2.3	Velocity vectors	94
8.2.2.4	Heat transfer coefficients	99
8.2.2.5	Conclusion.....	105
8.2.3	Round tubes without fins	106
8.2.3.1	Heat fluxes	107
8.2.3.2	Temperature distributions	108
8.2.3.3	Velocity vectors	112
8.2.3.4	Heat transfer coefficients	117
8.2.3.5	Conclusion.....	121

8.2.4	Square tubes with fins	122
8.2.4.1	Heat fluxes	123
8.2.4.2	Temperature distributions	124
8.2.4.3	Velocity vectors	129
8.2.4.4	Heat transfer coefficients	133
8.2.4.5	Conclusion.....	138
8.2.5	Square tubes without fins	139
8.2.5.1	Heat fluxes	140
8.2.5.2	Temperature distributions	140
8.2.5.3	Velocity vectors	145
8.2.5.4	Heat transfer coefficients	149
8.2.5.5	Conclusion.....	153
8.2.6	Rectangular tubes with fins.....	155
8.2.6.1	Heat fluxes	156
8.2.6.2	Temperature distributions	157
8.2.6.3	Velocity vectors	163
8.2.6.4	Heat transfer coefficients	169
8.2.6.5	Conclusion.....	175
8.2.7	Rectangular tubes without fins.....	177
8.2.7.1	Heat fluxes	178
8.2.7.2	Temperature distributions	179
8.2.7.3	Velocity vectors	183
8.2.7.4	Heat transfer coefficients	188

8.2.7.5	Conclusion.....	192
8.2.8	Summary on the 3D CFD results	193
8.2.8.1	Heater and back wall temperatures	193
8.2.8.2	Riser tube wall temperatures	193
8.2.8.3	Cavity air temperature	194
8.2.8.4	Radiation heat transfer	194
8.2.8.5	Convection heat transfer coefficients in the cavity	195
8.2.8.6	Convection heat transfer coefficients in the riser tubes	196
8.2.9	Concluding remarks.....	197
8.3	Results for the coupled cases	198
8.3.1	Manifolds modelled by using Flownex (Variation B)	198
8.3.2	Natural circulation coupled case (Variation C)	205
8.3.3	Summary of the 1D/3D coupled results	211
8.4	Comparison between the 3D and coupled 1D/3D results	212
CHAPTER 9 - CONCLUSIONS.....		216
9.1	Overview	216
9.2	Conclusion from 3D CFD studies	216
9.3	Conclusion from coupled 1D/3D CFD studies	217
9.4	Recommendations.....	218
BIBLIOGRAPHY		219
APPENDIX A MATERIAL PROPERTIES		224

A.1	Material properties.....	224
APPENDIX B MESH QUALITIES		230
B.1	Mesh quality.....	230
B.2	Mesh dependence study	230
B.3	Mesh number 1 (section 5.1.4).....	230
B.4	Mesh number 2 (section 5.1.4).....	233
B.5	Mesh number 3 (section 5.1.4).....	235
B.6	Mesh qualities for the 3D CFD	239
B.7	Round tubes with fins (section 6.2.1).....	239
B.8	Round tubes without fins (section 6.2.1)	241
B.9	Square tubes with fins (section 6.2.1)	244
B.10	Square tubes without fins (section 6.2.1)	247
B.11	Rectangular tubes with fins (section 6.2.1).....	250
B.12	Rectangular tubes without fins (section 6.2.1)	252
B.13	Mesh qualities for the coupled systems	255
B.14	Variation B (section 7.3.2).....	255
B.15	Variation C (section 7.4.2).....	257
APPENDIX C SOLUTION MONITORING		260
C.1	Introduction	260
C.2	Isothermal water region (section 5.1.3).....	260
C.3	Mesh dependence study	264
C.4	Mesh number 1	264

C.5	Mesh number 2	266
C.6	Mesh number 3	269
C.7	k- ϵ turbulence model.....	272
C.8	Fluent 3D model.....	274
C.9	Round riser tubes with fins.....	275
C.10	Round tubes without fins.....	276
C.11	Square tubes with fins	278
C.12	Square fins without fins	281
C.13	Rectangular with fins	282
C.14	Rectangular tubes without fins.....	283
C.15	Coupled simulations	285
C.16	Manifolds modelled by using Flownex (Variation B).....	285
C.17	Natural circulation Variation C	286
APPENDIX D ADDITIONAL PARAMETERS AND PROPERTIES CALCULATED		288
D.1	Additional parameters and properties	288
D.2	Circular tubes with fins	288
D.3	Circular tubes without fins.....	292
D.4	Square tubes with fins	297
D.5	Square tubes without fins	301
D.6	Rectangular tubes with fins	305
D.7	Rectangular tubes without fins.....	309
APPENDIX E PHILOSOPHIES IN COUPLING.....		313

E.1	Philosophies in coupling	313
	APPENDIX F ARTICLE TO BE SUBMITTED	314

LIST OF TABLES

Table 4-1. Results of the isothermal pressure drop experiments (Lisowski, 2013).....	44
Table 4-2. Experimental runs done by Lisowski (2013).....	44
Table 5-1. Pressure drop and mass flow rate division within the water region.	53
Table 5-2. Mesh dependence study properties.....	56
Table 6-1. Mesh quality for the tested geometry.....	69
Table 6-2. Boundary conditions for the various tested geometries.....	70
Table 7-1. Boundary conditions of Variation B.....	73
Table 7-2. k-values specified to account for secondary losses (variation B).	73
Table 7-3. Pipe lengths, k-values and node elevations for Variation C.	80
Table 8-1. Heat flux on the riser and fin surfaces (For the round tubes with fins case).	87
Table 8-2. Heat flux on the riser tube surfaces (For the round tubes without fins).....	107
Table 8-3. Selected results for the riser tubes.	124
Table 8-4. Selected results for the riser tubes.	140
Table 8-5. Selected heat fluxes for the case with rectangular tubes with fins.....	156
Table 8-6. Selected results from the case with rectangular tube without fins.	178
Table 8-7. Selected results for Variation B.....	199
Table 8-8. Selected results for Variation C from the 3D model.	206
Table D.2-1. Additional parameters (circular tubes with fins) associated with the front surface inside Riser 2.....	289
Table D.2-2. Additional parameters (circular tubes with fins) associated with the back surface inside Riser 2.....	290

Table D.2-3. Table Additional parameters (circular tubes with fins) associated with the left surface inside Riser 2.....	291
Table D.2-4. Additional parameters (circular tubes with fins) associated with the right surface inside Riser 2.....	292
Table D.3-1. Additional parameters (circular tubes without fins) associated with the front surface on the inside of Riser 2.....	293
Table D.3-2. Additional parameters (circular tubes without fins) associated with the back surface on the inside of Riser 2.....	294
Table D.3-3. Additional parameters (circular tubes without fins) associated with the left surface on the inside of Riser 2.....	295
Table D.3-4. Additional parameters (circular tubes without fins) associated with the right surface on the inside of Riser 2.....	296
Table D.4-1. Additional parameters (square tubes with fins) associated with the front surface on the inside of Riser 2.....	297
Table D.4-2. Additional parameters (square tubes with fins) associated with the back surface on the inside of Riser 2.....	298
Table D.4-3. Additional parameters (square tubes with fins) associated with the left surface on the inside of Riser 2.....	299
Table D.4-4. Additional parameters (square tubes with fins) associated with the right surface on the inside of Riser 2.....	300
Table D.5-1. Additional parameters (square tubes without fins) associated with the front surface on the inside of Riser 2.....	301
Table D.5-2. Additional parameters (square tubes without fins) associated with the back surface on the inside of Riser 2.....	302
Table D.5-3. Additional parameters (square tubes without fins) associated with the left surface on the inside of Riser 2.....	303
Table D.5-4. Additional parameters (square tubes without fins) associated with the right surface on the inside of Riser 2.....	304

Table D.6-1. Additional parameters (rectangular tubes with fins) associated with the front surface on the inside of Riser 2.	305
Table D.6-2. Additional parameters (rectangular tubes with fins) associated with the back surface on the inside of Riser 2.	306
Table D.6-3. Additional parameters (rectangular tubes with fins) associated with the left surface on the inside of Riser 2.	307
Table D.6-4. Additional parameters (rectangular tubes with fins) associated with the right surface on the inside of Riser 2.	308
Table D.7-1. Additional parameters (rectangular tubes without fins) associated with the front surface on the inside of Riser 2.	309
Table D.7-2. Additional parameters (rectangular tubes without fins) associated with the back surface on the inside of Riser 2.	310
Table D.7-3. Additional parameters (rectangular tubes without fins) associated with the left surface on the inside of Riser 2.	311
Table D.7-4. Additional parameters (rectangular tubes without fins) associated with the right surface on the inside of Riser 2.	312

LIST OF FIGURES

Figure 1.2-1. Conceptual depiction of the RCCS. (Shin et al., 2017a).	3
Figure 1.2-2. Schematic of the layout of the RCCS in the GT-MHR. (Neylan et al., 1994).	4
Figure 1.2-3. PRHRS of the HTR10 design. (Zhao et al., 2019).	5
Figure 1.2-4. Decay heat removal system. (Lommers et al., 2014).	6
Figure 1.2-5. Schematic representation of three of the cooling systems in the PBMR (Slabber et al., 2006).	7
Figure 1.2-6. VGM reactor (IAEA, 2000).	8
Figure 1.2-7. A RCCS design that makes use of water and air proposed by Lim et al. (2018).	9
Figure 1.2-8. Possible RCCS layout of the PMR200. (Bae et al., 2012).	10
Figure 2.2-1. Experimental facility layout (Pal et al., 2016).	17
Figure 2.8-1. Comparison of correlations and experimental results. (Shin et al., 2017b).	27
Figure 3.2-1. Wall treatment for different y^+ values. (Raghav, 2018).	32
Figure 3.4-1. Eulerian reference frame used by Flownex for mass flow. (M-Tech Industrial, 2019b).	36
Figure 4.2-1. The process layout of the experimental facility. (Lisowski, 2013).	42
Figure 4.2-2. Cross sectional view of the heated test section. (Lisowski, 2013).	42
Figure 4.2-3. Pressure drop as a function of flow rate (Lisowski, 2013).	43
Figure 5.1-1. Velocity distribution in the cubical cavity.	47
Figure 5.1-2. Temperature distribution within the cubical cavity.	47
Figure 5.1-3. Air-filled cavity geomtery.	48
Figure 5.1-4. Comparison between calculated and measured results.	49

Figure 5.1-5. Geometry used to model the water region.	51
Figure 5.1-6. Residual criteria for the isothermal simulation of the water region (k- ϵ model).....	52
Figure 5.1-7. Geometry used in the CFD model.	55
Figure 5.1-8. Lateral temperature distribution in the cavity.	57
Figure 5.1-9. Comparison of the temperature profile close to the heaterwall.	58
Figure 5.1-10. The comparison between the temperature profiles in the front cavity against the middle riser's front wall.	59
Figure 5.1-11. The comparison between the temperature profiles on the inside of the middle riser.	60
Figure 5.1-12. Comparison between the temperature profiles in the back cavity against the middle riser's back wall.	60
Figure 5.1-13. Comparison between the temperature profiles in the back cavity against the back wall.	61
Figure 5.1-14. The riser tube wall temperatures as obtained by using the k- ϵ turbulence model.	63
Figure 5.1-15. Riser tube wall temperature comparison between Fluent and Star-CCM+.	65
Figure 7.3-1. The variation B Flownex model.	76
Figure 7.4-1. Flownex model of the Variation C model.	79
Figure 8.2-1. Locations where temperature probes were placed.	85
Figure 8.2-2. Naming convention used in the presentation of the results.	86
Figure 8.2-3. Riser tube water temperature distributions.	89
Figure 8.2-4. Riser tube wall temperature distribution for Riser 1.	89
Figure 8.2-5. Riser tube wall temperature distribution for Riser 2.	90
Figure 8.2-6. Temperature distributions within the walls of Riser 3.	90

Figure 8.2-7. Fin temperature distribution.	91
Figure 8.2-8. Temperature contour of the fins and riser tube walls.	92
Figure 8.2-9. Air temperature distributions in the front and back cavity.	93
Figure 8.2-10. Heater and back wall temperature distributions.	93
Figure 8.2-11. Velocity vectors in the heated section.	94
Figure 8.2-12. Vectors on a section plane in the middle of the heated section.	95
Figure 8.2-13. Velocity vector in the back cavity.	96
Figure 8.2-14. Velocity profiles in the middle riser.	97
Figure 8.2-15. Velocity vectors in the middle riser (continued).	98
Figure 8.2-16. Aicher number as a function of elevation.	100
Figure 8.2-17. Convection heat transfer coefficient as a function of elevation.	101
Figure 8.2-18. Nusselt number ratio vs. Aicher number.	101
Figure 8.2-19. Convection heat transfer coefficients on the outer surfaces of the fins.	102
Figure 8.2-20. Convection heat transfer coefficients on the outer surfaces of the riser tubes.	103
Figure 8.2-21. Convection heat transfer coefficients on the surfaces of the heater – and back wall.	104
Figure 8.2-22. Geometry and naming convention as referred to in the results.	106
Figure 8.2-23. Riser tube temperature distributions.	108
Figure 8.2-24. Wall temperature distributions for Riser 1.	109
Figure 8.2-25. Wall temperature distributions for Riser 2.	110
Figure 8.2-26. Wall temperature distributions for Riser 3.	110
Figure 8.2-27. Cavity air temperature distributions.	111

Figure 8.2-28. Heater – and back wall temperature distributions.	112
Figure 8.2-29. Velocity vectors of the air in the top part of the heated section.	113
Figure 8.2-30. Velocity vectors in the top part of the front and back cavity.....	114
Figure 8.2-31. Velocity vectors at four elevations in the middle riser tube.....	115
Figure 8.2-32. Velocity profiles in the middle riser (continued).....	116
Figure 8.2-33. Convection heat transfer coefficient on the riser tubes' inner surfaces as a function of elevation.	118
Figure 8.2-34. Aicher number as a function of elevation for the middle riser.....	118
Figure 8.2-35. Nusselt number ratio vs. Aicher number for the middle riser.....	119
Figure 8.2-36. Convection heat transfer coefficients on the outer surfaces of the riser tubes as a function of elevation.	119
Figure 8.2-37. Convection heat transfer coefficient as a function of elevation on the heater- and back wall.	120
Figure 8.2-38. Geometry and naming convention as referred to in the discussion of the results.	122
Figure 8.2-39. Riser water temperature distributions.	125
Figure 8.2-40. Riser 2's surface temperatures as a function of elevation.....	125
Figure 8.2-41. The temperature distributions in the fin walls.....	126
Figure 8.2-42. Temperature contour on the front surfaces of the riser tubes and fins.	127
Figure 8.2-43. Heater- and back wall temperature distributions.	128
Figure 8.2-44. Air temperature distributions in the front and back part of the cavity.	128
Figure 8.2-45. Velocity vectors in the front and back cavity for the case with square tubes with fins.....	129
Figure 8.2-46. Velocity vectors in the front and back cavity (continued).....	130
Figure 8.2-47. Velocity vectors at four elevations in the middle riser.....	131

Figure 8.2-48. Velocity vectors in the middle riser (continued).....	132
Figure 8.2-49. Convection heat transfer coefficient for the inner surfaces of the inner riser tube.....	134
Figure 8.2-50. Aicher number as a function of elevation.....	134
Figure 8.2-51. Convection heat transfer coefficients for the heater- and back wall.	135
Figure 8.2-52. Convection heat transfer coefficient on the outer surfaces of the riser tubes.....	136
Figure 8.2-53. Convection heat transfer coefficients on the outer surfaces on the fins.....	137
Figure 8.2-54. Geometry used in the CFD model.	139
Figure 8.2-55. Water temperature distributions in the middle of the risers.	141
Figure 8.2-56. Riser tube wall temperatures for the middle riser tube.....	142
Figure 8.2-57. Temperature contour on the outer surfaces of the riser tubes.....	142
Figure 8.2-58. Air temperatures in the front and back side of the cavity plotted as a function of elevation.	144
Figure 8.2-59. Heater- and back wall temperatures as a function of elevation.	144
Figure 8.2-60. Velocity vectors in the front – and back cavity.	145
Figure 8.2-61. Velocity vectors in the front - and back parts of the cavity (continued).....	146
Figure 8.2-62. The velocity vectors in the middle riser tube.....	147
Figure 8.2-63. Velocity vectors in the middle riser tube (continued).....	148
Figure 8.2-64. Convection heat transfer coefficients for the inner surfaces of the middle riser tube.....	150
Figure 8.2-65. Aicher number as a function of elevation in the middle riser tube.	151
Figure 8.2-66. Nusselt number plotted as a function of Aicher number.....	151
Figure 8.2-67. Convection heat transfer coefficients on the front – and back cavity sides of the riser tubes.	152

Figure 8.2-68. Convection heat transfer coefficients as a function of elevation.	153
Figure 8.2-69. Geometry used in the CFD model.	155
Figure 8.2-70. Water temperature distributions in the middle of the riser tubes.....	157
Figure 8.2-71. Temperature scalar scene of the first three risers.	158
Figure 8.2-72. Riser tube wall temperature for the case with rectangular tubes with fins.	158
Figure 8.2-73. Fin temperature distributions for the case with rectangular tube with fins.	159
Figure 8.2-74. Temperature contour on the front surface of the risers and fins.	161
Figure 8.2-75. The temperature contour on the front and sides of the riser tubes.	161
Figure 8.2-76. Heater - and back wall temperature distributions.	162
Figure 8.2-77. Cavity air temperature distribution for the case with rectangular tube with fins.	163
Figure 8.2-78. Velocity in the y-direction on a probe in the front cavity next to the third riser.....	164
Figure 8.2-79. Velocity in the y-direction on a probe in front of Fin 4.	164
Figure 8.2-80. Vectors in the front and back cavity.	165
Figure 8.2-81. Vectors in the back cavity.	166
Figure 8.2-82. Vectors in the front cavity.	166
Figure 8.2-83. Velocity vectors in the one of the middle riser tubes.	167
Figure 8.2-84. The velocity vectors inside one of the middle risers (continued).	168
Figure 8.2-85. Convection heat transfer coefficients for the case with rectangular tubes with fins.	170
Figure 8.2-86. Nusselt number ratio vs. Aicher number for the case with rectangular tubes with fins.	171
Figure 8.2-87. Aicher number as a function of elevation for the water in the riser tubes.	172

Figure 8.2-88. Convection heat transfer coefficients on the outer surfaces of the fins.....	173
Figure 8.2-89. Convection heat transfer coefficient on the riser tubes' outer surfaces for the case with rectangular tubes with fins.	174
Figure 8.2-90. Convection heat transfer coefficient for the heater - and back wall as a function of elevation for the case with rectangular tubes with fins.....	175
Figure 8.2-91. Geometry used for the CFD simulation.....	177
Figure 8.2-92. Temperature distributions in the middle of the riser tubes.....	179
Figure 8.2-93. Riser tubes' wall temperature distributions for the case with rectangular tubes without fins.	180
Figure 8.2-94. Temperature contours in the lower part of the heated section.	181
Figure 8.2-95. Temperature distributions on the heater – and back wall.	182
Figure 8.2-96. Cavity air temperature distributions in the front - and back cavity.	183
Figure 8.2-97. Local y-velocity between second and third risers.	184
Figure 8.2-98. Velocity vectors of the air in the front and back part of the cavity.....	184
Figure 8.2-99. Vectors in the middle of the back cavity.....	185
Figure 8.2-100. Velocity vectors inside Riser 3 at four different elevations.	186
Figure 8.2-101. Velocity vectors in one of the middle risers (continued).	187
Figure 8.2-102. Convection heat transfer coefficient on the inner surfaces of Riser 3.....	188
Figure 8.2-103. Aicher number as a function of elevation.	189
Figure 8.2-104. Nusselt number ratio as a function of Aicher number.....	190
Figure 8.2-105. Heater – and back wall convection heat transfer coefficients.....	191
Figure 8.2-106. Convection heat transfer coefficient on the front and back sides of the riser tubes.	191
Figure 8.3-1. Water temperature distribution in the middle of the riser tubes.	200

Figure 8.3-2. Riser wall temperature distributions for Riser 1.....	201
Figure 8.3-3. Riser wall temperature distributions for Riser 2.....	201
Figure 8.3-4. Riser wall temperature distributions for Riser 3.....	202
Figure 8.3-5. Convection heat transfer coefficient on the respective surfaces as a function of elevation for Riser 2.	203
Figure 8.3-6. Aicher number as a function of elevation for the inner surfaces of the middle riser tube.....	204
Figure 8.3-7. Water temperature distribution in the riser tubes for Variation C.....	206
Figure 8.3-8. Wall temperatures in the first riser tube.	207
Figure 8.3-9. Wall temperatures in the second riser tubes.....	208
Figure 8.3-10. Wall temperatures in the third riser tube.	208
Figure 8.3-11. Convection heat transfer coefficient as a function of elevation.....	209
Figure 8.3-12. Aicher number as a function of elevation for Variation C.	210
Figure 8.4-1. Comparison between the wall temperatures between the 3D and coupled approaches.	212
Figure 8.4-2. Comparison between the convection heat transfer coefficients.	214
Figure A-1. Dynamic viscosity of water with the polynomial function.....	225
Figure A-2. Polynomial of conductivity in terms of temperature of water.....	225
Figure A-3. Polynomial function of specific heat capacity in terms of temperature for water.	226
Figure A-4. Polynomial of density in terms of temperature for water.	226
Figure A-5. Polynomial of specific heat capacity in terms of temperature for air.	227
Figure A-6. Conductivity polynomial in terms of temperature for air.	227

Figure A-7. Polynomial of density in terms of temperature for air.....	228
Figure A-8. Polynomial of viscosity in terms of temperature for air.	228
Figure B-1. Mesh number 1 generated by Ansys.....	231
Figure B-2. Orthogonal quality of mesh number 1.	231
Figure B-3. Skewness of mesh number 1.....	232
Figure B-4. Cell aspect ratio for mesh number 1.....	232
Figure B-5. Mesh generated for mesh number 2.	233
Figure B-6. Mesh skewness for mesh number 2.....	234
Figure B-7. Orthogonal quality for mesh number 2.	234
Figure B-8. Cell aspect ratio for mesh number 2.....	235
Figure B-9. Mesh generated for mesh number 3.	235
Figure B-10. Mesh skewness for mesh number 3.....	236
Figure B-11. Mesh orthogonal quality for mesh number 3.	237
Figure B-12. Cell aspect ratio for mesh number 3.....	237
Figure B-13. Contour of wall y^+ values for the finest mesh.....	238
Figure B-14. Mesh used for the round tubes with fins case.	239
Figure B-15. Mesh skewness for the case with round tubes with fins.	240
Figure B-16. Mesh orthogonal quality for the case with round tubes with fins.	240
Figure B-17. Cell aspect ratio distribution for the case with round tubes with fins.	241
Figure B-18. The mesh generated for the case of the circular riser tubes without fins.	242
Figure B-19. Cell skewness distribution for the case with round tubes without fins.....	243
Figure B-20. Cell orthogonal quality distribution for the case round tubes without fins.....	243
Figure B-21. Cell aspect ratio distribution for the case with round tubes without fins.	244

Figure B-22. Mesh for the case with square tubes with fins.	245
Figure B-23. Skewness distribution for the case with square tubes with fins.....	246
Figure B-24. Orthogonal quality distribution for the case with square tubes with fins.	246
Figure B-25. Cell aspect ratio distribution for the case square tubes with fins.....	247
Figure B-26. Mesh generated for the case square tubes without fins.....	248
Figure B-27. Cell skewness distribution for the case with square tubes without fins.	248
Figure B-28. Orthogonal quality distribution for the case with square tubes without fins.	249
Figure B-29. Cell aspect ratio distribution for the case square tubes without fins.....	249
Figure B-30. The mesh generated for the case with rectangular tubes with fins.	250
Figure B-31. Skewness distribution for the case with rectangular tubes with fins.....	251
Figure B-32. Orthogonal quality distribution of the case with rectangular tubes with fins.	251
Figure B-33. Aspect ratio distribution of the case with rectangular tubes with fins.	252
Figure B-34. The mesh generated for the case with rectangular tubes without fins.	252
Figure B-35. Cell skewness distribution for the case with rectangular tubes without fins.	253
Figure B-36. Orthogonal quality distribution for the case rectangular tubes without fins.....	254
Figure B-37. Aspect ratio distribution for the case with rectangular tubes without fins.	254
Figure B-38. Mesh skewness for the mesh generated for variation B.	256
Figure B-39. Mesh orthogonal quality for the mesh generated for variation B.....	256
Figure B-40. Aspect ratio distribution for the mesh generated for Variation B.	257
Figure B-41. Mesh skewness distribution for Variation C.....	258
Figure B-42. Orthogonal quality for the mesh for Variation C.....	258
Figure B-43. Aspect ratio distribution for variation C.....	259
Figure C-1. Residual for 0.37 kg/s case (k- ϵ turbulence).	261

Figure C-2. Residuals for the 0.45 kg/s case (k- ϵ turbulence).....	261
Figure C-3. Residuals for the 0.52 kg/s case (k- ϵ turbulence).....	262
Figure C-4. Residuals for the 0.67 kg/s case (k- ϵ turbulence).....	262
Figure C-5. Residuals for the 0.67 kg/s case (k- ω turbulence model).....	263
Figure C-6. Ansys residuals for Mesh number 1.	264
Figure C-7. Surface average of the outlet temperature of Mesh 1.	265
Figure C-8. Star-CCM+ residual criteria for Mesh number 1.	265
Figure C-9. Star-CCM+ surface averaged outlet temperature monitor for Mesh 1.	266
Figure C-10. Ansys residuals for Mesh 2.	267
Figure C-11. Surface average of outlet temperature obtained by Fluent for Mesh number 2.....	267
Figure C-12. Star-CCM+ residuals for Mesh 2.....	268
Figure C-13. Outlet temperature monitor for Mesh 2.	269
Figure C-14. Residuals for Mesh 3.	270
Figure C-15. Outlet temperature per iteration for Mesh 3.....	270
Figure C-16. Integral of heat flux on the riser tube inner surfaces (Mesh 3).....	271
Figure C-17. Residuals for the Star-CCM+ simulation for Mesh 3.....	271
Figure C-18. Surface average temperature monitor for Mesh 3.	272
Figure C-19. Residual criteria of the k- ϵ model in the verification chapter.....	273
Figure C-20. Surface integral of the heat flux on the outside surface of Riser 2.....	273
Figure C-21. Residual criteria for the Fluent 3D model case.....	274
Figure C-22. Area weighted average of the heat flux on Riser 2's outer surface.....	275
Figure C-23. Residuals for the case with round riser tubes with fins.	276

Figure C-24. Integral of heat flux monitor used for the case with round riser tubes with fins.	277
Figure C-25. Residuals for the case with round tubes without fins.	277
Figure C-26. Outlet temperature monitor for the case with round tubes without fins.	278
Figure C-27. Residuals for the case with square tubes with fins.	279
Figure C-28. Integral of the heat flux monitor for the case with square tubes with fins.	280
Figure C-29. Outlet temperature monitor for the case with square tubes with fins.	280
Figure C-30. Residuals for the case with square tubes without fins.	281
Figure C-31. Surface average heat flux monitor for the case with square tubes with fins.....	281
Figure C-32. Residuals for the case with rectangular tubes with fins.	282
Figure C-33. Average heat flux monitor for the case with rectangular tubes with fins.....	283
Figure C-34. Residuals for the case with rectangular tubes without fins.	284
Figure C-35. Heat flux monitor for the case with rectangular tubes without fins.	284
Figure C-36. Residuals for Variation B.	285
Figure C-37. Outlet temperature monitors for Variation B.	285
Figure C-38. Residuals of the Fluent simulation for Variation C.	287
Figure C-39. Heat energy added per iteration per riser for Variation C.	287

LIST OF SYMBOLS/ABBREVIATIONS

Symbol	Description	Units
α	Thermal diffusivity	m^2/s
A	Cross sectional flow area	m^2
β	Thermal expansion coefficient	1/K
Bo	buoyancy number	-
Cp	Specific heat capacity	J/kg-K
D_h	Hydraulic diameter	
ε	Emissivity	-
η	Property index	-
E	Energy	J
F	External forces	N
f	friction factor	-
g	Gravitational acceleration	m/s^2
Gr	Grashof number	-
h	Convection heat transfer coefficient	$\text{W}/\text{m}^2\text{-K}$
I	Identity matrix	-
$k_{\text{air}}/k_{\text{water}}$	conductivity	$\text{W}/\text{m-K}$
k-factor	Secondary loss factor	-
$\vec{v}$	Velocity vector	m/s
∇	Gradient	-
$\dot{m}$	Mass flow rate	kg/s
μ	Dynamic viscosity	$\text{kg}/\text{m-s}$
Nu	Nusselt number	-
ν	Kinematic viscosity	m^2/s
P	Pressure	Pa
Pr	Prandtl number	-
Ra	Rayleigh number	-
Re	Reynolds number	-
ρ	Density	kg/m^3
T	Temperature	K
τ	Shear stress	Pa
y+	Y plus	-
ω	Omega	-
z	elevation	m

Abbreviation	Description
CFD	Computational Fluid Dynamics
DCC	Depressurized Conduction Cooldown
GAMMA+	General Analyzer for Multi-component and

	Multi-dimensional Transient Application
GT-MHR	Gas Turbine Modular Helium Reactor
HPC	High-performance Computer
MHTGR	Modular High Temperature Gas-cooled Reactor
MWt	Mega Watt of thermal energy
PBMR	Pebble Bed Modular Reactor
RCCS	Reactor Cavity Cooling System
RPV	Reactor Pressure Vessel
SC-HTGR	Steam Cycle High Temperature Gas-cooled Reactor
SIMPLE	Semi Implicit Method for Pressure Linked Equations
UW	University of Wisconsin

CHAPTER 1 - INTRODUCTION

1.1 Introduction

The reactor cavity cooling system (RCCS) is one of the safety features designed into the fourth-generation nuclear reactors. This system is intended to remove decay heat from the reactor cavity. It operates completely passively and is designed to operate without external inputs. Designs exist for air-cooled and water-cooled RCCSs.

The accurate simulation of the phenomena in a reactor cavity cooling system is important for design calculations, to ensure reactor designs that are passively safe. The codes that are used to simulate the phenomena must therefore be verified and validated. One method to validate both a design and a simulation code is to build an experimental facility based on a prototype. In many cases scale models have been constructed of prototype designs. The facility constructed at the University of Wisconsin (Madison) is an example of such a scale model of an RCCS (Lisowski, 2013).

1.2 Background

1.2.1 Safety considerations for gas-cooled reactors

High temperature gas-cooled reactors (HTGRs) belong to the fourth generation of nuclear reactors. These reactors are intended for use both in power- and process heat generation. The PMR200 nuclear reactor, which produces 200 MWt, is intended to produce process heat for hydrogen production (Shin et al., 2017a). The GT MHR and MHTGR were designed for energy generation (Bechtel National INC et al., 1986; Neylan et al., 1994). In addition to the ability to produce high temperatures, gas-cooled reactors also have many safety features. These include: safe fuel design (fuel contained within the TRISO particles), negative temperature reactivity coefficients, low power density and passive heat removal systems (Neylan et al., 1994).

Depressurized loss of forced coolant accidents (DLOFC) is one of the postulated events that have been taken into account in the design of the safety systems in gas-cooled reactors (Lommers et al., 2014). In the event of a DLOFC, the heat transfer in the core occurs mostly by means of conduction. The acceptable limit of the temperature within the core is widely assumed

to be 1600 °C (Lommers et al., 2014). The design basis for cooling and safety systems is therefore based around this assumed temperature.

In the event of a DLOFC the heat is conducted away from the fuel into the other components in the core such as the graphite, the core barrel and the pressure vessel. The heat is then transferred into the reactor cavity by thermal radiation and natural convection from the reactor pressure vessel. The heat is finally absorbed by the reactor cavity cooling system (RCCS) which acts as an ultimate heat sink (Lommers et al., 2014, Corradini, 2012).

There are challenges in designing functional and practical RCCSs, according to IAEA (2000); the system's geometry is large, as necessitated by the geometry of the reactor cavity, which makes it difficult to inspect, clean or alter after installation. Another design consideration according to IAEA (2000) is that the heat removal must be sufficient under accident conditions, but not so large as to cause unnecessary heat losses during normal operating- or planned shutdown conditions.

In summary, the role of the RCCS is to remove the decay heat from the reactor cavity in a passive manner. It should be designed as simple as possible to provide the least probability of failure. The system must not be over designed in terms of heat removing capacity since it is undesirable under normal operating conditions.

1.2.2 Description of the RCCS

The RCCS can have the following basic components: cold downcomer, hot risers and heat rejection zone. The heat rejection can occur to the atmosphere, or by means of a heat exchanger. Depending on whether the system is active or passive, a feed pump or fan may be used. The focus of this work however is on the passive systems, which rely solely on natural circulation.

The coolant (typically water or air) is fed to the downcomer, either from a tank or from the atmosphere (for water and air respectively). The downcomer transports the coolant to the hot risers, which are installed on the circumference of the reactor cavity. The number of risers installed varies for the different designs, with typical numbers around 220 risers. The risers are heated by the thermal radiation and convection heat transfer from the reactor pressure vessel. The risers in turn heat up the coolant, inducing the buoyancy forces that drive the system. The cooler air in the downcomer, having a higher density, also pushes down on the hotter, less dense coolant, forcing it upwards. The heated coolant moves upward, out of the risers, where after the heat is rejected to the atmosphere, or to a heat exchanger and the cycle is repeated.

Figure 1.2-1 by Shin et al. (2017a) shows a conceptual depiction of an RCCS. The figure depicts the heat transfer from the reactor pressure vessel (RPV), which leads to the development of the natural circulation within the ducts of the RCCS.

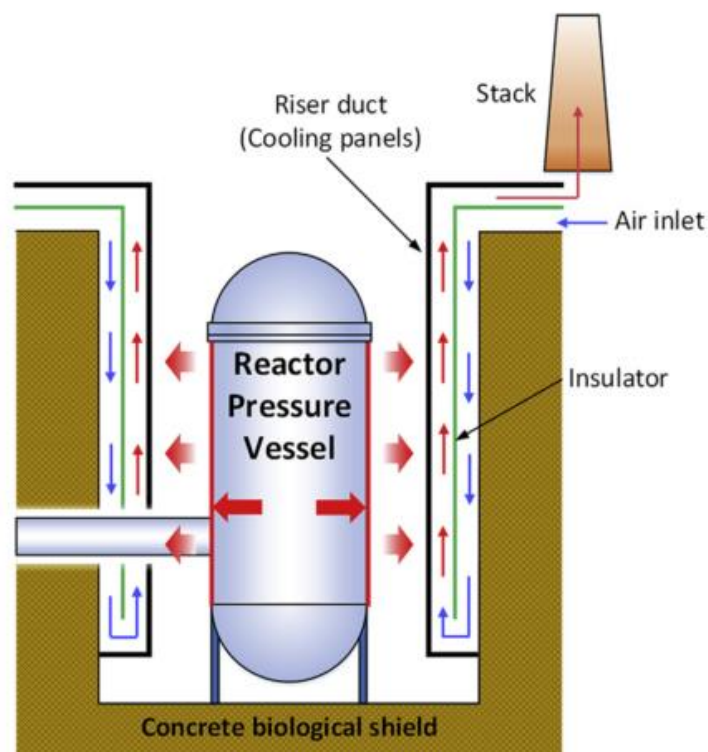


Figure 1.2-1. Conceptual depiction of the RCCS. (Shin et al., 2017a).

1.2.3 Reactor designs employing the RCCS

1.2.3.1 Primary air-cooling (GT-MHR)

General Atomics designed the Gas Turbine Modular Helium Reactor (GT-MHR). It produces 550 MWt through a direct Brayton cycle (Neylan et al., 1994). The core is of the prismatic block type (as opposed to the pebble bed type) and is helium cooled (Neylan et al., 1994). The RCCS of this design is the air-cooled configuration (Neylan et al., 1994).

Figure 1.2-2 shows the air-cooled configuration of the RCCS as implemented in the GT-MHR design. As illustrated in Figure 1.2-2, the air is drawn from the atmosphere and transported via the intake duct to the cooling panels. The heated air rises to exit again at the top of the intake/exhaust structure.

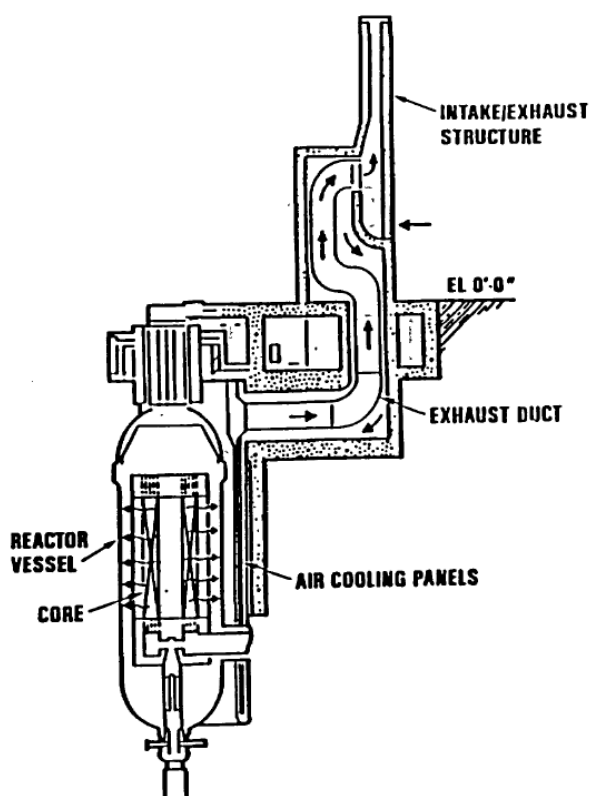


Figure 1.2-2. Schematic of the layout of the RCCS in the GT-MHR. (Neylan et al., 1994).

The arrows in Figure 1.2-2 indicate the direction of the flow, with the cold air being drawn downwards into the system, while the heated air flows upward. The system's operation is as follows: cold air enters through the intake/exhaust structure. The air flows downward under

gravity via the cold downcomer into the hot risers, which are installed all around the circumference of the reactor cavity. While in the hot risers, the air is heated, which induces the temperature gradient that is necessary for the natural circulation to occur. The hot air is collected from the risers into the hot plenum, which is installed at the top of the reactor cavity. The hot plenum guides the air into the intake/exhaust stack which guides the airflow out of the cavity where it is vented to the atmosphere (Neylan et al., 1994).

1.2.3.2 Primary water cooling, Secondary/tertiary air cooling (HTR-10).

The design concept of the HTR-10 reactor is shown in Figure 1.2-3. The primary coolant is water, which is cooled by an air-cooled condenser, which is cooled by air by natural circulation within a cooling tower (Zhao et al., 2019). The cooling system is sub-divided into two independent units.

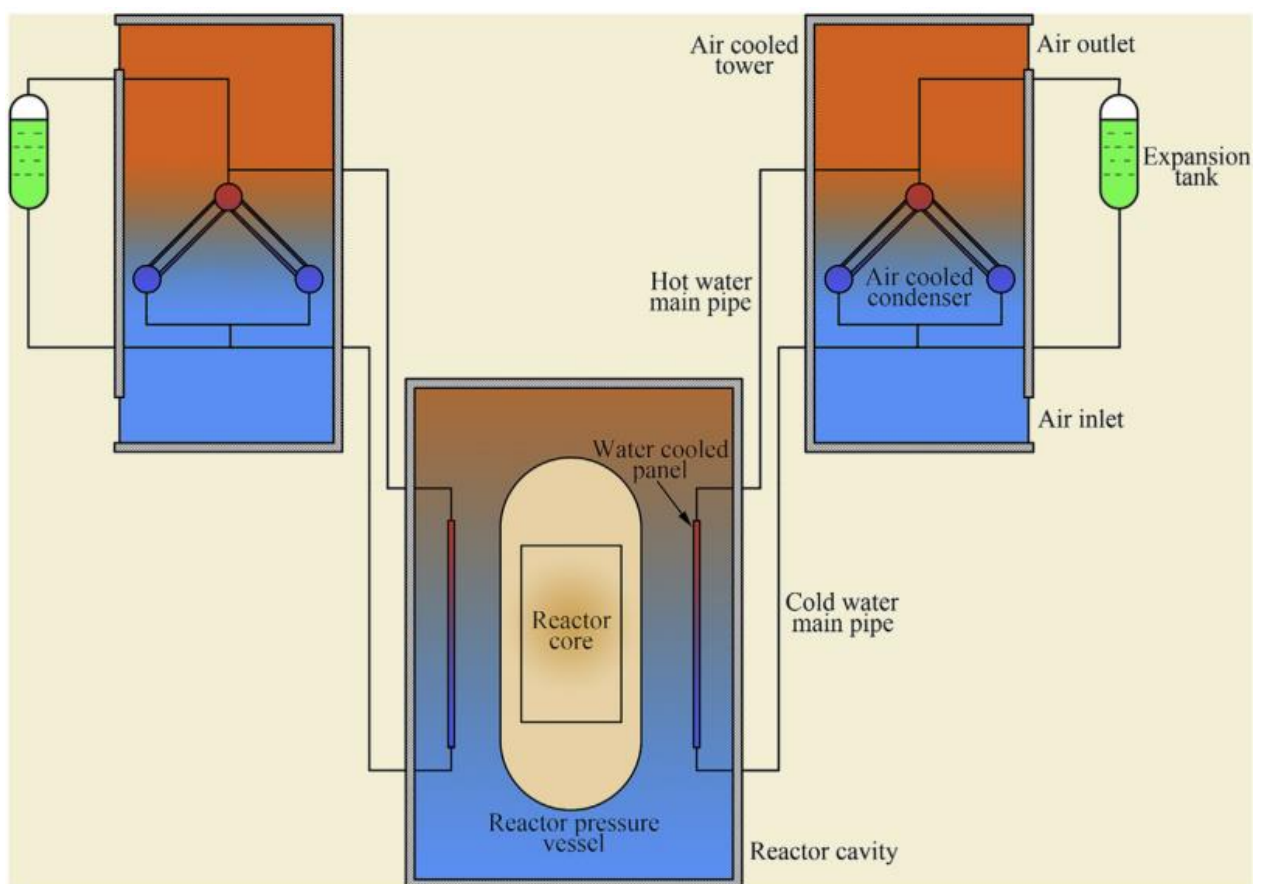


Figure 1.2-3. PRHRS of the HTR10 design. (Zhao et al., 2019).

1.2.3.3 Primary water cooling, with forced secondary air cooling (SC-HTGR)

Figure 1.2-4 by Lommers et al. (2014) shows the water-cooled configuration of the RCCS as implemented in the SC-HTGR design developed by AREVA in collaboration with NGNP industry alliance (Lommers et al., 2014). The SC-HTGR design is a helium cooled direct cycle, prismatic block reactor that would produce 625 MWt (Lommers et al., 2014).

The primary RCCS loop is water-cooled and additionally contains a secondary air-cooled loop. The addition of the water storage tank is necessary in this configuration. The heat rejection to the secondary loop occurs via the water storage tank. Forced flow is employed in the air-cooled loop to enhance the heat removal (Lommers et al., 2014).

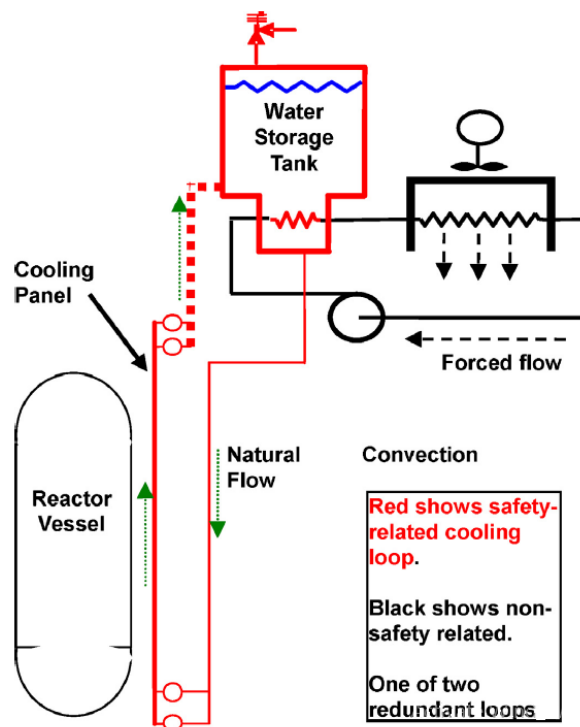


Figure 1.2-4. Decay heat removal system.
(Lommers et al., 2014).

1.2.3.4 Primary water cooling (PBMR, VGM)

The PBMR is a helium cooled direct cycle design, which is designed to produce 400 MWt.

In the PBMR, the RCCS was designed to run in parallel with other cooling systems, namely the core conditioning system and the core barrel conditioning system. These three systems are connected to the same heat exchangers (Slabber et al., 2006).

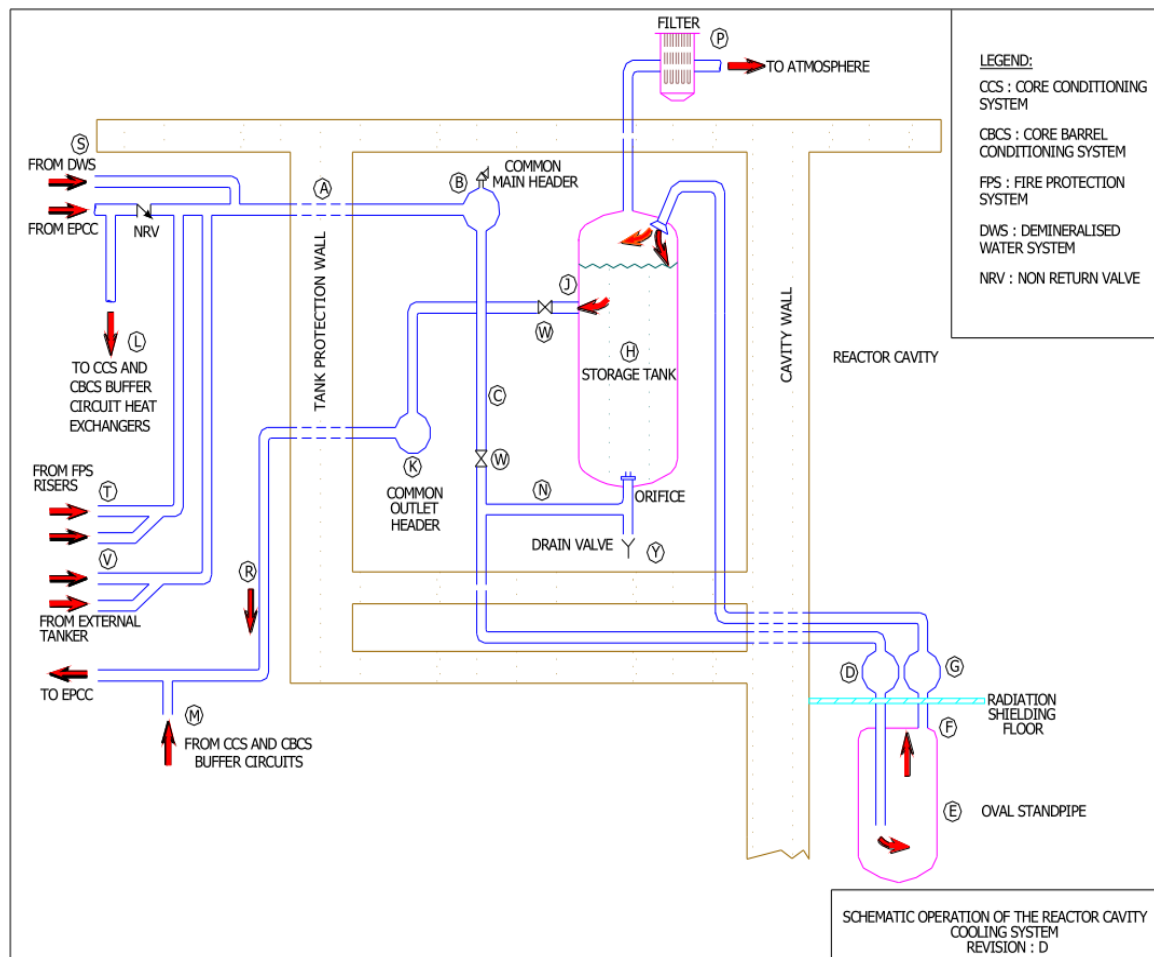


Figure 1.2-5. Schematic representation of three of the cooling systems in the PBMR (Slabber et al., 2006).

The intended working principal according to Slabber et al. (2006) is as follows: the heat in the cavity is transferred to the oval standpipe (E in Figure 1.2-5) which causes a density gradient in the water within the standpipe. The water rises as a result of the buoyancy force and travels upward to the header tank G and eventually ends up in the storage tank H. During passive operation, steam can escape out of the tank to the atmosphere. The filter, P, inhibits the ingress of contaminants from the outside from entering the system. The hot water exits the tank at J, travels through the common outlet header K and travels via the pipe network R to the heat exchangers. During normal operation, cold water is supplied from the common main header, B

to the intermediate header D, which in turn supplies coolant to the RCCS standpipes. During upset conditions, water will pass through an orifice and via pipe N to feed the intermediate header at D. During normal operating conditions the flow through pipe N is negligible (Slabber et al., 2006).

The VGM is a 200 MWt reactor design intended for process heat generation (IAEA, 2000). The RCCS of the VGM (shown in Figure 1.2-6) consists of three independent tube stacks installed in the reactor cavity. Water would be used as the working fluid (IAEA, 2000). The risers installed in the cavity are divided into three independent units, each having separate inlet and outlet manifolds (IAEA, 2000). This design feature could improve safety, since three systems would have to fail simultaneously before the cooling would be lost.

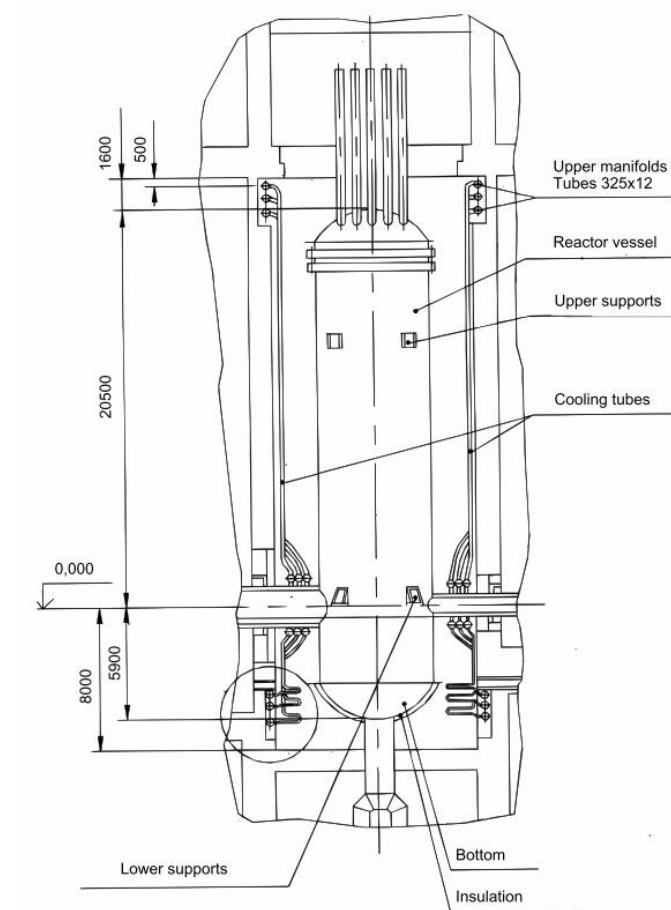


Figure 1.2-6. VGM reactor (IAEA, 2000).

1.2.3.5 Primary air cooling, with optional primary water cooling

Lim et al. (2018) proposed a design that makes use of a water jacket that is installed within the downcomer and is fed from a storage tank. The system (which is shown in Figure 1.2-7) is primarily air-cooled with air flowing over the outer surface of the water jacket. Under upset conditions where air-cooling is insufficient or inhibited in some way, the water jacket would burst due to the subsequent boiling of the water. The energy would then be dissipated by the boiling of the water (Lim et al., 2018).

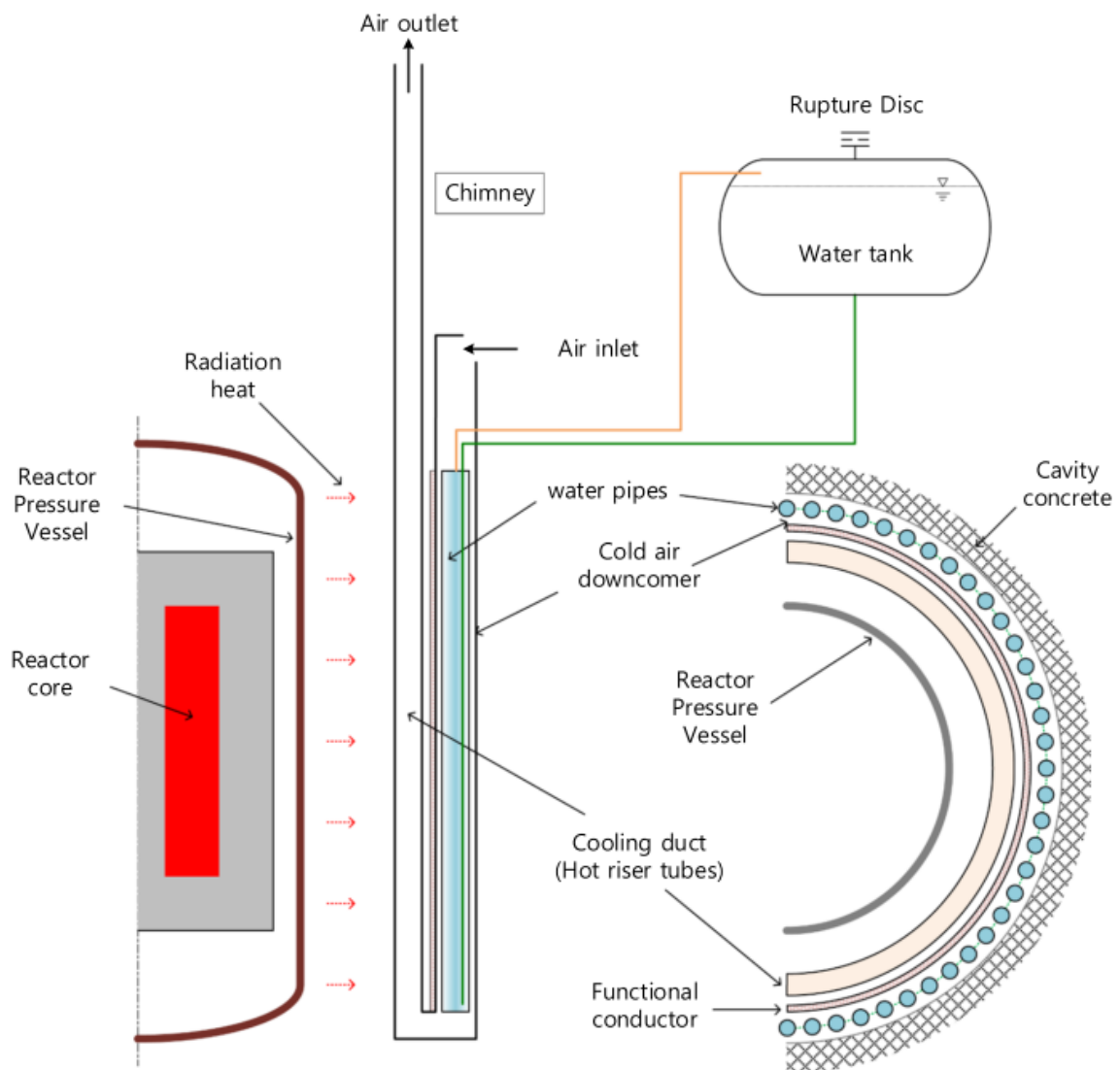


Figure 1.2-7. A RCCS design that makes use of water and air proposed by Lim et al. (2018).

1.2.3.6 Primary air cooling, secondary water cooling (PMR200)

The PMR200 is a VHTR under development in South Korea by the Korea Atomic Energy Research Institute (KAERI). The design produces 200 MWt and is intended to provide process heat for hydrogen production (Shin et al., 2017a). One of the possible layouts of the RCCS of the PMR200 is shown in Figure 1.2-8. The system consists of a water and air loop independently removing heat from the reactor cavity. The water loop was added since the water has better heat transfer capabilities than air (Bae et al., 2012). The air loop operates passively, while the water loop could make use of a pump.

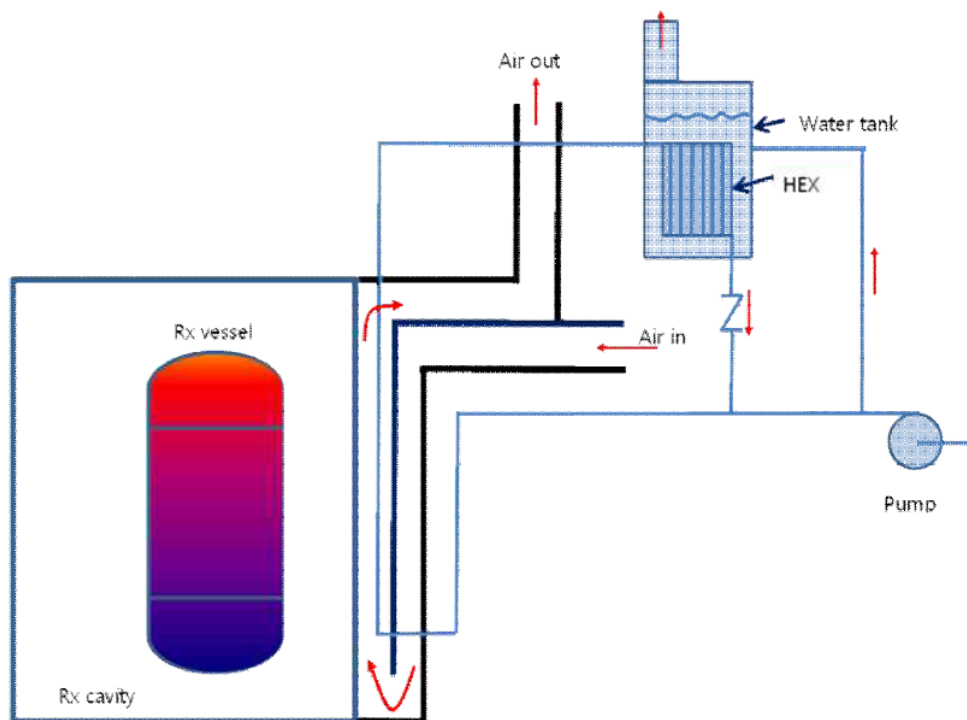


Figure 1.2-8. Possible RCCS layout of the PMR200. (Bae et al., 2012).

1.2.3.7 Simultaneous air- and water cooling by using a water pool

Another variation of the air-cooled RCCS configuration was suggested by Oh et al. (2009). In this configuration the air-cooled risers are submerged within a water pool, which is installed on the periphery of the reactor cavity. The system attempts to enhance the capabilities of the air-cooled configuration with the addition of the water. Water has a better heat transfer capability

than air, however the designs using a water cooled system are of greater complexity (Oh et al., 2009).

1.2.4 Scaling of experimental facilities

The performance and behaviour of the RCCS designs can be evaluated based on scaled down versions of the designs. The full scale designs are not typically constructed for financial reasons and due to the amount of space the full scale models would require (Bae et al., 2012). Scaling analysis studies are necessary to ensure that the important phenomena would be retained and not distorted by the scaling.

Bae et al. (2012) conducted a scaling analysis on the NACEF, as did Tzanos and Farmer (2006) for the NSTF. Lisowski (2013) also conducted a scaling analysis on the RCCS facility constructed at the University of Wisconsin (UW). The facility is described later in this work.

In the scaling analysis by Bae et al. (2012) it was decided to retain the horizontal scale of the design and to scale down the vertical height. The ratio between the non-dimensional numbers of the prototype and scale model had to be as close as possible to 1. The cavity radiation number and the temperature ratio were regarded as paramount, since radiation heat transfer would dominate in the cavity. The natural circulation / mixed convection regimes pose a challenge in terms of the scaling as they require complex treatment in calculating the correct heat transfer coefficients (Bae et al., 2012). Care should therefore be taken in the scaling of prototypes into experimental facilities. The type of flow that would be present in the system should also be accounted for, since the convection heat transfer may be over- or underestimated based on the regime (Bae et al., 2012).

Similarly to Bae et al. (2012), Tzanos and Farmer (2006) also used the cavity radiation number and temperature ratio similarity. In addition, the cavity convection ratio was also enforced.

1.2.5 Facility overview

There are several facilities that have been constructed to investigate the RCCS of gas-cooled reactors. Three of these facilities are of importance and will be discussed, namely the NACEF, NSTF and the RCCS facility constructed at the University of Wisconsin (Madison).

The Natural Cooling Experimental Facility (NACEF) is an experimental facility designed to represent the RCCS of the PMR200 designed in South Korea (Kim et al., 2014). The facility was scaled $\frac{1}{4}$ vertically and 1:1 laterally, yielding a structure that stands 14 m tall, with a heated cavity of 4 m in height (Kim et al., 2014). The air inlet is located inside the laboratory building, while the two outlet chimneys are located on the outside. Air enters a single inlet pipe and is then split by a manifold into 6 rectangular riser tubes for the height of the heated cavity (Kim et al., 2014). After being heated the air is combined in an outlet manifold into two chimneys, which vent the heated air into the atmosphere (Kim et al., 2014). The heated cavity emulates the reactor pressure vessel by means of heaters delivering 52 kW that are installed on one of the walls (Kim et al., 2014). The air flow rates and temperatures could be monitored at various locations in the facility, making an in-depth analysis possible.

Natural convection Shutdown heat removal Test Facility (NSTF) was constructed at Argonne National Laboratory to study the phenomena in the RCCS of a gas-cooled reactor (Lomperski et al., 2011). The facility represents a $\frac{1}{2}$ scale model of the prototype design (Lisowski et al., 2016). Similar to the NACEF, the NSTF also consists of a single inlet at the bottom elevation, a heated section and two chimneys at the outlet (Lisowski et al., 2016). In the heated section the air is diverted into 12 rectangular riser tubes, that represent a 19.03° radial slice of the prototype design (Lisowski et al., 2016). The various measurements taken during experiments include mass flow rates, surface and fluid temperatures and differential pressure drop (Lisowski et al., 2016). The measurements can be analysed to gain insight into the thermal hydraulic phenomena that would also occur in the prototype.

The University of Wisconsin (UW) (Madison) also constructed a scaled down version of a prototype RCCS (Lisowski, 2013). Unlike the NACEF and NSTF the facility constructed by UW represented the water-cooled RCCS design. The facility represents a $\frac{1}{4}$ scale model of an RCCS to be used in the MHTGR (Lisowski, 2013). Three riser tubes are installed in the heated cavity, which represents a 5° radial slice of the prototype cavity. In addition, the riser tubes have fins attached, separating the cavity into two isolated air-filled regions (Lisowski, 2013). Heaters are installed on one of the walls of the heated cavity, with a maximum power level of 42.5 kW (Lisowski, 2013). Measurements were taken at various positions in the facility by thermocouples that measured water and surface temperatures while volume flow meters measured the rate of flow of the water (Lisowski, 2013).

1.2.6 Summary

The RCCS is a system that enhances the safety of gas-cooled reactors such as the GT-MHR, SC-HTGR, PBMR and PMR200. The cooling can consist of a single loop or two interconnected loops using air, water or a combination of the two. The designs using the air-cooled configuration seem to be simpler but lack the heat transfer ability of the water-cooled configuration. Scaled down experimental facilities of the RCCS can be invaluable for investigating the performance and behaviour of the prototype designs. The integrity of the experimental results however is reliant on proper scaling of the facility by a scaling analysis study. Examples have been given of three of the experimental facilities that have been constructed for the analysis of system performance, namely the NACEF, NSTF and UW RCCS facility.

1.3 Problem statement and aim

The reactor cavity cooling system (RCCS) experimental facility constructed at the University of Wisconsin (Madison) (UW) has been simulated with simplifying assumptions by Lisowski (2013). The work done by Lisowski (2013) focussed primarily on experimental work, including single and two-phase flow at various levels of heater power. The numerical work done on the facility were simulations done by using RELAP and an independent separate 3D simulation in which the riser tubes and manifolds were simulated without the heater box. In the simulations done by Lisowski (2013) the heat flux boundary condition was specified on the surface of the riser/fin assembly. Although the overall result would be accurate with this method, it excluded the front and back cavities from the analysis.

The aim of this study is to expand on the simulations done by Lisowski (2013) on the UW experimental facility. The heated section will be simulated by including the front cavity, fins and riser assembly and back cavity by using 3D CFD codes such as Star-CCM+ 14.02.010 (Siemens PLM software, 2019) and Ansys Fluent 18.2 (Ansys, 2021b). In addition, 1D systems CFD methodology will be incorporated and coupled with the 3D CFD methodology.

Another contribution the study aims to make is to investigate the phenomena in the back cavity, since similar studies have overlooked this part of the system. Qualitative flow patterns will be investigated in the front and back cavities. Convection heat transfer coefficients will be calculated to quantify the convection heat transfer from each surface in the front and back cavities.

1.4 Scope

The scope of this study is the simulation of the UW experimental facility by using 3D CFD and 1D system CFD methodology. The geometry and published experimental data will be used to validate the solutions obtained by the CFD methodologies. Assumptions will be made where necessary and will be explicitly stated. The study will therefore not be a precise direct comparison of the facility but will nonetheless serve as a reference of the performance of similar systems. The study will focus on the single-phase steady state simulations, to evaluate the normal operating conditions of an RCCS. Natural circulation systems are subject to instabilities and a stability analysis could be done but is considered outside the scope of this study.

1.5 Layout of work

The work will be subdivided into chapters as follows: firstly, a literature survey (Chapter 2) will be done on previous studies relating to the reactor cavity cooling system, including scale models and simulated systems. A chapter will be devoted to describing the theoretical background used by the CFD codes to simulate the system. The 3D methodology will be verified by comparing the results from two different CFD codes. These results will be dependent on the mesh and is subject to a mesh dependence study, which will also be done. After the methodology has been verified, a study will be done where the system's performance will be investigated with several variations in the geometry. The variations will include changes in the shape of the riser tubes or including or omitting the fins in the design. A chapter will also be devoted to the coupling methodology, where the models of both 1D and 3D codes will be discussed, as well as the results obtained by this methodology. In closing the principal conclusions of the study will be listed and discussed.

CHAPTER 2 - LITERATURE SURVEY

2.1 Introduction

This chapter is intended to provide an in-depth overview of the reactor cavity cooling system (RCCS) in literature as found in high temperature gas-cooled reactors. Works of both an experimental and numerical nature are included with applications ranging from proposed designs to prototype models. Behaviours that are common in these systems are highlighted, such as the tendency for flow reversals to occur. The methodologies in simulating the RCCS can vary from full core/RCCS to RCCS only to radial slices of the system. Some of these works will be discussed.

2.2 Instabilities relating to the RCCS

Although two-phase flow was regarded as beyond the scope of this work, the instabilities that often occur in natural circulation loops should be broadly discussed

Instabilities in a nuclear reactor can be divided into three main causes, namely, thermal hydraulic instabilities, control system instabilities and coupled thermal hydraulic/neutronic instabilities (Prasad et al., 2007). The thermal hydraulic instabilities are of importance for this study.

According to Prasad et al. (2007) thermal hydraulic instabilities can be categorized within the categories static or dynamic. Static instabilities can be subdivided based on their behaviour into the categories of flow pattern transition, Ledinegg instability, geysering and boiling crisis. The dynamic instabilities can be caused by pressure drop oscillations as well as density wave oscillations, the latter of which is categorized further into a Type 1 and Type 2 instability. Type 1 instabilities occur at conditions where the sub-cooling at the inlet is high, while Type 2 occurs with low inlet sub-cooling.

Kim et al. (2014) conducted experiments on the natural cooling experimental facility (NACEF). The facility represents a $\frac{1}{4}$ scale model of the RCCS of the PMR200 gas cooled reactor design. The facility consists of an inlet plenum, 6 riser ducts that pass vertically through a heated cavity, followed by an outlet plenum and two chimneys. The preliminary tests provided insight into the

system's sensitivity to flow reversals. The flow direction is intended to be upward in both chimneys, but it was observed that one of the chimneys became a secondary inlet.

Experiments conducted by Lisowski et al. (2016) on the natural convection shutdown heat removal test facility (NSTF) also proved the facility's sensitivity to meteorological conditions. The NSTF is similar in design to the NACEF. It is a $\frac{1}{2}$ scale model of the air-cooled RCCS and contains 12 riser ducts in the heated test section. The authors note that the flow reversal problem was mitigated by increasing the flow resistance by means of a valve.

Numerical simulations done by Du Toit et al. (2014) considered an air-cooled RCCS and compared results between the codes GAMMA+ and Flownex. A temperature was specified as boundary condition on the surface representing the reactor pressure vessel. Temperatures on select surfaces in the cavity and at the chimney outlet were considered. A transient analysis was also done by using Flownex, in which the numerical model's sensitivity to flow reversals was tested. The pressure at the outlet was increased for a period of time, where after it would return to the original pressure. During the period, the pressure at the outlet would be higher than at the inlet. The numerical solution showed a flow reversal after the pressure increase. After the pressure at the outlet was lowered again, the flow remained in the reverse direction. (Du Toit et al., 2014). The temperatures calculated to occur on the concrete wall could prove detrimental under these conditions (Du Toit et al., 2014).

Pal et al. (2016) conducted physical and numerical experiments on a test facility based on the moderator cooling system intended for use in an advanced heavy water reactor (Pal et al., 2016). The cooling system is designed to circulate water over the moderator, thereby removing excess heat and prolonging the lifespan of the moderator (Pal et al., 2016). Water is fed to a calandria in the core, where the water is heated up. Buoyancy forces are subsequently induced, which forces the water upward, out of the core. The water travels through a pipe system, transporting the water to a heat exchanger where the excess heat is rejected, where after it is returned to the calandria to repeat the process (Pal et al., 2016). The facility representing the system is shown in Figure 2.2-1. A transient simulation was run, in which the heat input on the calandria tubes was gradually increased (Pal et al., 2016). Initially the water recirculated within the calandria and did not flow through the rest of the pipe network of the cooling system (Pal et al., 2016). As the time progressed the buoyancy forces forced the

heated water further up into the two calandria outlet chimneys, where further recirculation occurred (Pal et al., 2016). Where these two chimneys converged again, the stronger of the two streams overpowered the other to induce a recirculation within the chimney system. Eventually the buoyancy forces were strong enough to force the fluid upward in both chimneys completing the natural circulation loop (Pal et al., 2016).

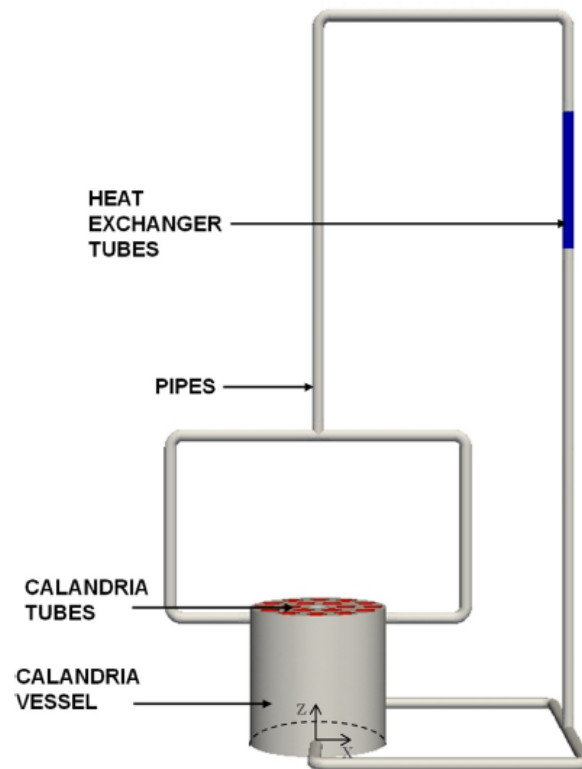


Figure 2.2-1. Experimental facility layout (Pal et al., 2016).

In summary, flow reversals seem to be a common occurrence in free convection systems. If the pressure at the outlet of the system rises higher than the internal pressure, flow reversal can occur in the RCCS. Partial reversals have been found in the experiments by Kim et al. (2014) and Lisowski et al. (2016). A full flow reversal was numerically simulated by Du Toit et al. (2014). The reversals have been found while simulating air-cooled and water-cooled systems. The buoyancy forces are directly dependent on the heat input, meaning sufficient addition of heat is needed by the cooling fluid to overcome system resistance and atmospheric conditions. The physical phenomena are sensitive, and it is therefore important that the numerical simulation codes can accurately account for the sensitivity.

2.3 Simulations of accident scenarios

A study by the IAEA (2000) done on the VGM reactor design simulated pressurised and depressurised cooldown scenarios. The RCCS was given a fixed inlet temperature and mass flow rate as boundary conditions. The heat transferred to the RCCS was mainly via the mechanism of radiation. The results from four different formulas for the Nusselt number was compared as a measure of convection heat transfer within the cavity.

Lommers et al. (2014) simulated a depressurized conduction cooldown scenario in an SC-HTGR, to determine whether the temperatures would remain below the acceptable limits. The DCC scenario assumed negligible convection in the reactor and the RCCS standpipes were given a constant temperature of 65 °C. The initial power was set to the expected full capacity power as a boundary condition. The simulated cool down of the system from full power was observed. The simulated temperature of the reactor pressure vessel was monitored as a function of time. The peak temperature was calculated to occur after about 100 hours (4 days) (Lommers et al., 2014). Further estimations were also done to obtain the possible peak temperatures under conservative assumptions. The maximum temperature obtained for the reactor pressure vessel under these assumptions is 455 °C. The maximum allowable temperature was considered as 538 °C (Lommers et al., 2014).

Haque et al. (2006) investigated the resultant temperatures during pressurized and depressurized conduction cooldown scenarios. A constant temperature of 65 °C was applied to the reactor cavity cooling system's outer shell as a boundary condition. The temperatures in the central graphite, core barrel, side reflector and pressure vessel were monitored as a function of time. In the pressurized case the density of the gas in the reactor was higher, which improved the natural circulation within the core. The monitored temperatures were therefore lower than in the corresponding depressurized case (Haque et al., 2006). The fuel was below the temperature limit of 1600 °C and the pressure vessel temperature didn't exceed 400 °C (Haque et al., 2006). The natural circulation within the core was negligible under the depressurized conditions, which lead to higher temperatures in the core than in the pressurized condition. The peak fuel temperature was 1587 °C and the peak reactor pressure vessel temperature was 476 °C in the depressurized case (Haque et al., 2006).

Lim et al. (2018) simulated a water-jacket concept RCCS design, in which an air-cooled RCCS is integrated with a water jacket that bursts under upset conditions (Lim et al., 2018). The design was simulated by using the code GAMMA+. The model consisted of $\frac{1}{2}$ radial sector of the geometry and included the components from the reactor core, reactor cavity, the RCCS and the surrounding concrete and soil. A depressurized conduction cooldown (DCC) event was simulated for a 350 MWt helium cooled reactor and the reactor pressure vessel (RPV), fuel and concrete temperatures were examined. The water-jacket design was compared to a baseline design that used only air cooling. The water capacity of the system was also tested for an inventory of 200, 300 and 400 tons of water (Lim et al., 2018). The water-jacket design showed improved cooling of the fuel, RPV and concrete above the baseline design. Before the water tank had been depleted the concrete was below the acceptable temperature, while the RPV and fuel exceeded the acceptable temperatures only marginally (Lim et al., 2018). After the water tank had depleted the fuel temperature kept decreasing while the RPV and concrete temperatures started to increase (Lim et al., 2018). The authors recommended that a water tank capacity of at least 300 tons should be used with this design (Lim et al., 2018).

In summary, full core/RCCS simulation models can be seen as a valuable tool in predicting the behaviour of the RCCS/reactor core under accident conditions. These models are used to predict the temperatures at various points in the system, such as the concrete wall of the enclosure (Lim et al., 2018) and the reactor pressure vessel wall (Haque et al., 2006; Lommers et al., 2014). Another observation that was made apparent in the literature is that heat transfer in the cavity occurs primarily by radiation.

2.4 Experimental and numerical models

Since the testing on full scale prototype models would be costly, other methods have been presented in literature, namely scaled down facilities and CFD models. The two methods are often combined to validate the CFD models.

Oh et al. (2009) compared computational and experimental results of an experimental facility which emulates the reactor cavity cooling system. The RCCS facility design entails a combination of water and air cooling; a forced circulation air cooling loop is submerged in a

water pool that is installed in the reactor cavity. The experimental facility is scale 1/10 of the prototype. In the experimental facility a heater is installed in the reactor vessel to emulate the reactor core. Heat is transferred to the water pool by radiation and natural circulation in the cavity. The water in the water pool can escape to the atmosphere through a valve, which is programmed to open when the pressure reaches 1.5 bar.

The code CFX (Ansys, 2021a) was used to simulate the phenomena in the facility. The convection heat transfer coefficient of the cooling air was calculated by using the temperatures measured during experiments. The applicable experimental results were used as boundary conditions to calculate the fluid velocity and temperature profiles in the coolant pipes. The authors concluded that validation of the code CFX was achieved, since the results correlated well with experimental results. The results were also compared with results from literature in which the code Fluent (Ansys, 2021b) was used. The trends exhibited by the results obtained by CFX and Fluent were the same.

Oh et al. (2009) also compared the temperatures from measurements with temperatures calculated using the code RELAP5-3D. A temperature of 207 °C was applied to the reactor vessel outer surface and a temperature of 28 °C was applied to the cavity's concrete outer wall. The calculated and measured temperatures along the vertical height of the cavity were compared. The maximum temperatures were found to be slightly above the middle of the cavity, with the lowest temperatures at the bottom of the cavity. The vessel support structures resulted in heat loss, which resulted in some differences in the measured and calculated temperatures.

2.5 Methods used for CFD modelling

Ampofo and Karayiannis (2003) investigated natural circulation within an air cavity. One of the walls was heated while the opposing wall was cooled. Precise measurements were taken to aid in the validation of turbulence models used by CFD codes. Special attention was given to the important measurements in close proximity to the hot wall, such as the velocity and temperature measurements and the derived turbulence parameters such as the turbulent kinetic energy and turbulent heat flux (Ampofo & Karayiannis, 2003). Results from a literature source were

compared with the experimental results. The study from literature was a CFD study that used the k - ϵ and LES turbulence models (Ampofo & Karayiannis, 2003). From the results obtained by Ampofo and Karayiannis (2003) the LES model outperformed the k - ϵ model, however both models gave less accurate results pertaining to the turbulence quantities (Ampofo & Karayiannis, 2003). The experimental and CFD temperatures and velocity profiles compared reasonably well for both turbulence models (Ampofo & Karayiannis, 2003).

Keshmiri (2014) tested three different CFD codes including the formulation of the k - ϵ and SST k - ω as implemented in Star-CD (CD-adapco, 2006). The turbulence models predicted the velocity and temperature distributions reasonably well but were less accurate in predicting the Reynolds stresses and turbulent kinetic energy distributions (Keshmiri, 2014). The Nusselt numbers calculated by the two turbulence models were larger than the experimental results within the mixed convection regime, which would over predict the heat transfer (Keshmiri, 2014).

Ji (2014) considered natural circulation within a square air cavity and compared the results from different meshes and turbulence models. Both 2D and 3D models were investigated with the k - ϵ , k - ω , RNG and SST turbulence models. Boundary conditions were applied on opposing walls to obtain a hot and a cold side. The temperature, velocity, kinematic energy and the scaled turbulence viscosity (calculated by dividing turbulent viscosity by the molecular dynamic viscosity) distributions were plotted and compared for the different turbulence models (Ji, 2014). The k - ω model performed the best overall in predicting the phenomena as evidenced by the correlation with the compared experimental results.

A study by Pal et al. (2016) considered natural circulation flow that would occur within the core of a heavy water reactor. An experimental test bench was constructed and used for comparison with CFD calculation results. The SIMPLE algorithm was used for steady state simulations and PISO scheme for transient cases (Pal et al., 2016). Convergence was assumed when the average of the scaled residuals reached values below $1e-4$ (Pal et al., 2016). The SST k - ω turbulence model was used and temperature contours and velocity vectors were examined to analyse the phenomena within the test bench (Pal et al., 2016).

Rajamohan et al. (2011) conducted experiments and numerical simulations on a test section that investigated effects of surface emissivity and surface roughness in the mixed convection regime. A square and rectangular cross-sectional cavity was tested in which two opposing walls were kept at constant temperatures. Thus, there was a hot wall, a cold wall and two adiabatic walls. A blower was installed to drive airflow through the cavity. The Nusselt number was calculated as a measure of the heat transfer that was occurring. It was found that the square cross-sectional area variation had a larger Nusselt number than the rectangular variation. The Nusselt number also showed a strong dependency to the surface roughness, since a polished surface had a higher Nusselt number than a surface with a high emissivity (Rajamohan et al., 2011).

Lomperski et al. (2011) used the CFD code Star-CCM+ to simulate the natural cooling shutdown heat removal test facility (NSTF) which is an experimental facility based on the RCCS of the GT-MHR. The SIMPLE algorithm was used with the realizable k- ϵ turbulence model with the 2 layer all y+ wall treatment enabled for the near wall region. The simulation was considered as converged when all residual criteria were below 1e-4. A few observations were made on the performance of the facility based on the velocity and temperature distributions. The temperature and velocity profiles in the 12 riser tubes in the facility were non-uniform owing to the asymmetrical inlet. The temperature distribution between the riser tubes becomes more uniform towards the outlet plenum (Lomperski et al., 2011).

A study by Oh et al. (2009) investigated the flow patterns and temperature distribution that would occur in a water-pool type air cooled RCCS facility. The facility represented a $\frac{1}{4}$ scale model of the geometry in the reactor cavity of the MHTGR. Air ducts were installed helically in the cavity submerged within a pool filled with water, on the periphery of the cavity. The code CFX was used to gain detailed insight into the velocity and temperature profiles that would develop within the air ducts (Oh et al., 2009).

Frisani and Hassan (2014) conducted a CFD study on the RCCS of a VHTR. Radial slices of the reactor cavity were simulated, investigating the effect of the inclusion on various numbers and pitch of riser tubes in the slice (Frisani & Hassan, 2014). A heat flux was applied on the RPV surface as a boundary condition. In some cases, a fixed temperature was specified on the riser tubes' surfaces as a boundary condition while in two cases the inlet velocity in the riser

tubes was specified (Frisani & Hassan, 2014). The realizable k- ϵ turbulence model was used with all y+ wall treatment (Frisani & Hassan, 2014).

Zhao et al. (2017) simulated the passive heat removal system of the HTR-10. The finned riser tubes were simulated in conjunction with the reactor cavity. The simulations entailed a study on the influence of the size of the radial slice of the cavity. A fixed temperature was specified on the RPV surface as a boundary condition. Slices containing $\frac{1}{2}$ tube/fin, 1 tube/fin, 2 tube/fin and 3 tube/fin assemblies were tested (Zhao et al., 2017). The subsequent temperature profiles were examined. Based on the comparison of the results, it was concluded that accurate results require at least two or more riser/fin assemblies to be simulated (Zhao et al., 2017).

In summary, the simulations pertaining to natural circulation make use of various turbulence models, such as the k- ϵ , k- ω and LES models. The consensus on the convergence criteria seems to be that the residual criteria have to be below 1e-4. The velocity and temperature profiles are regarded by many of the studies as the primary results. Some authors also paid close attention to non-dimensional parameters such as the Nusselt number and turbulence parameters such as the turbulent kinetic energy to give insights into the phenomena. The boundary conditions on the RPV surface could either be a fixed temperature or a heat flux.

2.6 1D and 1D/3D coupled methods

Zhao et al. (2019) used a 1D method to simulate a part of the RCCS (or passive residual heat removal system - PRHRS) of the HTR-10. The part of the system that passes through the cavity, was simulated by using 3D CFD by Zhao et al. (2019), since the detail could be calculated. However, the pipe network before and after the cavity doesn't need to be simulated with great detail. The pipe network before and after the cavity was therefore simulated in 1D, since the detailed results are not required as is the case within the cavity (Zhao et al., 2019).

Kööp (2018) coupled the systems thermal hydraulic code RELAP with the 3D CFD code Star-CCM+ in the simulation of the TALL test facility. RELAP was used to solve the 1D domain of the facility and Star-CCM+ was used to solve the 3D domain. The temperature rise across the 1D and 3D domains (which should be equal) were compared where after the heat added in the

1D domain would be adjusted accordingly. The relevant boundary conditions between the domains were exchanged between the codes until convergence had been achieved. The coupled codes were validated by simulating a transient experiment done in the TALL facility. The mass flow rate was monitored during experiments and compared with the simulated mass flow rate. The coupled codes' mass flow rate correlated well with the experimentally measured rate and showed a marked improvement above the mass flow rate calculated by the stand alone RELAP simulation (Kööp, 2018).

In work previously discussed under section 2.2.1, the 1D code Flownex and the multi-dimensional code GAMMA+ was used by Du Toit et al. (2014) and Rousseau et al. (2015). In these works, the RCCS was simulated and different parts could be simulated in different dimensions. In the GAMMA+ simulations, the riser tube walls were modelled as 3D, to account for radiation and convection on the outer sides of the tubes, as well as the conduction occurring in the walls along multiple axes. The cavity air was modelled as multiple 2D planes and the flow through the riser tubes was modelled in 1D. The GAMMA+ thus had an integrated approach to the coupling of the multi-dimensional domains.

Zhou et al. (2017) used 1D/3D coupled methodology to simulate the phenomena in the residual heat removal pumps used in some nuclear reactor designs. The 3D method was used to simulate phenomena in the pump, while 1D methodology was used for the pipe network connected to the pump. The code CFX was used for the 3D flow and the code Flowmaster was used for the 1D flow. A third code, MpCCI was used to transfer data between the codes on the interface (Zhou et al., 2017). An inlet pressure and outlet mass flow rate were transferred to the 3D domain. In the 1D domain the inlet pressure and mass source were specified as inputs for the domain boundary (Zhou et al., 2017). The values were revised for every timestep and iterated in this way until convergence was achieved.

In summary, 3D CFD codes are used to simulate domains where great detail is paramount. It is however computationally expensive. In contrast, 1D system CFD codes are relatively inexpensive computationally, at the cost of detail. The coupling of 1D/3D (or multidimensional coupling) can optimize the use of computational resources by using higher dimensions to simulate complex domains and lower dimensions to simulate less complex domains. In some cases, the multi-dimensional approach is integrated into the code, such as GAMMA+, which can

combine 1D, 2D and 3D domains in a model. Other approaches use a 3D code (e.g. CFX or Fluent) coupled with 1D codes (e.g. RELAP, Flownex), where one of the codes enables the transfer of solution data. Another approach uses a third code to transfer the data between the 1D and 3D codes. The data transferred consists of parameters such as pressures, mass flow rates or velocities, on the boundaries between the domains. In addition, temperatures and heat energy/heat fluxes could also be transferred, both for direct use or to calculate the needed adjustment of other related parameters. An iterative solver on both ends repeatedly revise the boundary conditions until convergence is achieved.

2.7 Philosophies in coupling

Kööp (2018) simulated a facility by using a combination of 1D and 3D methodology and formulated two distinct philosophies in the coupling of codes for simultaneous simulation. Kööp (2018) specified “domain overlapping” and “domain decomposition”. The former was defined as the 1D method solving the entire loop, while the 3D methodology solves the 3D domain and the 1D method is corrected by results from the 3D domain (Kööp, 2018). The latter was defined as the 1D methodology solves the 1D region, while the 3D domain is solved by CFD and boundary conditions are exchanged between the domains (Kööp, 2018).

Following on the work of Kööp (2018) different philosophies were formulated specifically applicable to the current study. These philosophies are listed and discussed in Appendix E.

2.8 Mixed convection

Jackson et al. (1989) summarized some of the work related to mixed convection done up until 1989. The work detailed the differences in the modelling of the mixed convection regime in various scenarios, including laminar - and turbulent flow, upward - and downward flow. The literature on the laminar mixed convection considers only the aiding flow scenario, i.e. heated upward flow. Jackson et al. (1989) noted that the laminar opposing flow scenario (cooled downward flow) is subject to instability (according to the work of Scheele et al. (1960)) and few

experiments were available on the topic. The works concluded on the aided laminar flow yielded that the regime caused an enhancement of the heat transfer. The turbulent aided regime in contrast was found to impede heat transfer, while turbulent opposing flow enhanced the heat transfer (Jackson et al., 1989). The non-dimensional parameters used by the authors cited by Jackson et al. (1989) to describe the phenomena were the Nusselt number, which showed the impairment or enhancement of the heat transfer, and the Grashof number which is used to determine the transition between regimes.

Aicher and Martin (1997) conducted experiments on aided and opposed mixed convection flow in vertical pipes. The authors also investigated the length to diameter ratio by testing different geometries for the heated pipe. The experimental results were used to develop correlations for the Nusselt number which was calculated by superimposing the free and forced Nusselt numbers. The results of the aided mixed convection were especially dependent on the geometry, seeing a decrease in Nusselt number with an increase in length to diameter ratio.

Celata et al. (1998) conducted experiments on the upward flow of water in a heated pipe. The experimental results showed the deterioration of heat transfer of the aided flow mixed convection regime. Correlations were developed based on the results to calculate the Nusselt numbers and subsequent convection heat transfer coefficients associated with the heat transfer.

Symolon et al. (2004) conducted experiments on mixed convection flow in vertical riser tubes. Tubes with different surface roughnesses were tested, thus investigating the effects of the friction factor on the transition between forced and mixed convection flow (Symolon et al., 2004). A correlation was developed for the Nusselt number from the experimental data. The correlation also includes the effect of the friction factor in addition to the non-dimensional buoyancy number (Symolon et al., 2004).

Shin et al. (2017b) experimentally investigated vertical flow of air in the riser of an experimental facility. The facility was constructed to study the mixed convection regime that could also occur in a reactor cavity cooling system (Shin et al., 2017a). The facility was dubbed the Riser Heat transfer Experimental Facility (RHEF) and differs from the NSTF or NACEF facilities in that it was designed such that all other parts of the system except for the riser tube could be controlled (Shin et al., 2017a).

The correlation developed by Symolon et al. (2004) was tested against the experimental results. The correlation was modified by changing some of the quotients and powers in the proposed correlation, to improve the correlation between the experimental and theoretically predicted results for the specific case (Shin et al., 2017b).

The comparison between the experimental results, Symolon's correlation (Symolon et al., 2004) and the modified Symolon's correlation (Shin et al., 2017a) is shown in Figure 2.8-1. For the specific case at hand, the original form of Symolon's equation was accurate until a buoyancy number of 1000 was reached. Thereafter the modified correlation was much more accurate (Shin et al., 2017a).

Also apparent in Figure 2.8-1, is the decrease in Nusselt number at the onset of the mixed convection regime at the buoyancy number of 0.01×10^4 . The Nusselt number eventually increases again after the free convection regime becomes dominant subsequent to the increase in buoyancy number.

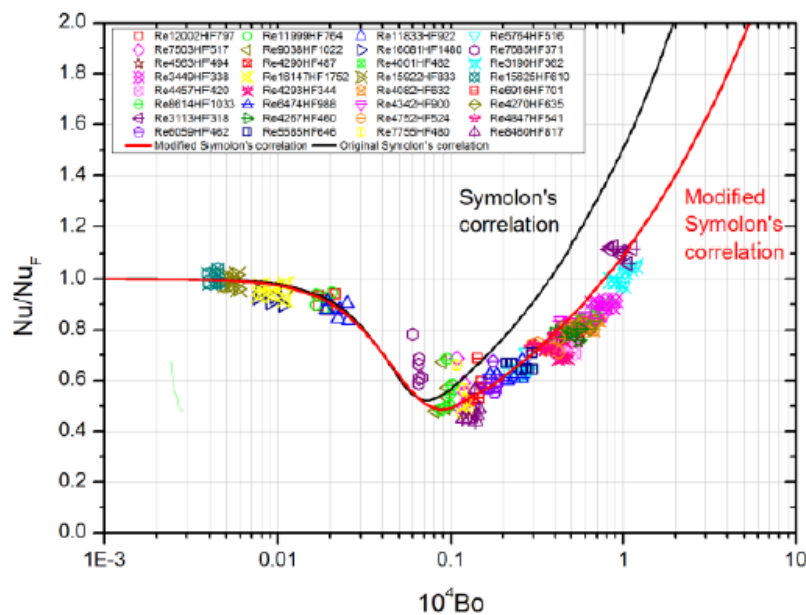


Figure 2.8-1. Comparison of correlations and experimental results. (Shin et al., 2017b).

The following conclusions can be drawn from the above literature on mixed convection. Aided turbulent mixed convection results in the deterioration of heat transfer. The Nusselt number is an important parameter and is used to observe the changes in heat transfer. Correlations were developed for the Nusselt number in all of the above-mentioned literature.

From the cases of Symolon et al. (2004), Shin et al. (2017a), Aicher and Martin (1997) and Celata et al. (1998) it seems that the correlation that is used to calculate the Nusselt number is dependent on geometry and experimental conditions, making every case a unique challenge. The transition to and from the mixed convection regime could therefore also be slightly different for each case. In terms of heat transfer, the mixed convection regime is less desirable, since a lower Nusselt number will result in lower convection heat transfer. Furthermore, quantities such as the Nusselt number, buoyancy number and friction factor are important in designs where the mixed convection regime is possible.

2.9 Conclusion

Simulations and experimental works pertaining to the RCCS have been discussed, with the focus on the methodologies used during simulation of these systems. 3D CFD provides a tool to predict the phenomena that occurs in the RCCS. 1D methodology provides system level information about the phenomena. The combination of the methodologies seems logical where strengths of the respective codes can be exploited to obtain the best balance between accuracy and computational resources.

CHAPTER 3 THEORETICAL BACKGROUND

3.1 Introduction

The theoretical background of importance to the study is discussed in this chapter. The discussion gives a brief overview of the formulations of the conservation laws and equations describing turbulence applied by computational fluid dynamics codes (Ansys Fluent and Star-CCM+). An overview is also given on the user specified mixed convection theory applied by the 1D code (Flownex SE) in this study.

3.2 Computational Fluid Dynamics (CFD)

The process to be modelled consists of the convection and radiation heat transfer that occurs in the front and back cavities, convection within the riser tubes and the conduction in the fin and riser tube walls.

CFD codes such as Ansys and Star-CCM+ use the control-volume approach to discretize the domain into a finite number of interacting sub domains (control volumes). The fundamental equations are then integrated over the control volumes to yield algebraic equations for the discretized variables such as velocity and temperature. A linearization of the algebraic equations is carried out to yield a matrix of linear equations of the discretized variables that are solved.

The fundamental equations used to simulate the domain include the equations for the conservation of momentum, mass and energy. The equation for the conservation of momentum is given as (Ansys, 2017a):

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\tau}) + \rho \vec{g} + \vec{F} \quad (1)$$

The first term on the left accounts for momentum change during transients and the second term accounts for changes in momentum due to fluid moving over the boundary of the control volume. The third term accounts for the effect of a change in pressure on the momentum, the fourth accounts for the effect of the stress tensor, the effect of buoyancy and external forces are accounted for by the fifth and sixth terms respectively. ρ represents the density (in kg/m³), $\vec{v}$ the velocity vector (m/s), p the pressure, $\vec{g}$ the gravitational acceleration (m/s²) and $\vec{F}$ the external

forces. ∇ is the operator for the partial derivative of the relevant quantity. The stress tensor, $\bar{\tau}$ in (1) is given by (Ansys, 2017a):

$$\bar{\tau} = \mu \left[\left(\nabla \vec{v} + \nabla \vec{v}^T \right) - \frac{2}{3} \nabla \cdot \vec{v} \mathbf{I} \right] \quad (2)$$

Where μ is the dynamic viscosity (kg/m-s) and $\mathbf{I}$ is the unit tensor.

The conservation of mass is given as follows (Ansys, 2017a):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (3)$$

The first term accounts for changes in mass due to transient effects, while the second term accounts for the movement of fluid over the control volume boundaries. The last term, S_m accounts for any mass sources.

The conservation of total energy, E is implemented in Fluent in the form (Ansys, 2017a):

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot \left(k_{\text{eff}} \nabla T - \sum_j h_j \vec{J}_j + (\bar{\tau}_{\text{eff}} \cdot \vec{v}) \right) + S_h \quad (4)$$

The first term accounts for energy changes due to transient effects, the second term for changes in energy due to fluid movement over control volume boundaries. The term within brackets account for changes in energy due to conduction, diffusion and viscous dissipation, respectively. S_h represents energy sources within a control volume. k_{eff} is the effective conductivity and is a function of the thermal conductivity and the turbulent thermal conductivity (W/m-K). T is the temperature (K), h_j is the enthalpy, $\vec{J}_j$ is the flux at which a species j diffuses and $\bar{\tau}_{\text{eff}}$ is the effective stress tensor.

Additionally, turbulence models were used in conjunction with the conservation equations. The k- ω SST model is formulated in Ansys as follows (Ansys, 2017a):

The equation for the kinetic energy, k :

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial X_i}(\rho k u_i) = \frac{\partial}{\partial X_j} \left(\Gamma_k \frac{\partial k}{\partial X_j} \right) + G_k - Y_k + S_k \quad (5)$$

The equation for the specific dissipation rate, ω :

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial X_j}(\rho \omega u_j) = \frac{\partial}{\partial X_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial X_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (6)$$

The first terms (in eqns. (5) and (6)) account for changes in the quantity due to transient effects, the second terms due to the transport of fluid over a given control volume. The third term accounts for the diffusion of the respective quantity, with G and Y representing generation and dissipation. S in the equations represent user defined source terms where applicable. X_j represents position within a reference frame. Γ is the effective diffusivity, while D_ω is the term accounting for cross-diffusion.

In the present study, one of the algorithms used by Fluent was the SIMPLE algorithm (Semi-Implicit Method for Pressure Linked Equations). The algorithm is initialized by guessing an initial pressure field, which is used to solve the discretized momentum equations. The solution to the momentum equations is then used to adapt the initial pressure. The corrected pressures are then used to re-calculate the velocity field. The other discretized transport equations are solved at this point where after a convergence check can be done. The process is done iteratively until the required convergence is achieved (Versteeg & Malalasekera, 2007).

In modelling any CFD application, the appropriate use of wall functions is important. Wall functions reduce computational resources by approximating the velocities close to a wall, instead of directly modelling the velocity profile. y^+ is the non-dimensional distance from the wall to the first node in the mesh of the flow field next to the wall. Figure 3.2-1 shows for values up to $y^+ \approx 5$, $u = y^+$. The log law approximation alternatively is an adequate approximation between $30 < y^+ < 300$. Between y^+ of 5 and 30 and above 300 the deviation of these approximations from the typical velocity profile is apparent.

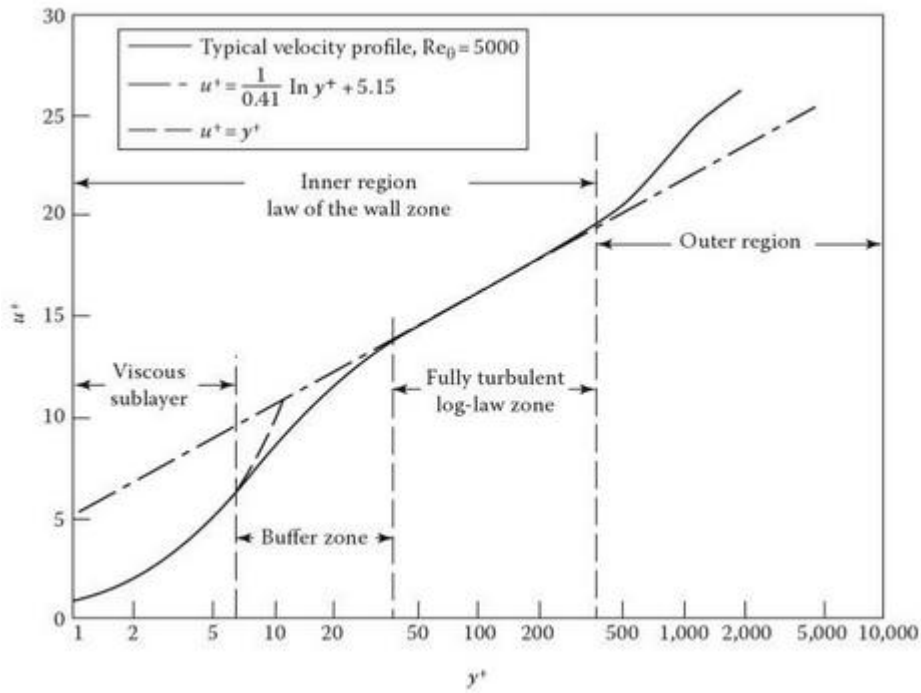


Figure 3.2-1. Wall treatment for different y^+ values. (Raghav, 2018).

3.3 Non-dimensional numbers

The principal non-dimensional numbers for analysing the heat transfer in the current work is the Nusselt number, Grashof number, Rayleigh number, Reynolds number, Aicher number and buoyancy parameter.

The Grashof number represents the ratio between the buoyancy forces and the viscous forces acting on a fluid and is given by Incropera et al. (2013):

$$Gr = \frac{g\beta(T_s - T_\infty)D_h^3}{\nu^2} \quad (7)$$

Where β is the thermal expansion coefficient ($1/K$), T_s , T_∞ represent the surface and free stream temperatures (in K). D_h represents the hydraulic diameter (or characteristic length) (in m) and ν the kinematic viscosity (m^2/s).

The Prandtl number is the ratio of momentum and thermal diffusivity and is given by Incropera et al. (2013):

$$Pr = \frac{C_p \mu}{k} = \frac{\nu}{\alpha} \quad (8)$$

Where C_p is the specific heat capacity (J/kg-K), k is the thermal conductivity (W/m-K) and α is the thermal diffusivity (m²/s).

The Rayleigh number is the product of the Grashof and Prandtl numbers and is given by Incropera et al. (2013):

$$Ra = \frac{g\beta(T_s - T_\infty)D_h^3}{\nu\alpha} \quad (9)$$

The Reynolds number is the ratio of viscous to inertial forces acting on a fluid and is given by Incropera et al. (2013):

$$Re = \frac{\rho u D_h}{\mu} \quad (10)$$

However, $\dot{m} = \rho u A$, thus $\rho u = \frac{\dot{m}}{A}$ therefore:

$$Re = \frac{\dot{m} D_h}{\mu A} \quad (11)$$

Where $\dot{m}$ is the mass flow rate (kg/s) and A is the cross sectional flow area (m²).

The Nusselt number is of paramount importance when considering the convective heat transfer and is dependent on the Rayleigh, Reynolds and Prandtl numbers.

The Nusselt number is the ratio of convection to conduction heat transfer and is given by Incropera et al. (2013):

$$Nu = \frac{h D_h}{k} \quad (12)$$

Where h represents the convective heat transfer coefficient ($W/m^2 \cdot K$).

A distinction is made between the forced, free and mixed convection Nusselt numbers. The forced convection Nusselt number can be calculated by Incropera et al. (2013):

$$Nu_{\text{forced}} = 0.021 \cdot Re^{0.8} \cdot Pr^{0.4} \cdot \eta \quad (13)$$

Where η the property index, is given by Incropera et al. (2013):

$$\eta = \left(\frac{\mu_f}{\mu_w} \right)^{0.14} \quad \text{for liquids or} \quad (14)$$

$$\eta = \left(\frac{T_f}{T_w} \right)^{0.5} \quad \text{for gases}$$

The free convection Nusselt number for vertical flow is given by:

$$Nu_{\text{free}} = \left(0.825 + \frac{0.387 Ra^{\frac{1}{6}}}{\left(1 + \left(\frac{0.492}{Pr} \right)^{\frac{9}{16}} \right)^{\frac{8}{27}}} \right)^2 \quad (15)$$

The mixed convection Nusselt number for heated upward flow is calculated by Celata et al. (1998):

$$Nu_{\text{mixed}} = \left(|Nu_{\text{forced}}^3 - Nu_{\text{free}}^3| \right)^{\frac{1}{3}} \quad (16)$$

The Aicher number is a parameter than can be used to determine the flow regime. It is calculated by Aicher & Martin (1997):

$$\text{Aicher} = \frac{\text{Ra}^{\left(\frac{1}{3}\right)}}{\text{Re}^{0.8} \text{Pr}^{0.4}} \quad (17)$$

When $0.05 < \text{Aicher number} < 0.2$ the regime is mixed convection. Aicher numbers larger than 0.2 indicate free convection and smaller than 0.05 indicate forced convection (Aicher & Martin, 1997).

Another important variable in analysing natural convection is the buoyancy parameter, which is given by Shin et al. (2017b) and Symolon et al. (2004):

$$\text{Bo} = \frac{\text{Gr}}{\text{Re}^3 \text{Pr}^{0.5}} \quad (18)$$

The ratio of the calculated Nusselt number and the forced convection Nusselt number is commonly plotted as a function of the buoyancy number. The powers used for the Reynolds and Prandtl numbers vary slightly between authors, Shin et al. (2017a) used 2.7 and 0.5 respectively, Behzadmehr et al. (2003) used 3.425 and 0.8 respectively. This results in a graph which is effective in illustrating the effect of the mixed convection regime on the Nusselt number and thus the convection heat transfer.

3.4 1D System CFD theory

Flownex uses an Eulerian reference frame M-Tech Industrial (2019b) in which the region is discretized into fixed control volumes. Flow of mass, momentum and energy over the boundaries of the control volumes are used to characterize the phenomena. Figure 3.4-1 shows the implementation of the Eulerian reference frame used by Flownex. The normal vectors on the surface of the volume determine the sign (positive or negative) of the movement over the boundaries. Flow into the volume is negative, while outflow is positive (M-Tech Industrial, 2019b).

The conservation of mass used for each control volume is given by M-Tech Industrial (2019b):

$$\frac{Dm}{Dt} = \oint_{CS} (\rho \vec{V} \cdot d\vec{A}) + \frac{\partial}{\partial t} \int_{CV} \rho \, dv \quad (19)$$

Where m is the mass in kg.

Similarly the equation for the conservation of momentum, M-Tech Industrial (2019b):

$$\oint_{CS} \vec{V} (\rho \vec{V} \cdot d\vec{A}) + \frac{\partial}{\partial t} \int_{CV} \rho \vec{V} \, dv = \oint_{CS} \vec{T} \cdot d\vec{A} + \int_{CV} \vec{B} \cdot dv \quad (20)$$

Equation (20) is similar to Equation (19) but includes the velocity vector at the first and second terms. These terms represent the nett change in momentum as a result of flow over the control surface (CS) and change in momentum within the control volume (CV), respectively. The third and fourth terms in Equation (20) represent the momentum change due to the stress tensor ($\vec{T}$) and body force ($\vec{B}$) respectively. In Equation (19) and (20) v denotes volume (m³) while $\vec{V}$ denotes a velocity vector (m/s).

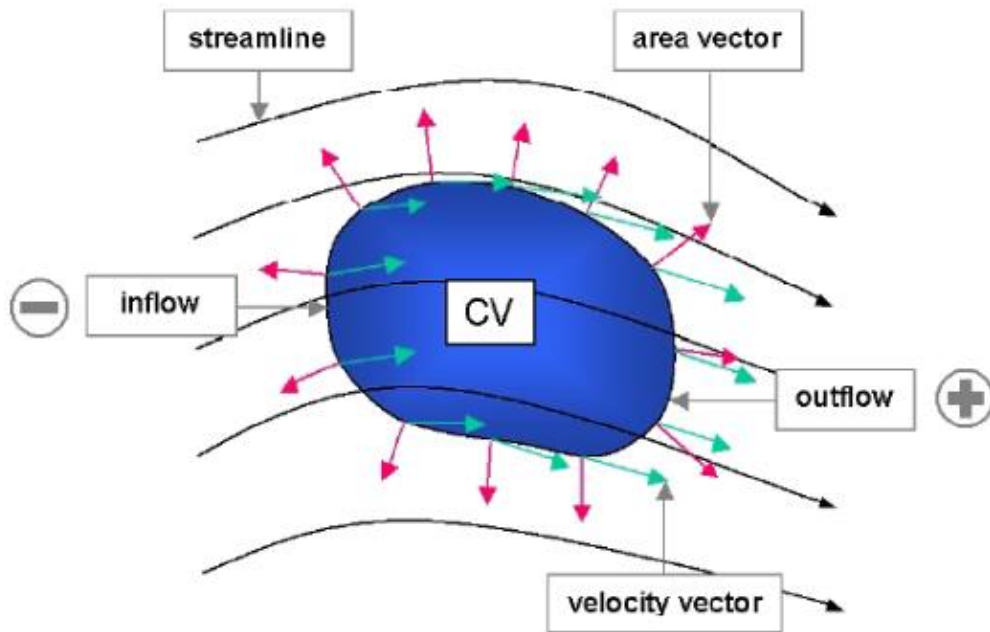


Figure 3.4-1. Eulerian reference frame used by Flownex for mass flow. (M-Tech Industrial, 2019b).

The conservation of energy in Flownex is implemented in the form, M-Tech Industrial (2019b):

$$\frac{D(me)}{Dt} = \oint_{CS} e (\rho \vec{V} \cdot d\vec{A}) + \frac{\partial}{\partial t} \int_{CV} \rho e dv = Q_H - \dot{W} \quad (21)$$

Where m is the mass, Q_H represents the heat transfer rate to the control volume, while $\dot{W}$ represents the rate of work done by the fluid. e represents the specific total of the internal, kinetic and potential energy of the fluid.

Q_H can be transferred by several ways including conduction, convection and radiation. The implementation of the conduction heat transfer in Flownex is as follow, M-Tech Industrial (2019b):

$$Q_H = -kA \frac{\partial T}{\partial n} \quad (22)$$

Where n represents the normal direction with respect to which the derivative is taken, typically normal to a surface.

The equation used to account for the heat transfer by convection is given by M-Tech Industrial (2019b):

$$Q_H = -hA(T_S - T_\infty) \quad (23)$$

The radiation heat transfer between two surfaces is calculated by the equation, M-Tech Industrial (2019b):

$$Q_H = \epsilon \sigma A (T_1^4 - T_2^4) \quad (24)$$

ϵ represents the emissivity of the radiating surface, σ is the Stefan-Boltzmann constant and T_1 and T_2 the temperatures of any two radiating surfaces.

The work defined in Equation (21) is written in the form, M-Tech Industrial (2019b):

$$W = \int_{r1}^{r2} \vec{F} d\vec{r} \quad (25)$$

Where a Force $\vec{F}$ causes a displacement from $r1$ to $r2$.

The flow resistance components (shown as 16, 17 and 18 in Figure 7.3-1) use pressure inputs from the scripting component to calibrate a flow admittance, which is the inverse of flow resistance. The total pressure drop in the component is calculated by, M-Tech Industrial (2019a):

$$\Delta p_0 = \frac{|\dot{m}| \dot{m}}{A_f} + \rho g \Delta z \quad (26)$$

Where Δp_0 is the total pressure drop across the resistance component, $\dot{m}$ is the mass flow rate, A_f is the flow admittance and Δz is the change in elevation across the component.

Also, the inverse of the admittance is equal to, M-Tech Industrial (2019a):

$$\frac{1}{A_f} = \frac{f L}{D} \frac{1}{2 \rho A^2} \quad (27)$$

Where f is the friction factor and L is the flow channel length. The admittance is therefore a function of pressure losses across a component, excluding the pressure loss due to changes in elevation.

The solution is obtained in Flownex by executing a pressure correction algorithm with the following steps (M-Tech Industrial, 2019b):

- 1) Values are guessed for the pressures at the nodes.

- 2) Mass flow rates are calculated based on the guessed pressures and $\Delta p - \dot{Q}$ relationships. (Note that $\dot{Q}$ denotes a volume flow rate not to be confused with the heat transfer rate Q_H used earlier).
- 3) A test is then done to check whether continuity is satisfied for the nodes.
- 4) Pressures are adjusted based on the previous check to establish continuity.
- 5) The mass flow rate is subsequently updated to match the adjusted pressures.
- 6) Steps 1 to 5 are repeated until convergence is achieved.
- 7) The energy equation can then be solved.
- 8) Steps 1 to 7 are repeated until convergence is reached.
- 9) Advance to a new time step and repeat steps 1 to 8 (for transient cases only).

The algorithm employs a segregated solver, as can be derived from the mass flow and pressures being solved first, followed by the solution of the energy equation repeated sequentially.

3.5 Conclusion

The relevant theory has been discussed, focussing on a broad overview of the theory used by the CFD codes and the non-dimensional numbers used to assess the heat transfer. The conservation equations for mass, momentum and energy are used by systems CFD and CFD codes to set up a matrix of equations to be solved. The CFD codes, in addition also use turbulence models, in this case the $k-\omega$ model, which resolves the eddies that occur as a result of turbulence. The dimensionless distance between the wall and the first node in the fluid region, y^+ , should be below 5 or $30 < y^+ < 300$ for the velocities in the region of the boundary layer to be calculated correctly.

The SIMPLE algorithm is often used in CFD codes, which solve the matrix of equations iteratively by continually improving the pressures based on solutions of the velocity fields obtained by the guessed pressure.

The algorithm used by Flownex uses a relationship of pressure and mass flow rate, which also initialises with guessed pressures, which are corrected by the mass flow rate that are subsequently calculated.

The Nusselt number is of paramount importance when considering the convective heat transfer and is dependent on the Rayleigh, Reynolds and Prandtl numbers. The Nusselt number is calculated differently based on the regime i.e. free convection, mixed convection or forced convection.

CHAPTER 4 FACILITY DESCRIPTION

4.1 Introduction

The facility that was modelled is described in this chapter. The facility was chosen since the data regarding the geometry, materials, boundary conditions and experimental results were available. It was constructed at the University of Wisconsin (UW) (Madison) and represents a scale model of the reactor cavity cooling system (RCCS) (Lisowski, 2013).

4.2 UW facility overview

The facility is 5 m tall and represents a $\frac{1}{4}$ scale model of the reactor cavity cooling system of a gas-cooled reactor. The cavity portion represents a 5 degree radial slice of the full prototype and thus contains three riser tubes of the 227 of the full scale design (Lisowski, 2013). The loop consists of a water tank, downcomer, heated section and adiabatic chimney. 34 heater elements are installed in the heated section to emulate the heat emitted from the reactor pressure vessel (Lisowski, 2013). The heaters have a maximum power of 42.5 kW and different power profiles can be configured, including axial, linear and radial (Lisowski, 2013). The riser tubes are connected by 6 mm plates welded onto the tubes at their centreline (Lisowski, 2013). Thermal insulation was applied extensively on the entire facility to inhibit losses to the environment (Lisowski, 2013). The steady state operation of the loop is ensured by a second loop where a heat exchanger is used to remove the heat from the feed water tank, thereby keeping the tank outlet temperature constant (Lisowski, 2013).

The experimental facility is shown in Figure 4.2-1. Water is fed from the tank labelled “C” into the heated test section, “B”. The temperature of the coolant water in the tank is kept at a constant temperature by means of the heat exchanger “D”. Figure 4.2-2 shows a cross sectional view of the heated test section with dimensions provided in cm. The figure also shows the position of the heater elements on one of the walls of the cavity. The spacing between the heater elements was subject to an optimization study (Lisowski, 2013). The figure also shows the spacing of the three riser tubes and the fins between the riser tubes. Note the fin/tube assembly divides the heated section into two separate parts, hereafter referred to as the front and back cavity. The cavity containing the heater elements is hereafter considered as the front cavity.

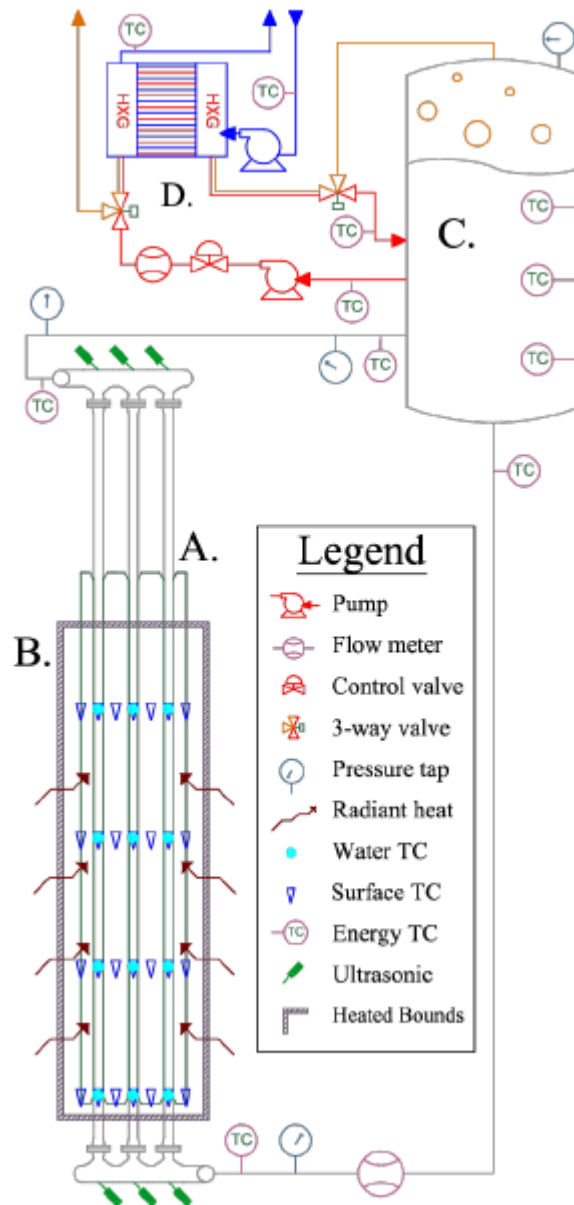


Figure 4.2-1. The process layout of the experimental facility. (Lisowski, 2013).

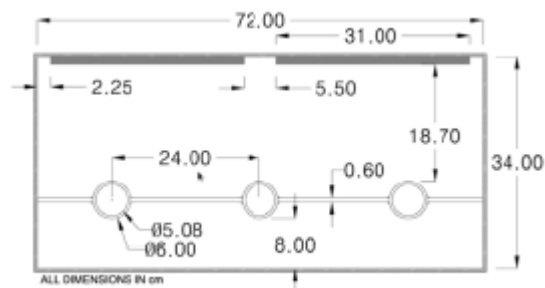


Figure 4.2-2. Cross sectional view of the heated test section. (Lisowski, 2013).

Measurements were taken on multiple locations in the facility, with the highest density of measurements taken in the heater box (Lisowski, 2013).

The author reported that care was taken to insulate the system on all of the components, including the heated section, pipe network and tank.

Isothermal experiments of fixed flow were conducted to calculate the secondary losses caused by the various components in the loop. Differential pressures were measured at various points to isolate the components across which the frictional and minor losses could be calculated.

Loss coefficients associated with the downcomer/lower network, the test section, chimney, water tank and the complete loop were considered (see Table 4-1). Figure 4.2-3 shows a comparison between the measured and analytically calculated pressure drop, as a function of flow rate. The graph shows an increasing pressure drop with increasing mass flow rate for the tested mass flow rates. The intersect point was 5.8 g/min, which was the mass flow rate used to calculate the values in Table 4-1.

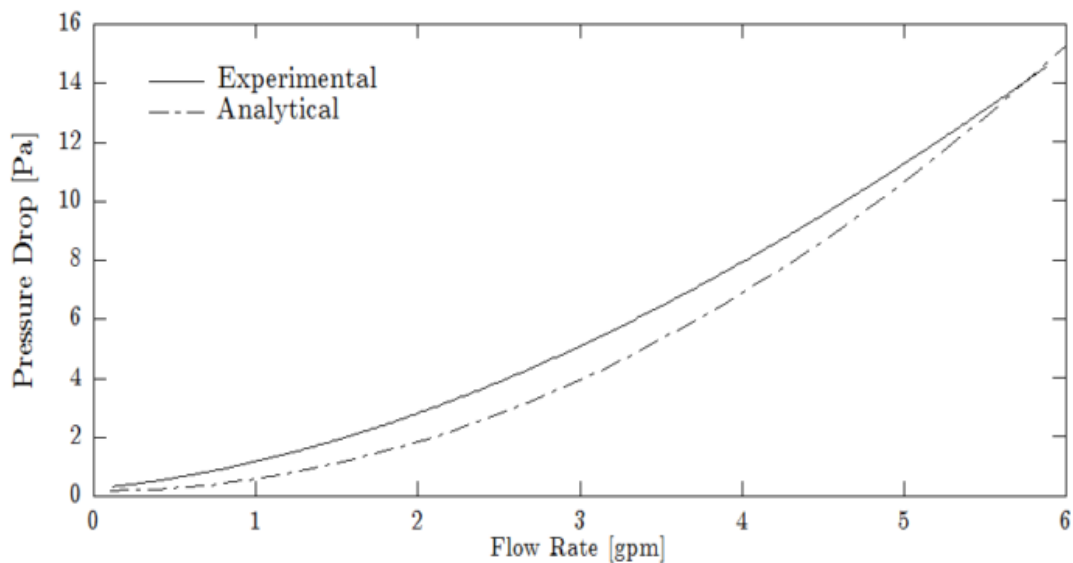


Figure 4.2-3. Pressure drop as a function of flow rate (Lisowski, 2013).

Table 4-1. Results of the isothermal pressure drop experiments (Lisowski, 2013).

Component	Volume [L]	Diameter [cm]	Frictional Loss [Pa]	K factor [-]
DC, lower network	58.9	10.2	2.75	2.78
Test Section	40.2	5.1	8.63	8.72
Chimney	21.9	10.2	3.07	3.10
Water tank	1,200	91.4	-	-
Complete Loop	1,321	-	14.45	14.60

Single phase and two-phase experiments were performed (Lisowski, 2013). Five variations of the steady state experimental results are shown in Table 4-2. The power of the heaters was adjusted which created the driving buoyancy force resulting in the mass flow rate and temperature rise in the risers (Lisowski, 2013). The HXG energy was removed by the heat exchanger from the storage tank (Lisowski, 2013). The window in which the steady state condition was maintained is also indicated (Lisowski, 2013).

Table 4-2. Experimental runs done by Lisowski (2013).

		Run007 <i>Steady Load</i>	Run008 <i>Median</i>	Run006 <i>Decay Load</i>	Run009 <i>Extended</i>	Run029 <i>Scoping</i>
Power	kW	4.63	7.27	9.91	12.56	15.19
Heat Flux	kW/m ²	2.83	4.44	6.06	7.67	9.29
Mass Flow	kg/s	0.37	0.45	0.52	0.58	0.67
Inlet	°C	30.25	29.98	29.99	29.90	30.49
Outlet	°C	32.67	33.16	33.82	34.27	34.88
ΔT	°C	2.42	3.18	3.83	4.37	4.39
Riser energy	W	3,778	5,979	8,402	10,646	12,254
HXG energy	W	3,633	6,058	8,289	10,370	11,873
Heater power	W	4,686	7,265	10,015	12,406	14,822
Heat losses	W	791	999	1,140	1,272	2,140
Rise rate	°C/m	1.92	3.49	4.63	6.25	9.02
Window	min	130	132	130	130	210

4.3 Conclusion

The UW facility is a scale model of the RCCS of a gas-cooled reactor. The facility was described, and the experimental results were listed. Since the boundary conditions, engineering drawings detailing the geometry and some results are available, it should be possible to model the thermal-hydraulic behaviour of the facility using computational fluid dynamics (CFD) methodologies.

CHAPTER 5 - METHODOLOGY VERIFICATION

5.1 Introduction

The methodology is described in this chapter. Different methods of analysis were used and applied to the experimental facility. The heated section of the facility was isolated and simulated by using the 3D CFD codes Ansys Fluent (version 18.2) and Star-CCM+ (version 14.02.010).

5.1.1 Simple cubical cavity simulation

A simple cubical cavity was simulated to test whether the methodology was correct as implemented in the CFD code. Secondly, it was tested whether a wall with an adiabatic outer surface had to be modelled explicitly or whether the wall could be omitted and an adiabatic (inner) wall surface would be sufficient.

The geometry that was tested was a cube of 0.05 m x 0.05 m x 0.05 m. Air was chosen as the coolant and atmospheric pressure was chosen as the reference pressure. Two opposing walls were given temperature boundary conditions (300 K and 450 K respectively) while the other sides of the cube were set to be adiabatic. The shell was given a wall thickness of 6 mm. In the case with the shell the temperature boundary conditions were specified on the outside surface of the shell. The mesh size for the air region was the same and the maximum face size/tet size was set to 0.3 mm and 5 prism layers were used on all internal sides of the cubical cavity. The SST k- ω model was used and surface-to-surface radiation was enabled. A value of 0.8 was specified for the emissivity of all the surfaces participating in the radiation heat transfer.

The velocity magnitude and temperature distributions obtained (on the centre line between the walls) by using Fluent are shown in Figure 5.1-1 and Figure 5.1-2. The graph shows the increase of the velocity magnitude close to the wall within the boundary layer, where after the velocity magnitude decreased towards the middle of the cavity. The methods show a similar trend, but the configuration without a shell shows higher velocities in the boundary layer and bulk regions.

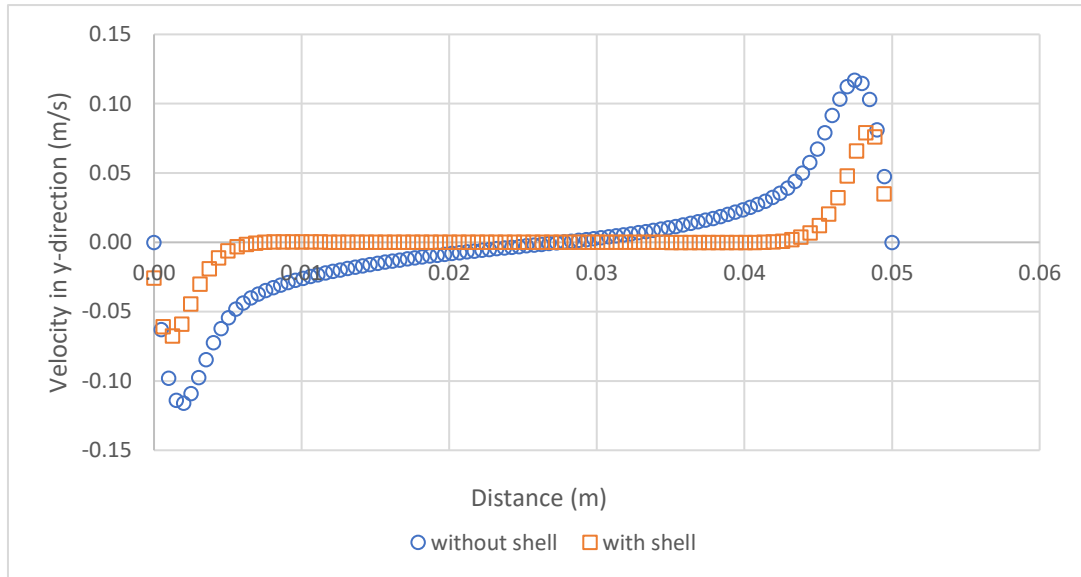


Figure 5.1-1. Velocity distribution in the cubical cavity.

The temperature distributions in the cavity is shown in Figure 5.1-2 and were also similar except for the apparent temperature decrease at 0.04 m. The temperatures show a steep decrease and increase close to the cold and hot sides respectively. The temperature distributions calculated by the two models were similar in the boundary layer. The cubical cavity without the shell calculated a slightly higher temperature in the middle of the cavity.

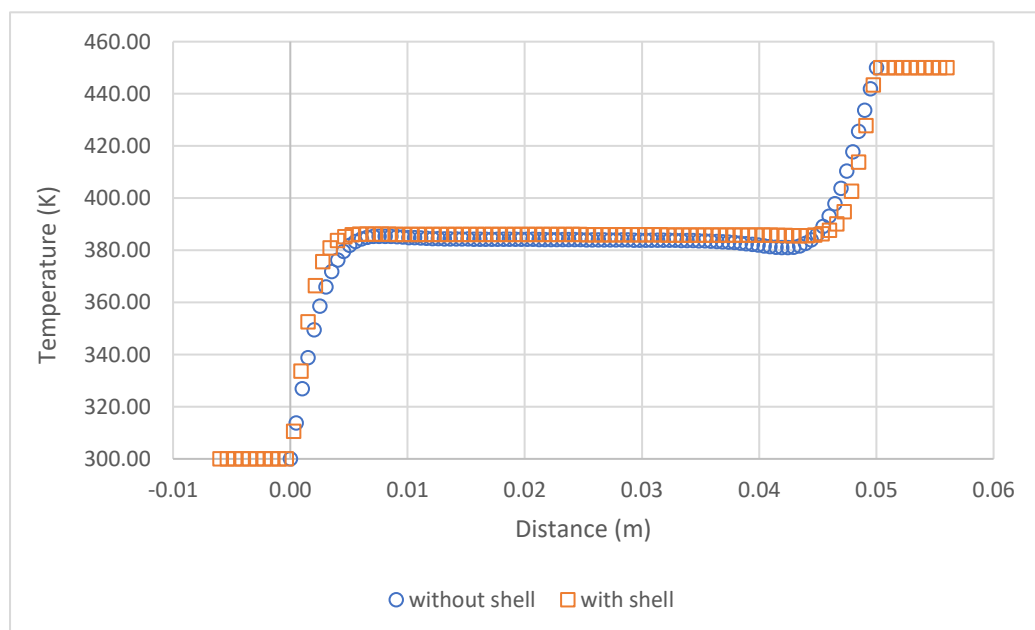


Figure 5.1-2. Temperature distribution within the cubical cavity.

The result indicate that a shell region is important and should be explicitly included when simulating convection and radiation, since the shell could have a non-negligible influence on the velocity and temperature profiles.

5.1.2 Natural circulation in an air-cooled cavity

Section 5.1.1 tested whether it was necessary to explicitly model the walls surrounding a fluid domain. Section 5.1.2 aims to validate the simulation methodology by comparing the results with experimental results. The study of Zitzmann et al. (2005) was used for validation purposes. The geometry that was simulated was based on the “Benchmark 3” study in Zitzmann et al. (2005). The geometry consisted of an air-filled cavity measuring 750 mm in length, 750 mm high and 375 mm wide. Two opposing walls were given temperature boundary conditions of 50 °C and 10 °C (see Figure 5.1-3).



Figure 5.1-3. Air-filled cavity geomtery.

The hot and cold walls had a thickness of 6 mm, while the top and bottom walls had a thickness of 1.5 mm (Tian & Karayiannis, 2001). A mild steel was used and therefore a carbon steel was specified in the CFD code. The side vertical walls are considered adiabatic since the experimental facility had guard cavities on these walls minimizing the heat losses to the environment (Tian & Karayiannis, 2001).

The geometry was meshed by specifying a maximum face and “tet size” of 2 mm and a global growth size of 1.2 was used. Inflation layers were specified on all surfaces with a first layer thickness of 0.1 mm and a growth rate of 1.2. A total of 8 prism layers were inserted in this manner. A hexahedral dominant mesh was generated by these settings with a total of 34643142 cells. The SST k- ω model was used, energy transfer was enabled, but radiation heat transfer was neglected. The simulation was run for ≈ 5200 iterations, where after the residuals were below 10^{-4} .

Zitsmann et al. (2005) presented a comparison between results obtained by using the SST k- ω model and experimental results published by Tian & Karayiannis (2001). The results from Zitsmann et al. (2005) and Tian & Karayiannis (2001) are compared with results obtained by using Fluent in Figure 5.1-4. The temperatures were taken at mid height starting from the surface of the heated plate and extending 15 mm into the cavity.

The Fluent results extracted for Figure 5.1-4 were taken on a line probe which sampled 100 equally spaced points. Error bars were added to the experimental results to show variations of 5%. The results seem to show reasonable agreement. The Fluent results in the near wall region shows good agreement, below 5% but over predicts the temperatures further from the wall.

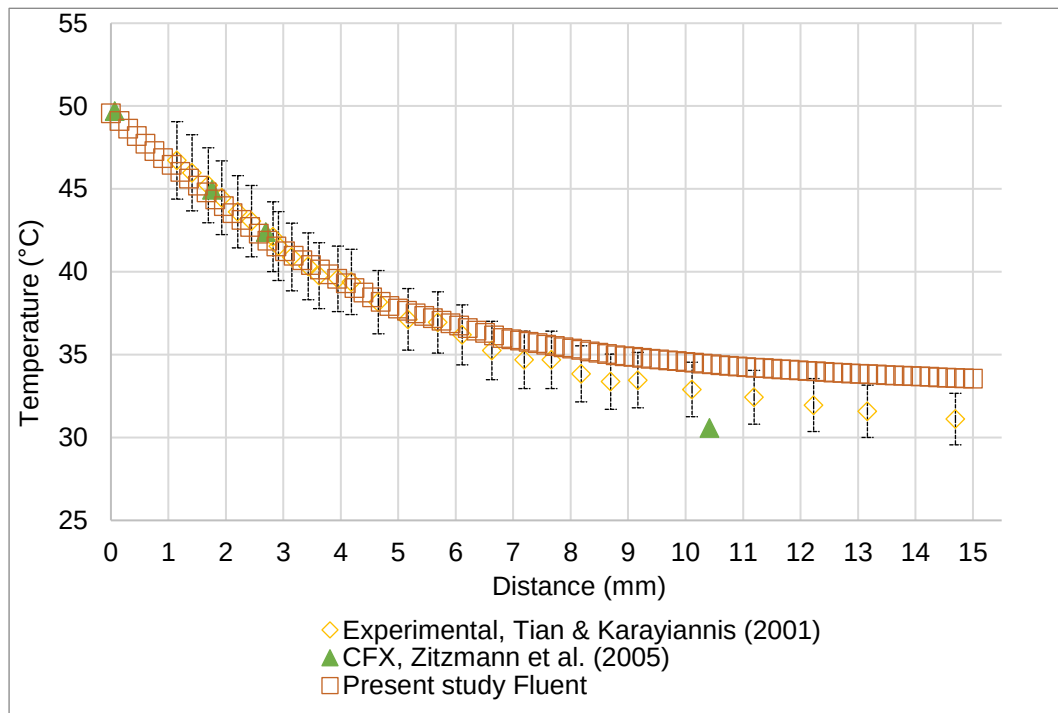


Figure 5.1-4. Comparison between calculated and measured results.

Both the results obtained by using CFX by Zitzmann et al. (2005) and Fluent in the present study predicted the temperature profile close to the wall (up to 6 mm) well. For the temperatures obtained by using Fluent the error was close to 5% at 10 mm, reaching a maximum error of around 8% at 15 mm. The CFX results under predicted the temperature at 10 mm while the Fluent results over predicted the temperature. The exact reason is unknown but is expected to be due to small differences in the implementation of the different codes (for example Fluent uses a cell centred approach while CFX a vertex centred approach).

There could be differences in the fluid properties used by Zitzmann et al. (2005) and the present study. The present study assumed the air could be approximated to an ideal gas, while it is uncertain which fluid property approach Zitzmann et al. (2005) used. Lastly, continuous improvements are made to CFD codes and as such the version used by Zitzmann et al. (2005) (CFX-5) has had multiple updates between 2005 and the present study. The present study used Fluent version 18.2 which was released in 2018. It is possible that the results could differ depending on the version and date of release.

Zitzmann et al (2005) noted that the SST $k-\omega$ model showed the best performance in terms of agreement with the experimental results. Other turbulence models were therefore not tested in the present study. The SST $k-\omega$ model in both Zitzmann et al (2005) and the present study showed similar degrees of deviation from the experimental results further away from the wall. An in depth investigation into this region was considered to be beyond the scope of the present study.

It was concluded that the results obtained by using the SST $k-\omega$ model showed adequate agreement with experimental results. The methodology was also verified against the results of another code used by other authors.

5.1.3 Isothermal simulation of the riser tubes

An isothermal simulation was done on a sub part of the water region of the facility, namely the bottom manifold, the three risers and the outlet manifold (shown in Figure 5.1-5). The 5 different mass flow rates corresponding to the 5 heater power levels done by Lisowski (2013) were simulated, with the aim of investigating the effect of the mass flow rate on the pressure drop.

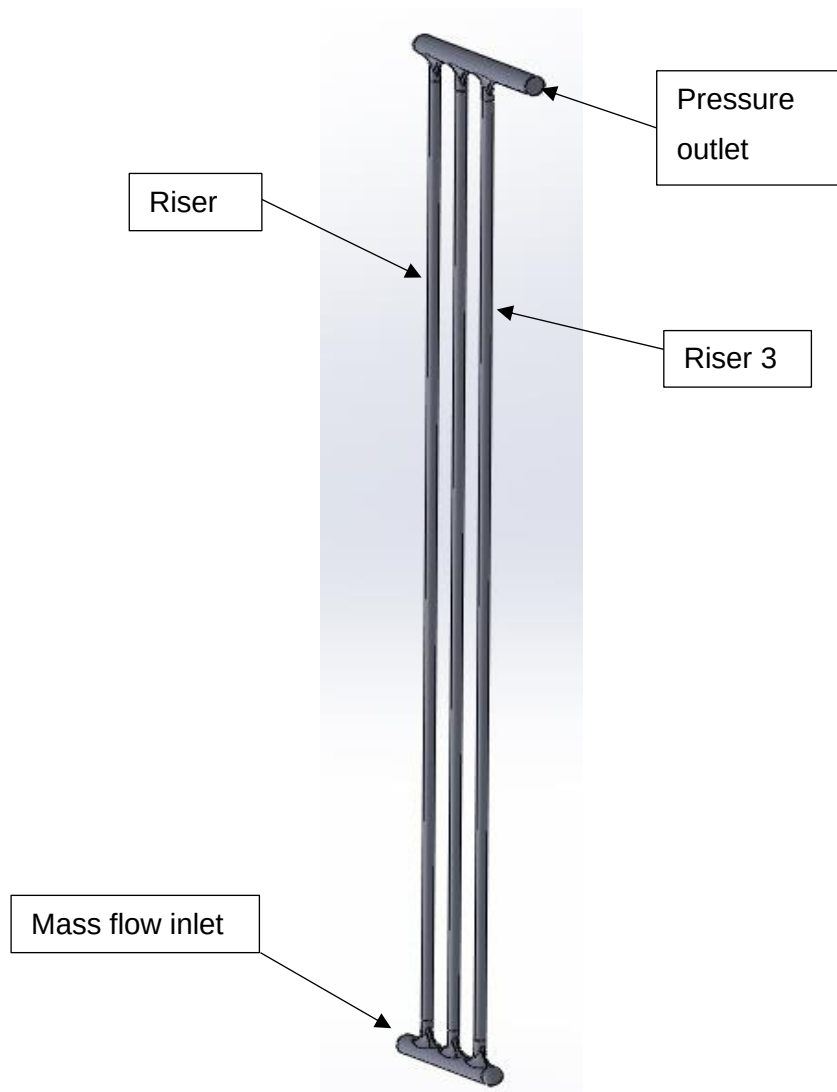


Figure 5.1-5. Geometry used to model the water region.

The mesh consisted of 7 660 154 elements and 2 815 984 nodes. The maximum face size of 5 mm was used for the riser tubes with the size function set to proximity and curvature, with the

growth rate set at 1.05. A different sizing was applied to the manifolds, namely a maximum face size of 3 mm with the sizing function set to the curvature and a growth rate of 1.0. 8 prism layers were used on all of the boundaries. A first layer thickness of 0.3 mm was specified for the riser tubes and 0.2 mm for the manifolds. The values were obtained by trial and error runs until the y^+ values on the surfaces were below 5. The orthogonal quality was found to have a value of 0.825, which was deemed acceptable (Ansys, 2017b).

The Fluent simulation used the standard k - ϵ turbulence model with scalable wall functions. The SIMPLE algorithm was used with second order spatial discretization for pressure, momentum and energy while first order upwind discretization was applied for turbulent kinetic energy and turbulent dissipation rate. The simulation was run until the residual criteria were below 10^{-4} , which occurred after 1200 iterations (see Figure 5.1-6). The standard k - ω was also tested on the same mesh without the level of convergence obtained by the k - ϵ model.

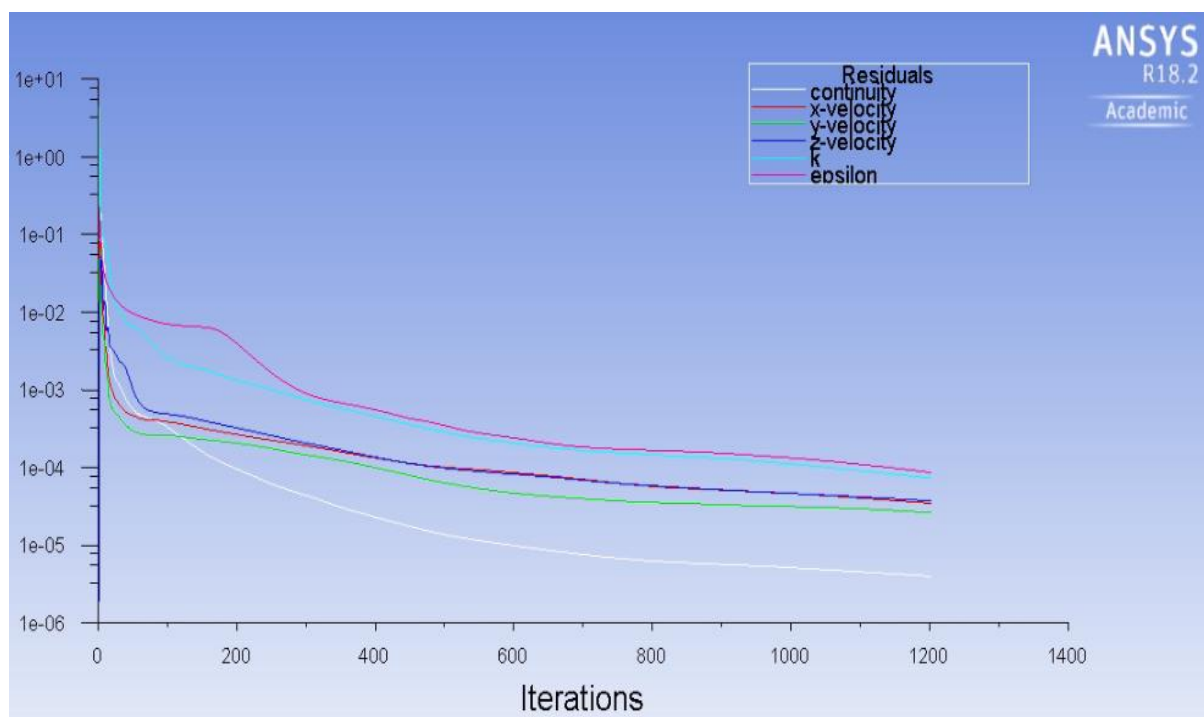


Figure 5.1-6. Residual criteria for the isothermal simulation of the water region (k - ϵ model).

A mass flow rate was specified at the inlet corresponding to the mass flow rates obtained in the experimental cases by Lisowski (see Table 4-2). A pressure of 113 kPa was specified at the outlet, which assumed that the level in the water tank was at 80%.

The density and viscosity of water were taken as constant at 998.2 kg/m³ and 0.00103 kg/m-s respectively.

Table 5-1 shows the inlet pressure and pressure drop that was calculated for a given total mass flow rate. Also shown is the mass flow rate division between the riser tubes that was obtained by the simulations.

Table 5-1. Pressure drop and mass flow rate division within the water region.

Mass flow rate (kg/s)	Pressure at inlet (Pa)	Pressure drop (Pa)	Mass flow rate (Riser 1) (kg/s)	Mass flow rate (Riser 2) (kg/s)	Mass flow rate (Riser 3) (kg/s)
0.37	113 008	8	0.1227	0.1231	0.1242
0.45	113 012	12	0.1490	0.1499	0.1511
0.52	113 016	16	0.1723	0.1732	0.1745
0.58	113 019	19	0.1927	0.1935	0.1938
0.67	113 025	25	0.2228	0.2227	0.2245

Table 5-1 shows that pressure drop within the water region increased with an increase in mass flow rate. The results represent the total pressure losses caused by frictional forces and form (minor) losses. The surface roughness in Fluent was calculated by using the standard wall roughness model, with a roughness height of 0 m and roughness constant of 0.5. The mass flow rate found in Riser 1 (see Figure 5.1-5) was the lowest and Riser 3 had the highest mass flow rate for all the cases. The difference between the maximum and minimum mass flow rates were within approximately 1.5%. From the results in Table 5-1 the conclusion can be drawn that the resistance associated with Riser 3 was the least.

Appendix F shows an article written on an investigation on the secondary loss k-factors associated with various parts of the system. A 1D model was set up by using Flownex and the model comprised of the entire loop (downcomer, heated section, chimney and coolant tank). The article details different parts of the system where the k-factors could be lumped and still provide an accurate solution. The investigation also showed a dependence of the k-factors on the mass flow rate and heat input. The values given in Lisowski (2013) are representative for the case with the lowest heat input (of the 5 cases done by Lisowski (2013)). The k-factors for the other cases had to be calculated. This was done by adjusting the k-value used by Flownex until the mass flow rate corresponded with the mass flow rates measured during experiments.

In summary, the results indicate that a single set of data (of loss factors) is possibly not sufficient to characterize the facility. This should be investigated in more detail but falls outside the scope of the thesis.

5.1.4 Mesh dependence and verification study

The geometry that was used in the CFD models is shown in Figure 5.1-7. The positions of the front cavity, back cavity, inlets, outlets, fins and heater wall are indicated in the figure. The lateral line probe from which the temperatures were extracted is also shown. The probe was placed at the middle elevation of the heated section originating on the heater wall, intersecting the front cavity, middle riser tube and back cavity ending on the back wall.

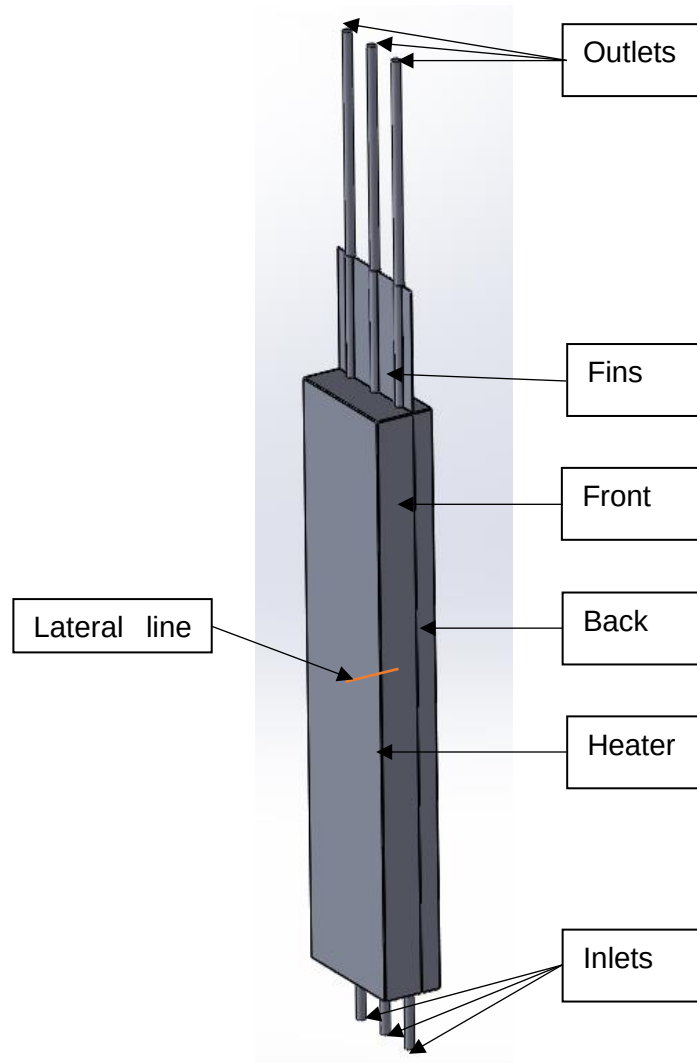


Figure 5.1-7. Geometry used in the CFD model.

The solution obtained by Fluent and Star-CCM+ had to be confirmed to be independent of the mesh size. Three different mesh sizes were investigated with progressively more elements. Additionally, by comparing the results obtained by Fluent and Star-CCM+ it was also aimed to verify the codes against each other.

Ansys Fluent was used to generate the mesh, which was then also imported into Star-CCM+. The comparison between the two sets of results would then be meaningful, since the mesh and boundary conditions would be the same. Table 5-2 shows the properties of the three meshes that were tested. The mesh quality was judged based on the face angle, orthogonal quality and aspect ratio as defined by the Fluent Meshing user's guide (Ansys, 2017b). The quality was within the acceptable range for all three of the tested meshes. The meshes' base size and the number of prism layers was varied to generate the three meshes for testing. The face size was therefore varied from 7 mm to 5 mm to 3 mm, while the number of prism layers were increased from 5 to 8 to 11.

Table 5-2. Mesh dependence study properties.

	Mesh 1	Mesh 2	Mesh 3
Maximum face size (mm)	7	5	3
Max tet size (mm)	7	5	3
Max prism layers	5	8	11
Prism layer transition ratio	0.5	0.5	0.5
Elements ($\times 10^6$)	3.548028	9.866360	42.074876
Minimum orthogonal quality	0.492333	0.59515	0.70148
Average orthogonal quality [▲]	0.94992	0.95002	0.95833
Average skewness [†]	0.1574	0.15664	0.13156
Maximum aspect ratio [*]	6.5038	9.9895	10.132

[†]Acceptable skewness < 0.5 (Ansys, 2017b)

^{*}Acceptable aspect ratio < 100 (Ansys, 2017b).

[▲]Acceptable orthogonal quality > 0.333 (Ansys, 2017b).

The SIMPLE algorithm was used within Fluent, with surface-to-surface radiation, gravity enabled and the SST k- ω turbulence model with low Reynolds number corrections. Star-CCM+ used the segregated flow solver, also with the SST k- ω with low Reynolds corrections.

The boundary conditions were as follow: velocity inlets with velocity fixed at 0.0958 m/s, inlet temperature fixed at 303.15 K and outlet pressure at 101300 Pa. A flux of 5748 W/m² was specified on the heater wall. The boundary conditions reflect Run009 of Table 4-2.

Figure 5.1-8 shows a comparison between the temperatures obtained by the two codes on the three different meshes. The temperatures were taken on a line probe that originated on the heater wall (at -0.243 m in the figure) and passed through the cavity and the middle riser, followed by the back cavity and ends on the back plate of the back cavity.

The correlation between the temperatures calculated by the two codes and three meshes was good.

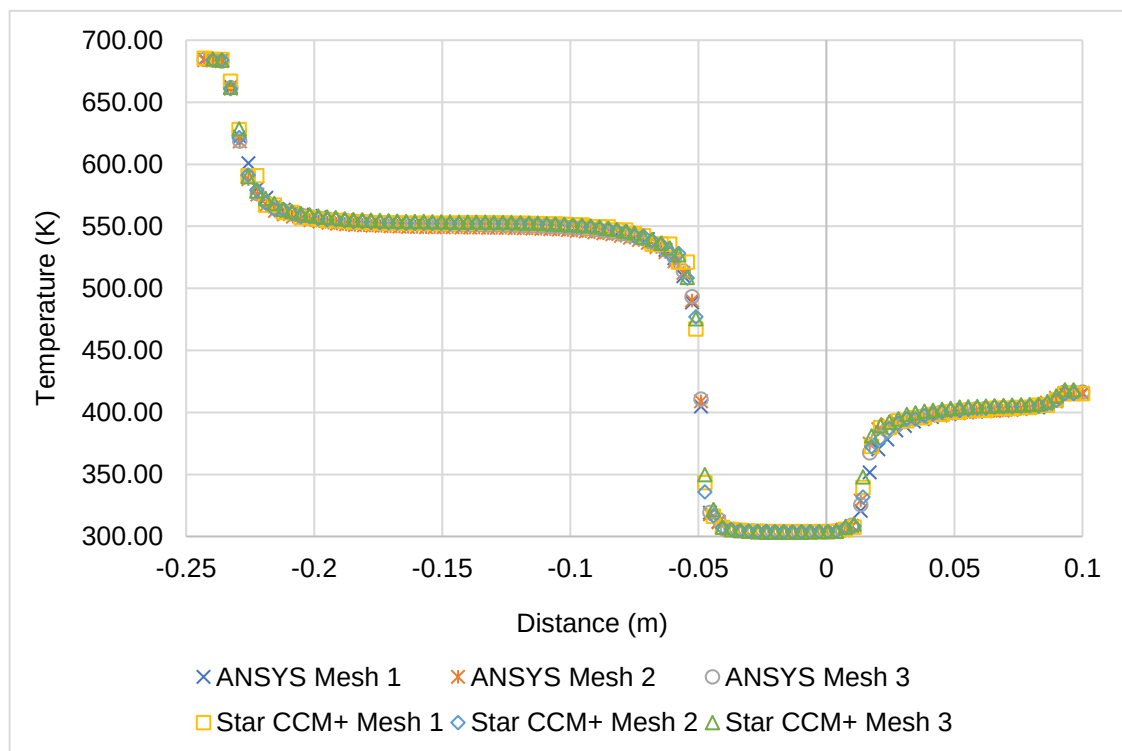


Figure 5.1-8. Lateral temperature distribution in the cavity.

Figure 5.1-8 is enhanced in Figure 5.1-9 through Figure 5.1-13, with a closer view of the near wall regions. The enhanced figures specifically show the temperature profiles next to the heater wall, front and back cavity sides of the riser wall, inside the riser and against the back wall. Error bars have been added to the temperatures calculated by the Fluent mesh 3 configuration. The temperatures were obtained from a line probe that sampled 50 discrete points. This resolution provided a general view of the trends of the temperature profiles. The locations shown don't match exactly, since each code and each mesh spaced the points slightly differently.

Figure 5.1-9 shows the high temperature of the heater wall (685 K) followed by the sudden drop to 550 K of the air in the front cavity. The error bars show 5% variation from the Fluent Mesh 3 results. The error bars indicate the good agreement between the results with most temperatures falling well within the 5% margin.

The cooling effect of the riser tube is evident in the cooling from 550 K to 470 K (Figure 5.1-10) in the vicinity of the front wall of the riser tube. The error bars in Figure 5.1-10 show a 3% variation from the results calculated from Fluent Mesh 3. The results show good agreement as the variations were largely within this range.

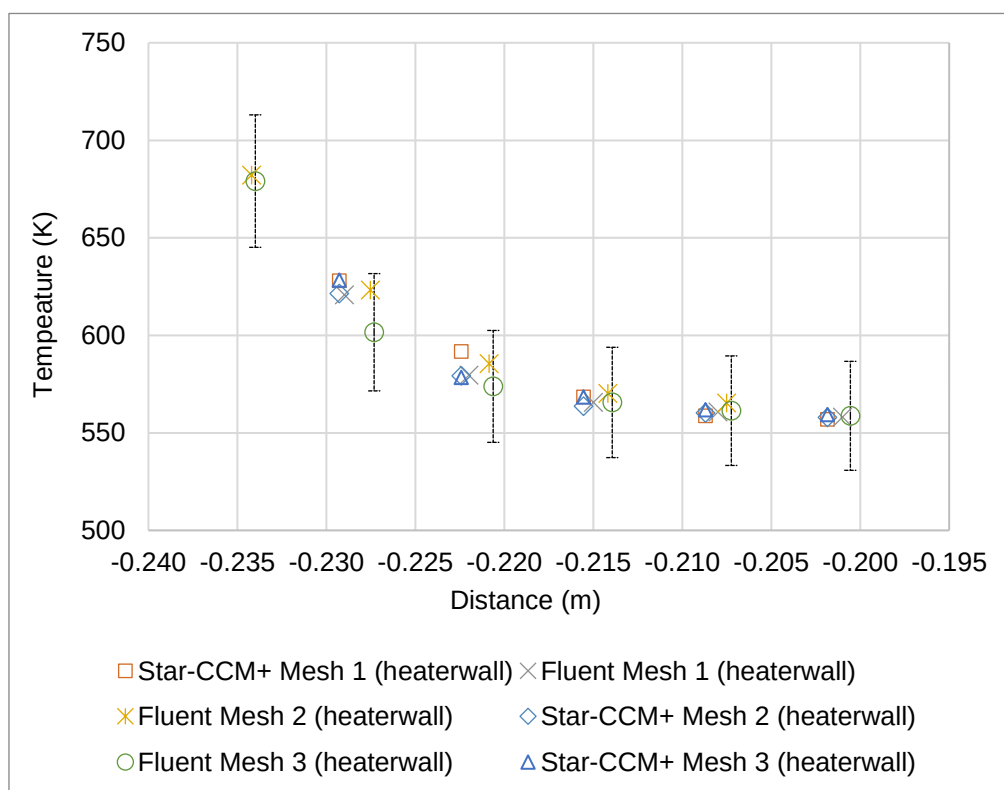


Figure 5.1-9. Comparison of the temperature profile close to the heaterwall.

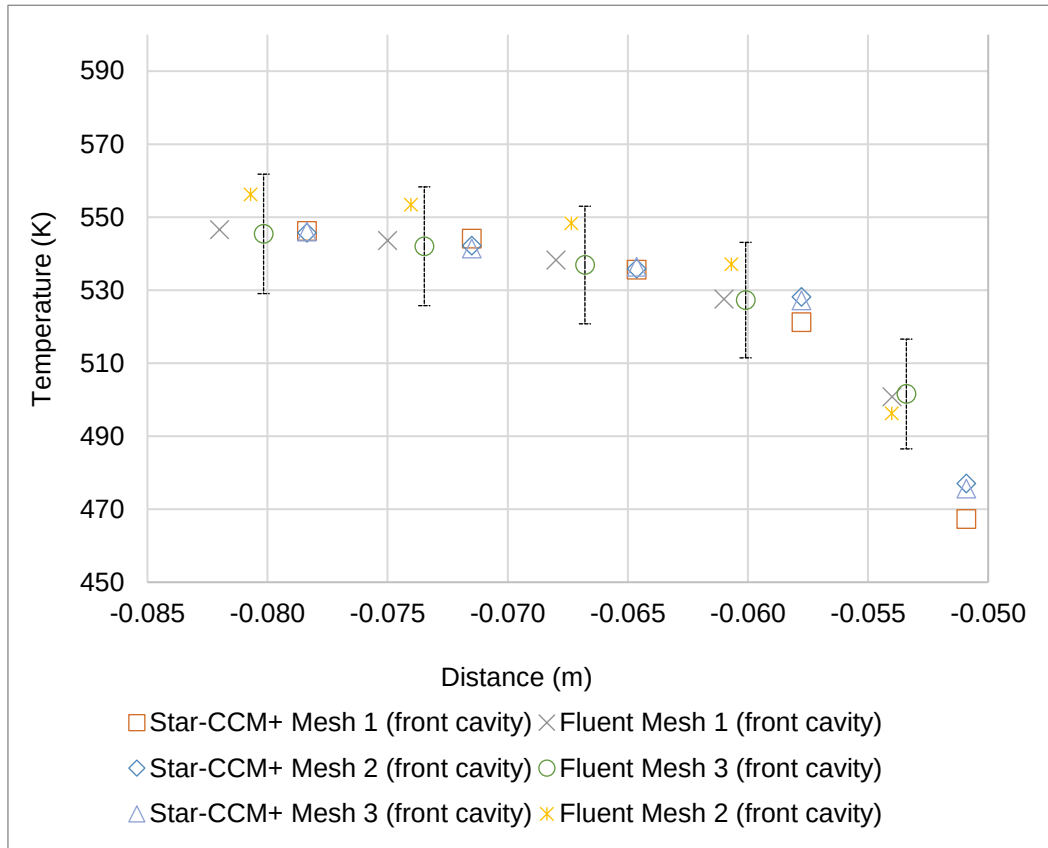


Figure 5.1-10. The comparison between the temperature profiles in the front cavity against the middle riser's front wall.

Figure 5.1-11 shows the temperature distributions across the riser tube. The temperature was noticeably higher on the front side, followed by the back side, with a uniform temperature distribution in the bulk fluid region of the riser tube. The errors bars indicate 1% variation from the results calculated from Fluent Mesh 3. The temperatures show very good agreement as the temperatures were all well within 1% of each other.

Figure 5.1-12 shows the temperature distributions in the back cavity next to the riser tube wall. The error bars in Figure 5.1-12 showed a 5% variance from the Fluent Mesh 3 results. The temperatures are in good agreement.

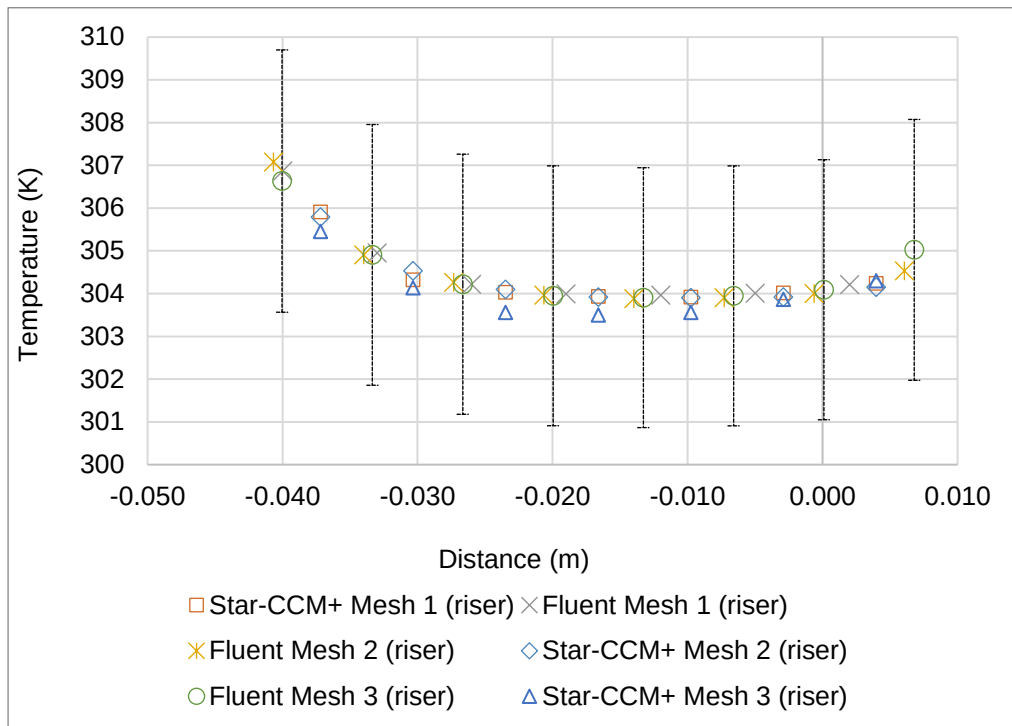


Figure 5.1-11. The comparison between the temperature profiles on the inside of the middle riser.

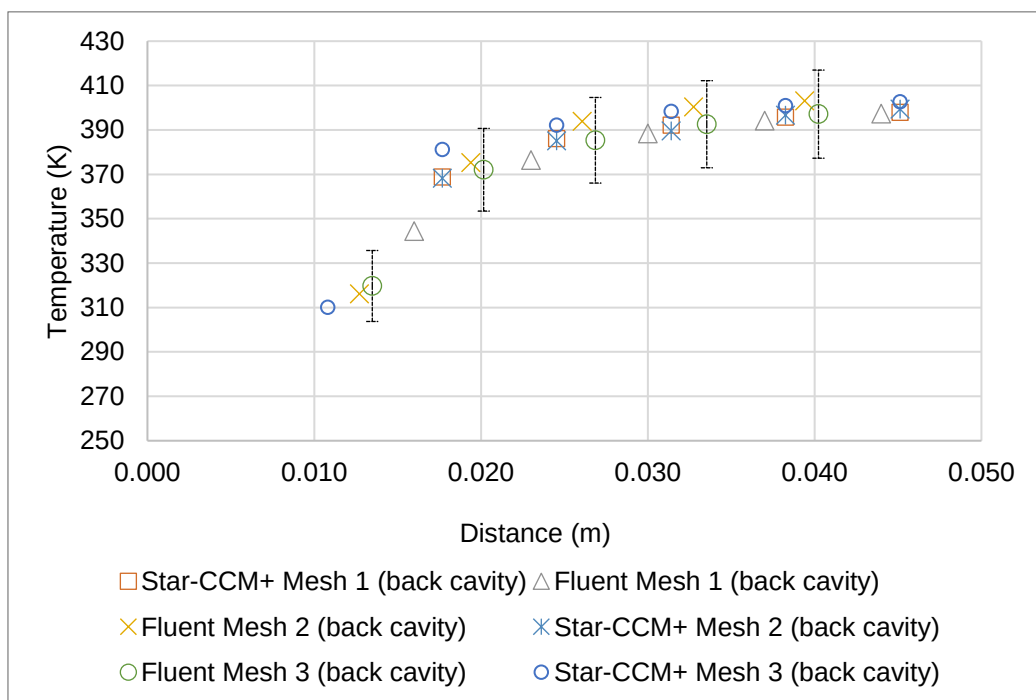


Figure 5.1-12. Comparison between the temperature profiles in the back cavity against the middle riser's back wall.

Figure 5.1-13 shows the temperature profiles against the back wall. The error bars show a variation of 1 % with the results obtained for Fluent Mesh 3. The temperature profiles show good agreement.

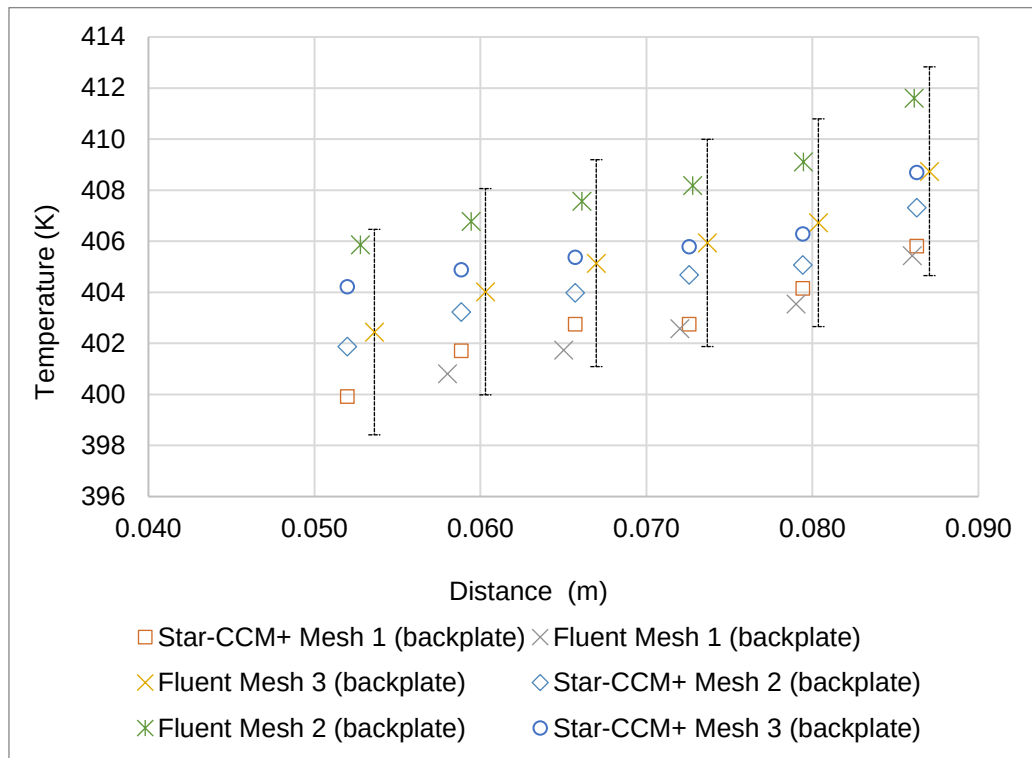


Figure 5.1-13. Comparison between the temperature profiles in the back cavity against the back wall.

In conclusion, the temperature distributions were all within 5% of each other. It was decided to use the intermediate mesh, Mesh 2, which provided enough detail while having reasonable computational time and resources.

5.1.5 k- ϵ turbulence model

The turbulence models used by other authors were discussed in Section 2.5. The SST k- ω model was used by Keshmiri (2014), Ji (2014) and Pal et al. (2016). The k- ϵ mode was used by Ampofo and Karayiannis (2003), Keshmiri (2014), Ji (2014) and Lomperski et al. (2011). It was therefore deemed necessary to test the k- ϵ model as it was just as prevalent in the literature as the SST k- ω model.

The model's sensitivity to differences in the turbulence models was also tested. The k- ϵ model with the two layer turbulence model was tested for this purpose. This turbulence model uses the low y^+ model for $y^+ < 5$ and high y^+ for the $y^+ > 30$. The test was conducted by using Star-CCM+ and the resulting temperature distributions are shown in Figure 5.1-14. The same mesh was used as with the case with circular tubes with fins (see Section 6.2.1). Convergence was assumed after the residual criteria was below 10^{-4} and a monitored heat flux stabilized.

The k- ϵ model was tested by using Fluent with scalable wall functions and showed sufficient convergence (residuals $< 10^{-4}$), but the wall y^+ values were between 5 and 30. This indicated that the boundary layer wasn't properly solved by the turbulence model. The same mesh when used in Star-CCM+ appeared to return lower y^+ values than when used in Fluent.

According to Siemens PLM Software (2019) the low y^+ model, high y^+ and all y^+ models are appropriate combinations with the SST k- ω and standard k- ω models when using Star-CCM+. The different k- ϵ models each has a recommended wall treatment methodology. The standard k- ϵ requires the high y^+ , while the realizable k- ϵ requires all y^+ and the standard k- ϵ low-Re requires low y^+ or all y^+ (Siemens PLM Software, 2019).

The low y^+ model requires the y^+ value should be ~ 1 , the high y^+ model requires $y^+ > 30$ and all y^+ requires $y^+ < 1$ or $y^+ > 30$ (Siemens PLM Software, 2019).

According to (Ansys, 2017a) the k- ω formulation is y^+ insensitive. This means that the k- ω formulation was preferable in terms of obtaining appropriate y^+ values. The “enhanced wall treatment” model is used for the k- ϵ models and requires $y^+ \sim 1$ (Ansys, 2017a). The “scalable wall functions” methodology could also be used and requires $y^+ < 11$, while standard wall functions requires $y^+ > 11$ (Ansys, 2017a).

Figure 5.1-14 shows the comparison between the temperature distributions in Riser 2's wall predicted by the different turbulence models as calculated by using Star-CCM+. Error bars were added to indicate percentage differences between the temperatures calculated by the different models. The left and right side walls had the highest temperatures due to the

conduction from the fins. The temperatures of the side walls all fall within a 1% margin of each other. The back wall temperatures were the lowest, with the different models predicting distributions within 0.5% of each other. The front wall temperature distributions were also found to be within 0.5%.

The $k-\varepsilon$ model showed a higher sensitivity to local effects, which is visible in the oscillatory behaviour in the temperature distributions. The most notable examples were the side and front wall temperatures. The local effects stabilized however and showed smaller oscillations above the elevation of 2.5 m.

Based on the results obtained it was decided to use the SST $k-\omega$ model for subsequent simulations.

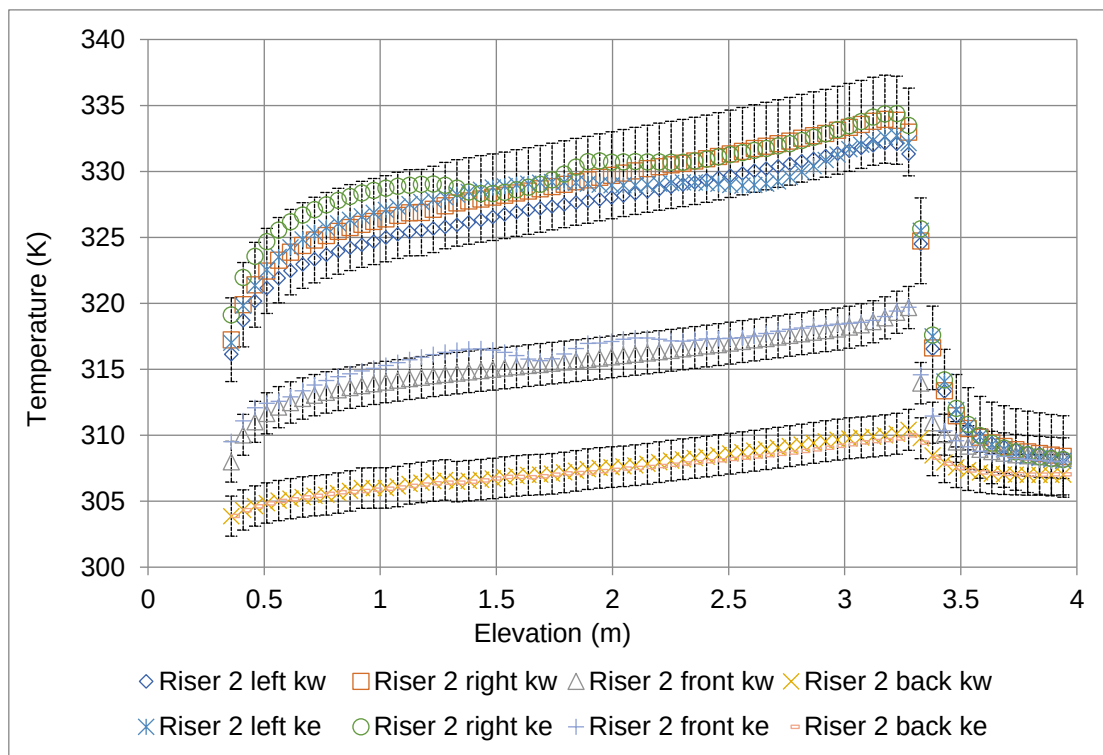


Figure 5.1-14. The riser tube wall temperatures as obtained by using the $k-\varepsilon$ turbulence model.

In conclusion the results compare reasonably well between the two turbulence models, with the corresponding distributions being within 1% of each other.

5.1.6 Fluent vs. Star-CCM+ 3D simulation

The 3D simulations reported in Chapter 6 were done by using Fluent to create a mesh but were simulated by using Star-CCM+. This was done to complete the mesh dependence study where the results from Star-CCM+ and Fluent were compared for the exact same mesh for verification purposes. Subsequent simulations therefore used meshes that were subject to the verification.

For further verification purposes, one of the cases that was presented in Section 8.2 was done in which Fluent was used for both meshing and simulation. The boundary conditions were in other words identical to the conditions specified in Section 6.2.2. The mesh was identical to the mesh used for Section 7.4.

Figure 5.1-15 shows the wall temperatures in the middle riser tube, as sampled on a line probe in the middle of the wall, at four locations. Temperatures obtained by using Fluent are compared with the temperatures obtained by using Star-CCM+ (which is presented in Section 8.2.2.2).

The temperatures of the side walls (in Figure 5.1-15) were the highest, owing to the conduction from the fins. The temperatures of the side walls predicted by the codes were approximately within 2% of each other.

The back wall (in Figure 5.1-15) had the lowest temperatures and the corresponding temperatures calculated by the two codes fell within 1% of each other.

The temperatures calculated for the front wall (in Figure 5.1-15) fell between the temperatures of the back wall and side walls. The temperature distributions calculated by the two codes fell within 1% of each other.

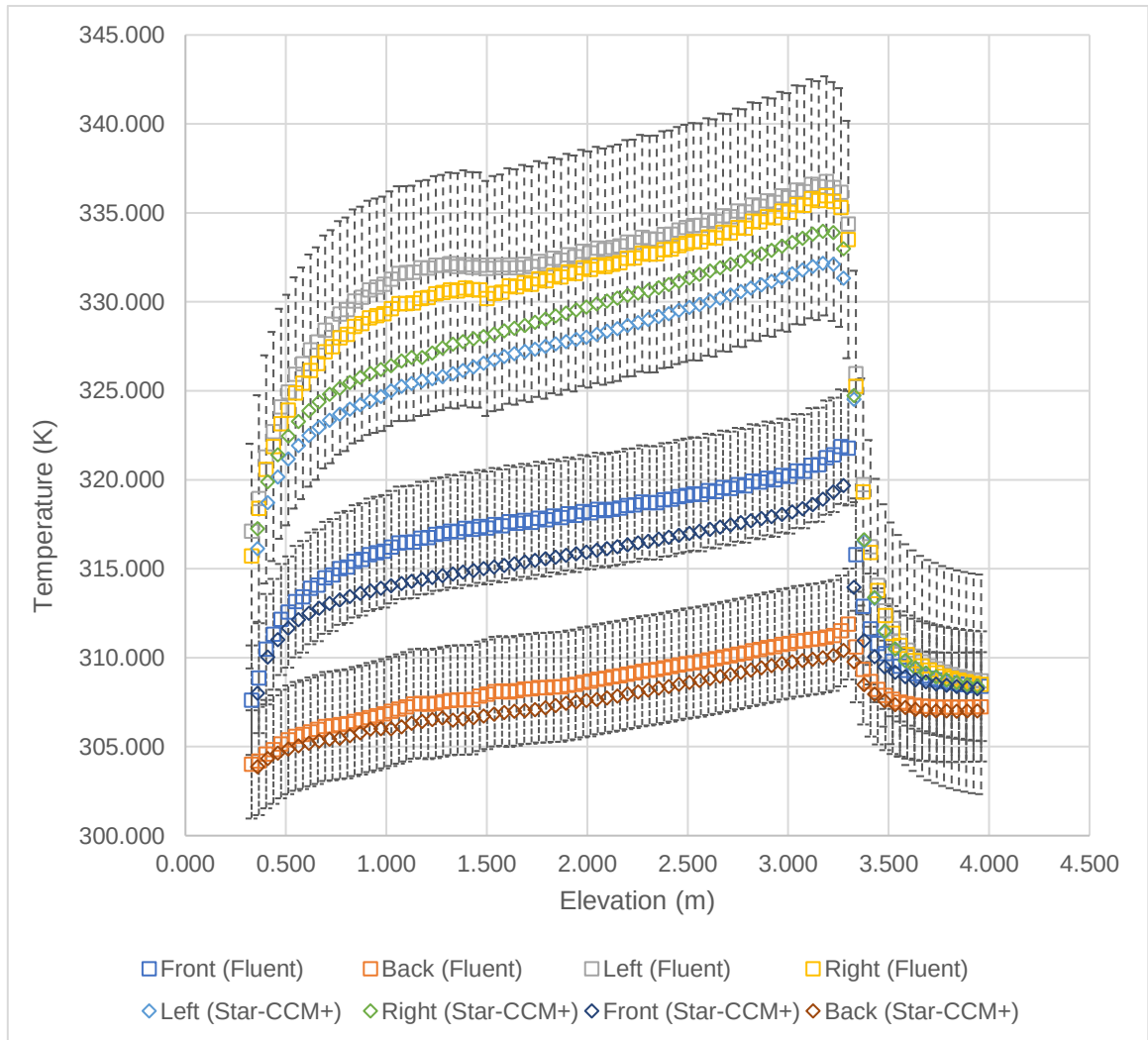


Figure 5.1-15. Riser tube wall temperature comparison between Fluent and Star-CCM+.

In conclusion the results obtained by the Star-CCM+ and Fluent models show good agreement when the same boundary conditions are used. The front and back wall temperature distributions were within 1% of each other and the side walls within 2%. It can be concluded based on the good agreement between the results that the methodology had been verified.

5.1.7 Conclusion

The temperature and velocity profiles in a fluid region, such as a simple cavity, is dependent on a shell region surrounding the fluid region. It is therefore important to include a shell region to obtain more accurate results.

The SST k- ω model was compared with the numerical and experimental work of other authors in simulating natural circulation within a cavity. The methodology applied to natural convection in an enclosed cavity compares reasonably well when compared with the numerical and experimental results from literature. The SST k- ω models were in good agreement with the measured temperatures close to the hot wall but over predicted temperatures further away from the wall.

Simulations were done to determine pressure drops at the various mass flow rates listed in Table 4-2. It was found that the pressure drop increased with an increase in mass flow rate and that appropriate minor loss factors were needed. The investigation shown in Appendix E details the required lumped values for the minor loss factors, as well as where the values could be specified in a 1D model.

A mesh dependency study has been conducted and 3 different meshes have been tested for the UW RCCS facility. A lateral probe was used to measure temperatures in the heated section and the temperatures were compared between the 3 meshes. The corresponding temperatures on the probes compared well with each other, implying mesh independence at the settings used to obtain the meshes. The second mesh was chosen for further simulations, since it provided the best compromise between mesh size and resolution of the results.

The k- ϵ turbulence model was tested by simulating the geometry in Chapter 7 and the results were compared with the corresponding results that were obtained by using the k- ω model. The

temperature distributions around the middle riser tube walls were compared and showed a reasonable degree of agreement between the two different turbulence models.

The models described in Chapter 6 were simulated by using Fluent instead of Star-CCM+ and the results were compared. The temperature distributions around the middle riser tube walls were compared and show reasonable agreement between the two codes.

Based on the cases that were done in this chapter the methodology was assumed to be verified and validated to an acceptable extent.

CHAPTER 6 – 3D CFD MODELLING OF THE FACILITY

6.1 Introduction

The heated section of the facility was simulated by using Star-CCM+ (version 14.02.010) and Ansys Fluent (version 18.2). The mesh dependence study made use of the mesher that is built into Ansys. It was therefore decided to use the same meshing method as in the mesh dependence study. A high-performance computer (HPC) cluster was available and necessary for use during simulations. The machine was maintained by another department and had the latest version of Star-CCM+ before an updated version of Ansys was installed. The simulations were therefore run by using Star-CCM+ but with a mesh imported from Ansys Fluent.

The geometry dependence of the system's performance was investigated by simulating several variations in the design of the riser tubes. The geometries simulated were: 3 circular tubes with fins, 3 circular tubes without fins, 3 square tubes with fins, 3 square tubes without fins, 6 rectangular tubes with fins, 6 rectangular tubes without fins. The square and rectangular geometries were chosen for the simplicity of the cross-sections and since these profiles can be manufactured.

The dimensions of the square and rectangular tubes were calculated such that the cross-sectional areas would be the same as the circular tubes. The square tubes therefore had an inner length and width of 0.04502 m. In the case of the rectangular tubes, a ratio of 5:1 was assumed for the length to the width. This was because a similar ratio was used for the rectangular tubes in the scale model facilities of the NACEF and NSTF discussed earlier (Kim, 2014; Lomperski, 2011). The rectangular tubes therefore had an inner length of 0.07118 m and width of 0.01424 m.

6.2 Model descriptions

6.2.1 Mesh properties

The settings to generate the meshes were similar to that used in the mesh dependence study. For the circular tube cases the maximum face size/tet size was set to 5 mm and the growth rate of 1.2 was chosen. Eight prism layers were specified on the cavity and riser tube surfaces with transition ratio of 0.5 and growth rate of 1.2. For the rectangular tubes the maximum face/'tet' sizes were set at 5 mm, with the sizing set to proximity and curvature. The relevance centre was set to 'fine' and a growth rate of 1.05 was chosen. 16 prism layers were placed on the inner surfaces of the riser tubes, also with a growth factor of 1.05 and a transition ratio of 0.5. On all the surfaces in the cavities a maximum of 8 prism layers were used, also with a growth rate of 1.05 and transition ratio of 0.5. The mesh was made coarser for the outer walls and the riser tube walls above the heated section. For these walls the size was set such that hexahedral blocks with sides of 5 mm length would be formed. This limited the cells in the regions that were deemed less critical.

The quality of the various cases is reported in Table 6-1. The primary measures used were the orthogonal quality, skewness and aspect ratio. The details pertaining to the various meshes is reported in Appendix B. The three parameters were within the acceptable limits and the meshes were thus considered to be of adequate quality.

Table 6-1. Mesh quality for the tested geometry.

	Circular with fins (reference)	Circular without fins	Square tubes with fins	Square tubes without fins	Rectangular tubes with fins	Rectangular tubes without fins
Number of cells	13 882 460	15 790 419	17 639 691	12 389 080	18 904 545	16 798 015
Orthogonal quality	0.93601	0.98049	0.95882	0.9473	0.91232	0.90856

Cell skewness	0.15688	7.4478e-2	0.10646	0.13074	0.19075	0.20167
Aspect ratio	2.5967	2.0174	2.1983	2.3823	4.4132	4.6786

6.2.2 Boundary conditions and solver properties

The boundary conditions used were as follows (Table 6-2):

The outlet pressure was based on the approximated pressure that the system would experience as calculated by the 1D system code, Flownex. The velocity was calculated based on the experimental mass flow rate and the density of water at 30°C and 160 kPa. The heat flux was calculated from converting the heater power from Run009 in Table 4-2. The riser power of Run009 was used which neglect losses to the environment and was converted to a flux representative of the entire wall (as opposed to individual heater elements).

Table 6-2. Boundary conditions for the various tested geometries.

Boundary condition	Value
Inlet velocity (per riser)(m/s)	0.0958
Inlet temperature (K)	303.15
Outlet pressure (gauge) (Pa)	14 000
Heat flux on the heater wall (W/m ²)	4933.31

The outer edges of the geometry were considered adiabatic. The emissivity of the fins and tubes were assumed to be 0.5 and the emissivity of the other surfaces to be 0.9 (Lisowski et al., 2012).

The segregated energy solver in Star-CCM+ was used, with the SST k- ω turbulence model with the low-Reynolds corrections option applied in both the air and water regions. The grey thermal band surface-to-surface radiation model was selected for the air region and gravity was

specified for both the air and water regions. The segregated solid energy model was specified for the steel region.

The segregated energy model's relaxation factors were set to 0.99 for the fluid region and 0.999 for the solid region.

The fluid properties for the air and water were set to be calculated by polynomials as a function of temperature. The polynomials are listed in Appendix A. The specific heat capacity, density and conductivity of steel were specified in Star-CCM+ as the same values of these properties in the libraries of Ansys.

The temperatures were limited as follow: for the air region between 300 K and 900 K and for the water region from 300 K to 369 K. It was of particular importance for stability's sake to limit the water region's temperature as the polynomials would calculate inaccurate values outside of the range. The limit also kept the water from entering two-phase flow, which was outside of the conditions of the experimental run and beyond the scope of this study.

The residuals and a riser tube exit temperature and/or the flux on the middle riser's surface was measured as an indication of the level of convergence. The solutions were considered as converged once the residuals were below a value of 10^{-4} together with an outlet temperature or heat flux which had stabilized. The solution monitoring of the simulations is reported in Appendix C.

6.2.3 Conclusion

The different geometries to be investigated in the 3D CFD simulations have been described. The boundary conditions will remain the same between the models in terms of the type and in terms of the values of the boundary conditions.

The material properties are to be calculated by using polynomials as a function of temperature. The polynomials are listed in Appendix A.

A summary of the mesh qualities has been listed, with the complete details listed in Appendix B

Details of the solver has also been discussed, including the turbulence model, relaxation factors and temperature limits that were used.

The criteria for convergence and monitored variables have also been discussed. The solution monitors for the various simulations are reported in Appendix C.

CHAPTER 7 – COUPLED 1D/3D MODELLING OF THE FACILITY

7.1 Introduction

The coupling of 1D and 3D codes was done to take advantage of the strengths of each respective code. The 1D methodology excels at modelling complete systems but with limited detail, while the 3D methodology excels at predicting phenomena with substantial detail. Modelling a 3D system requires a considerable amount of computational resources and time. Therefore, the 3D methodology was used strategically where the flow field would be complex while the 1D was used to model the rest of the loop. The 3D methodology was thus used to model the heated section of the facility. Appendix D discusses the possible philosophies that were considered as coupling methodologies. The method that was chosen was an indirect coupling method in which the domains were solved in parallel between the 1D and 3D codes. This methodology has the codes operating independently to a large extent, with the added benefit of a feedback loop that iteratively corrects and improves the accuracy of the shared boundary conditions. The coupled simulations were done by using a desktop workstation, since it was easier to couple the codes on a machine running Windows as an operating system (The HPC used for the simulations in Chapter 6 is a Linux machine).

7.2 Model variations

Several variations were possible in terms of the coupling strategy. The pipe network between the manifolds were modelled by using the 3D methodology in Fluent (version 18.2), while the rest of the system was modelled by using the 1D approach of Flownex. The first model variation (Variation B) fixed the mass flow rate on the Flownex side and a flow admittance (described in section 3.4) was calculated. The outlet temperature was transferred as an output from Fluent to Flownex. The second model variation (Variation C) the mass flow rate was not fixed, allowing the fluid flow to develop numerically. The heat flux on the riser tubes' inner surfaces was transferred as an output from Fluent to Flownex. A third variation was tested where the manifolds were also included in the 3D analysis. The manifolds required a substantial number of elements and the model was sensitive to guess values and tended to diverge easily. It was therefore abandoned as the computational efforts were larger than the gained accuracy.

7.3 Variation B model (Manifolds modelled using Flownex)

7.3.1 Flownex model (Variation B)

The implementation of the variation B model in Flownex is shown in Figure 7.3-1. The risers were coupled individually at a point after the inlet manifold and at a point before the outlet manifold. The server link transferred three different inlet mass flow rates, inlet temperatures and outlet pressures. The boundary conditions are shown in Table 7-1. The free surface in the tank (top node on pipe 31 in Figure 7.3-1) was assumed to be atmospheric with the temperature (on the bottom node of pipe 33 in Figure 7.3-1) and mass flow rate (on pipe component 2 in Figure 7.3-1) specified as found during experiments.

Table 7-1. Boundary conditions of Variation B.

Parameter	Value
Tank free surface pressure (kPa)	101.325
Tank outlet temperature (°C)	30
Mass flow rate	0.58 kg/s

The pipe roughness was assumed as 15 μm Munson et al. (2006) and the secondary losses were accounted for by specifying the k-values given in Table 7-2. The dimensions were obtained from engineering drawings by Lisowski (2013).

Table 7-2. k-values specified to account for secondary losses (variation B).

Identifier in Figure 7.3-1	Name	k-forward; k-reverse	Length (m)	Elevation in; Elevation out (m)
1	Inlet to downcomer (outlet of tank)	0.5; 1	0.145	5.253; 5.108

2	Vertical downcomer pipe	0.3; 0.3	4.956	5.108; 0.152
3	90° bend (vertical to horizontal)	0.25; 0.25	0.239	0.152; 0
4	Horizontal down pipe	0; 0	0.328	0; 0
5	Horizontal bend in the down pipe	0.25; 0.25	0.239	0; 0
6	Venturi inlet	0.130503; 0.182124	0.102	0; 0
7	Venturi flow meter main part	0; 0	0.481	0; 0
8	Venturi outlet	0.182124; 0.130503	0.102	0; 0
9	Horizontal bend in the down pipe	0.25; 0.25	0.12	0; 0
10	Inlet to manifold	0.3; 0.3	0.145	0; 0
11, 12	Lower manifold horizontal	0.3; 0.3	0.24	0; 0
13, 14, 15	Lower manifold vertical inlet pipes	2; 2	0.173	0; 0.173
16, 17, 18	Total riser length	N/A – calculated by codes	5.173	0.173; 5.346
19, 20, 21	Upper manifold vertical pipes	2; 2	0.166	5.346; 5.512
22, 23	Upper manifold horizontal pipes	0.3; 0.3	0.24	5.512; 5.512
24	Upper manifold outlet to chimney	0.3; 0.3	0.515	5.512; 5.512

25	Chimney 90° horizontal bend	0.25; 0.25	0.239	5.512; 5.512
26	Chimney horizontal pipe	0; 0	0.679	5.512; 5.512
27	Chimney 90° bend from horizontal to vertical	0.25; 0.25	0.239	5.512; 5.665
28	Chimney vertical pipe	0; 0	0.268	5.665; 5.932
29	90° bend from vertical to horizontal	0.25; 0.25	0.239	5.932; 6.085
30	Chimney horizontal to tank	1; 1	0.353	6.085; 6.085
31	Tank upper reservoir	0; 0	0.724	6.809; 6.085
32	Tank middle reservoir	0; 0	0.254	6.085; 5.831
33	Tank lower reservoir	0; 0	0.578	5.831; 5.253

The black arrows in Figure 7.3-1 show the transfer of the mass flow rate, inlet temperature and outlet pressure to a scripting component. The scripting component rounds the values down to a chosen number of decimal places before sending the data to Fluent (red arrow). The blue arrows transfer the flow admittance to the resistive components 16, 17 and 18. The inlet pressure sent to Flownex from Fluent was used along with the given outlet pressure. The pressure drop across the component, excluding the pressure drop from a change in elevation, is then used to calculate the admittance. The admittance therefore represents pressure drops caused by frictional and secondary losses.

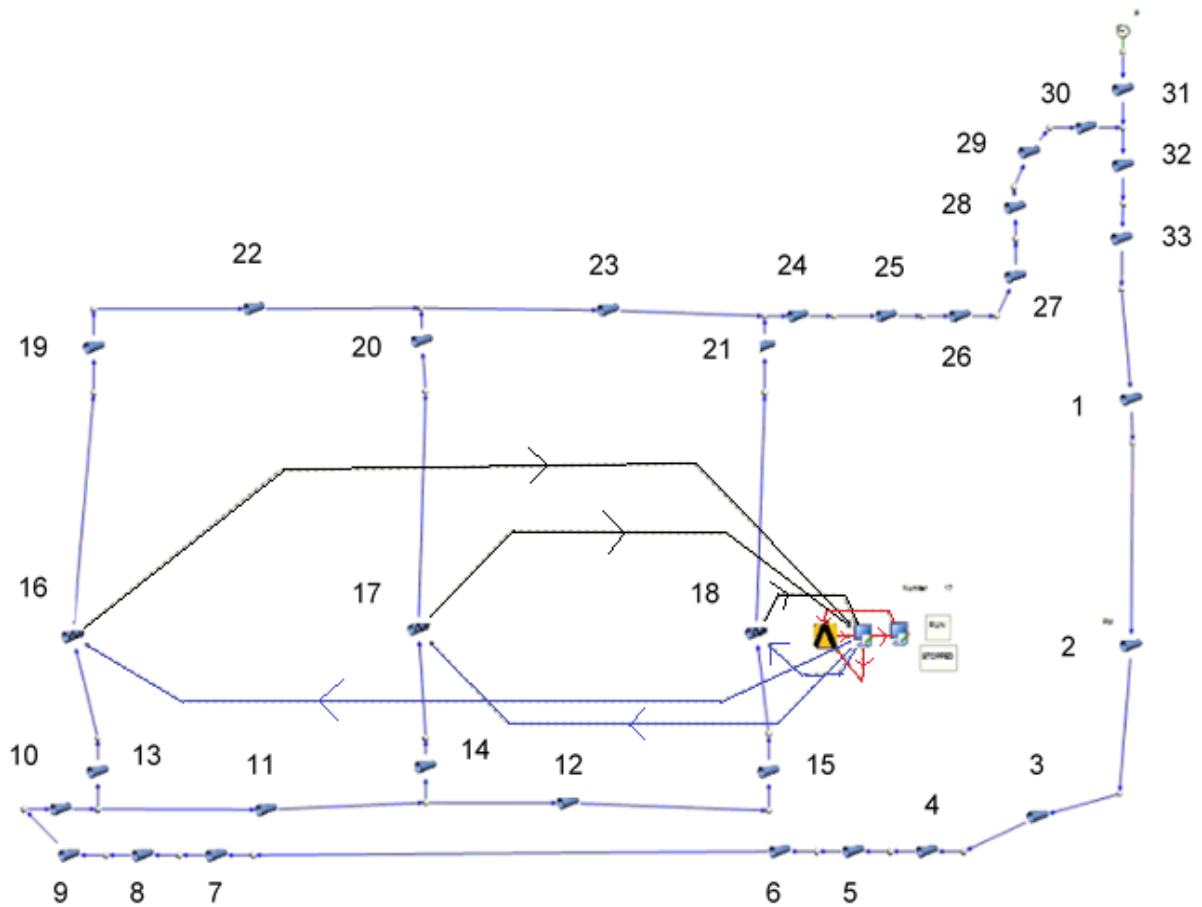


Figure 7.3-1. The variation B Flownex model.

The other scripting component analyses the difference between the calculated inlet pressures between the codes and sends a run command until the convergence criteria are met (relative error between inlet pressures < 0.1). Also set by this component was the number of iterations executed by Fluent per iterations performed by Flownex. Better stability was obtained when Fluent initially ran at least 500 iterations before transferring data between codes. After the initial 500 iterations data could be transferred at 100 iteration intervals.

Rounding functions were used on the parameters that were transferred to reduce the instability caused by small oscillations in the transferred boundary conditions. The outlet pressure was rounded to 0 decimal places (to provide 6 significant figures), the inlet temperature to 2 decimal places (provides 6 significant figures) and the inlet mass flow rate to 4 decimal places (provides 4 significant figures). The rounding was done to provide at least 4 significant figures.

7.3.2 Fluent model (Variation B)

The Fluent geometry that was used was the same as the geometry used in the 3D CFD case with round tubes with fins (shown in Figure 5.1-7).

The SIMPLE pressure-velocity coupling scheme solver was used by Fluent. The spatial discretization chosen was second order for pressure and second order upwind for momentum and energy, while first order upwind was used for turbulent kinetic energy and specific dissipation rate. The relaxation parameters were as follow: 0.99 for energy, 0.9 for body forces and density, 0.05 for momentum, 0.8 for the turbulent kinetic energy and specific dissipation rate, 0.3 for pressure and 1 for turbulent viscosity.

The models used by Fluent were energy, the SST $k-\omega$ turbulence model, low Reynolds corrections and the surface-to-surface radiation model.

The temperature was limited between 300 K and 900 K as previous simulations showed the temperatures were within this range.

The material properties were specified for air and water in terms of a polynomial as a function of temperature. The polynomials used for the fluid properties are given in Appendix A.

The boundary inlet mass flow rate, inlet temperature and outlet pressure were transferred from Flownex. The only boundary condition specified within Fluent was the heat flux on the heater wall. The value specified was 4933.31 W/m^2 which was calculated as described in Section 6.2.2.

The temperature at the outlet of the riser tubes were subsequently calculated and transferred to Flownex.

7.4 Variation C model (natural circulation)

In the Variation B model, the mass flow rate was specified on the Flownex side of the coupled simulation. In Variation C the mass flow rate was not specified, so that it would be calculated by the coupled codes.

7.4.1 Flownex model (Variation C)

Figure 7.4-1 shows the implementation of the model by using Flownex. The model was very similar to the model shown in Figure 7.3-1 of Variation B. The 'flow resistance' elements were replaced with the appropriate pipe elements in accordance with the geometry of the risers and the heat energy was transferred from Fluent instead of the outlet temperature. The heat energy calculated by using Fluent was applied to the pipe components numbered 19, 20, 21 in Figure 7.4-1.

The arrows in Figure 7.4-1 show the direction of the data transfer between the codes. The scripting component on the left that received the data performs a rounding operation for the sake of stability. The rounding function rounded the outlet pressure and inlet temperature to 4 decimal places. Additionally, the mass flow rate was rounded to 6 decimal places, which resulted in at least 6 significant figures for the transferred parameters.

A limit had to be placed on the fixed heat transfer in Flownex that was transferred from Fluent. Assuming equal distribution of heat energy each riser should receive approximately 3548 W. The heat energy added by Fluent was checked by a scripting component to ensure a minimum value of 3400 W. If the value was lower Flownex would specify the value of 3400 W to be used. Enforcing the limit improved the stability, since the heat energy approached the final value only after approximately 2000 iterations.

Without the limiter heat energies much lower than the final value would be used within Flownex, resulting in Flownex calculating much lower mass flow rates and flow reversals where none should occur. The low mass flow rates or reversals were in turn transferred to Fluent, causing the solution to diverge.

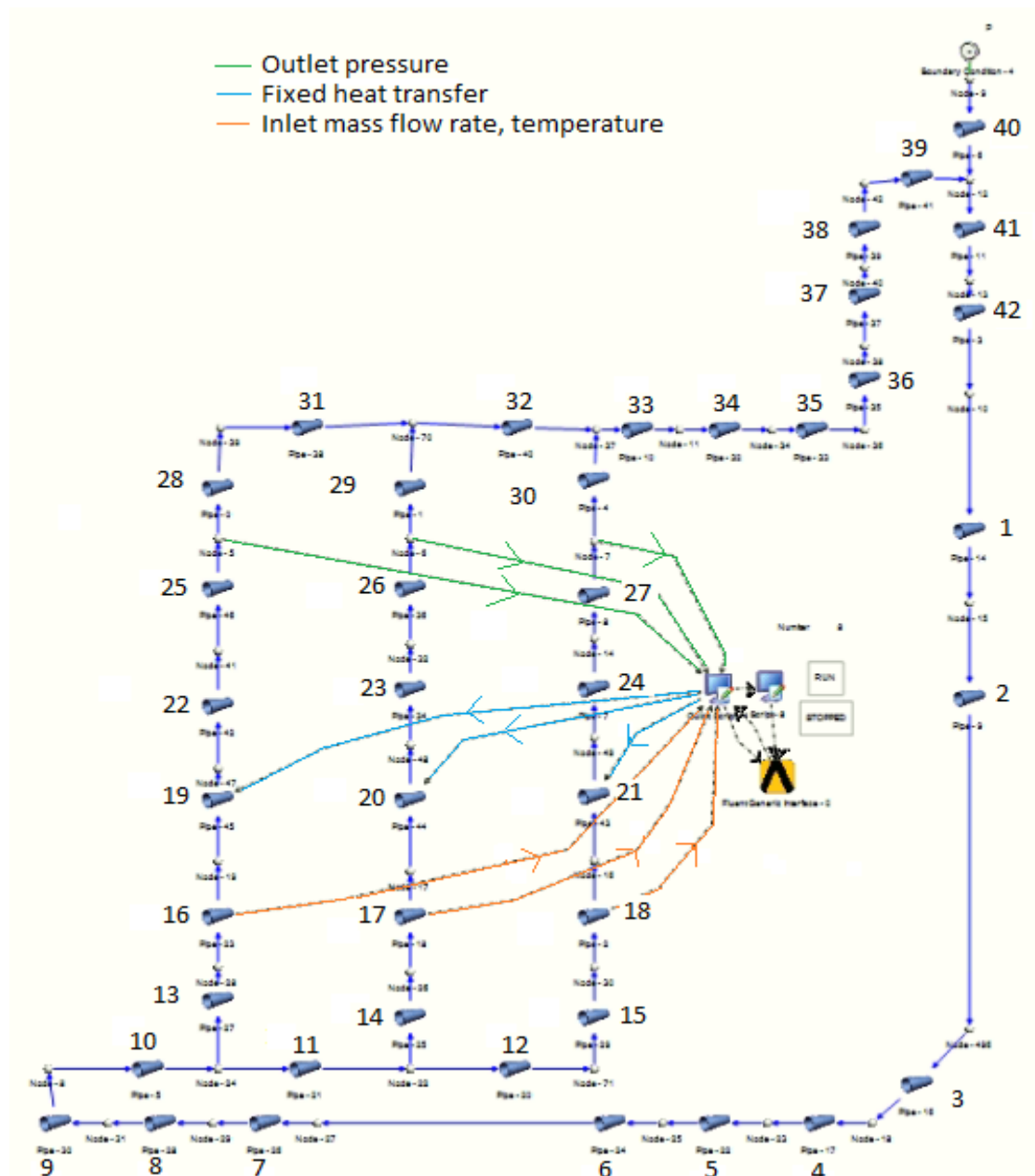


Figure 7.4-1. Flownex model of the Variation C model.

The boundary conditions specified in the Flownex model were the free surface pressure in the tank (101.325 kPa) and the tank's outlet temperature (at 30°C, at number 41 in the figure).

The pipe lengths, k-factors and node elevations are given in Table 7-3. The geometric data were obtained from Lisowski (2013) and the k-values were estimated from Munson et al. (2006). The k-value however had to be calibrated for the mass flow rate, as the study shown in Appendix E and section 5.1.3 showed the k-values' dependence on the mass flow rate. The

value associated with the horizontal pipe at the tank inlet (33 in Table 7-3) was decreased from 1 to 0.3.

Table 7-3. Pipe lengths, k-values and node elevations for Variation C.

Identifier in Figure 7.4-1	Location	k-forward value; k-reverse value	Length (m)	Elevation in ; elevation out (m)	Diameter (m)
1	Tank outlet	0.5 ; 1	0.145	5.253 ; 5.108	0.102
2	Vertical downcomer	0.3 ; 0.3	4.956	5.108 ; 0.152	0.102
3	Vertical pipe bend	0.25 ; 0.25	0.239	0.152 ; 0	0.102
4	Horizontal bottom pipe	0 ; 0	0.328	0 ; 0	0.102
5	Horizontal bottom pipe bend	0.25 ; 0.25	0.239	0 ; 0	0.102
6	Venturi inlet	0.131 ; 0.182	0.102	0 ; 0	0.102
7	Venturi	0 ; 0	0.481	0 ; 0	0.077
8	Venturi outlet	0.182 ; 0.130	0.102	0 ; 0	0.102
9	Horizontal bottom pipe bend 2	0.25 ; 0.25	0.120	0 ; 0	0.102
10	Bottom manifold (inlet)	0.3 ; 0.3	0.26	0 ; 0	0.102
11,12	Bottom manifold	0.3 ; 0.3	0.24	0 ; 0	0.102

	(horizontal)				
13, 14, 15	Bottom manifold (vertical)	2 ; 2	0.173	0 ; 0.173	0.0508
16, 17, 18	Risers' inlet length	0 ; 0	0.328	0.173; 0.501	0.0508
19, 20, 21	Risers' heated length	0 ; 0	3.01	0.501; 3.511	0.0508
22, 23, 24	Risers' unheated (finned)	0 ; 0	0.622	3.511; 4.133	0.0508
25, 26, 27	Risers' unheated length	0; 0	1.213	4.133; 5.346	0.0508
28, 29, 30	Top manifold (vertical)	2 ; 2	0.166	5.346 ; 5.512	0.0508
31, 32	Top manifold (horizontal)	0.3 ,0.3	0.24	5.512; 5.512	0.0508
33	Top manifold (outlet)	0.3	0.515	5.512; 5.512	0.0508
34	Chimney's horizontal bend 1	0.25 ;0.25	0.239	5.512 ; 5.512	0.102
35	Chimney's horizontal 1	0 ; 0	0.679	5.512 ; 5.512	0.102
36	Chimney's vertical bend	0.25 ; 0.25	0.239	5.512 ; 5.665	0.102
37	Chimney's vertical pipe	0 ; 0	0.268	5.665 ; 5.932	0.102
38	Chimney's vertical bend 2	0.25 ; 0.25	0.239	5.932 ; 6.085	0.102

39	Chimney's horizontal 2 (tank inlet)	1 ; 1	0.353	6.085 ; 6.085	0.102
40	Tank upper	0 ; 0	0.724	6.809 ; 6.085	0.914
41	Tank middle	0 ; 0	0.254	6.085 ; 5.831	0.914
42	Tank lower	0 ; 0	0.578	5.831 ; 5.253	0.914

7.4.2 Fluent model (Variation C)

The Fluent model used the SIMPLE algorithm with second order upwind spatial discretization for momentum, energy, turbulent kinetic energy and turbulent dissipation rate. Body forced weighted discretization was used for pressure. The SST k- ω turbulence model was used with low Reynolds corrections. The surface-to-surface thermal radiation model was used and gravity was activated.

The fluid properties were calculated as polynomials as a function of temperature (the details are shown in Appendix A).

The under-relaxation factor for energy was set to 0.9 for the sake of stability. The factors for density and body forces were decreased to 0.9. The factor for momentum was gradually decreased throughout the simulation to a minimum value of 0.05. Limits were imposed on the minimum and maximum allowable temperatures and pressures for the sake of stability.

The heater wall's flux was specified as a boundary condition within Fluent, while the other boundary conditions were transferred from Flownex. These boundary conditions were the inlet mass flow rate, inlet temperature and outlet pressure.

The fins and tube surfaces were assigned an emissivity of 0.5 and all other surfaces an emissivity of 0.9.

7.5 Conclusion

The models that were simulated has been discussed, including the 3D as well as the 1D/3D models. Engineering drawings from Lisowski (2013) were used to derive the geometry for the 1D and 3D models. The minor loss factors were subsequently derived from the geometry and was subject to isothermal simulations in 1D and 3D to investigate the dependencies. The minor loss analyses are shown in section 5.1.3 and also in Appendix F. The simulation of the buoyancy forces necessitates variable fluid properties. To that end, polynomials were fitted to property curves which were used to calculate fluid properties as a function of temperature. The full details of these polynomials are shown in Appendix A. The boundary conditions have been discussed. A heat flux boundary conditions has been used to emulate the heater wall. Inlet mass flow rates and outlet pressures have been prescribed for the coupled modes and inlet velocities and outlet pressures have been prescribed for the 3D models. The values of the boundary conditions were based on Run009 done by Lisowski (2013) (see Table 4-2). The modelling of thermal radiation in the cavity of the heater box would be accounted for by surface-to-surface radiation and the SST k- ω model would be used to simulate turbulence.

CHAPTER 8 RESULTS

8.1 Introduction

The results of the simulations previously discussed are presented in this chapter.

Results are discussed first for the 3D simulations that were done using Star-CCM+. The results are divided into three sections, namely, the heat transferred to the surfaces in the cavity, the temperature profiles, velocity profiles and lastly the derived convection heat transfer coefficients are discussed. The six simulations were the circular tubes, square tubes and rectangular tubes, for both configurations with and without fins. The other properties necessary to calculate the convection heat transfer coefficients are omitted in this chapter for the sake of brevity but are shown in Appendix D.

In the coupled simulations the water temperature and wall temperatures in the riser tubes are presented. The convection heat transfer coefficients on the water side of the riser tubes were also calculated and are also presented. Two variations were simulated, namely one where the mass flow rate was fixed and one where the mass flow was allowed to develop naturally as determined by the numerically calculated buoyancy forces.

8.2 Results for the 3D cases

8.2.1 Overview

Figure 8.2-1 shows a top view of the heater section and superimposes the locations of the line probes that were used to obtain the temperature distributions that are presented in the chapter. The probes consist of a line with 100 discrete points that are located on the axis normal to the figure. The orientation of left vs. right is shown in Figure 8.2-1. Three probes were placed for the heater wall, back wall, and cavity air temperature. The numbering of these probes is done from left to right, with the first position being shown in Figure 8.2-1. The probes associated with the walls (risers' walls, heater wall, back wall and fins) were placed in the middle of the wall. The middle of the wall was chosen as placing the probe on the surface was prone to mistakenly sample cells that are representative of the fluid film. Since the temperature gradients in the walls were small compared to the gradient in the cavity (as seen from Figure 5.1-8), it was theorized that the error would be smaller using the temperature at the middle of the wall. The cavity air temperature monitors were placed roughly in the middle of the cavity between a riser

tube surface and the opposing cavity wall. The locations were considered representative of an average temperature within the region based on the results of the mesh dependency study.

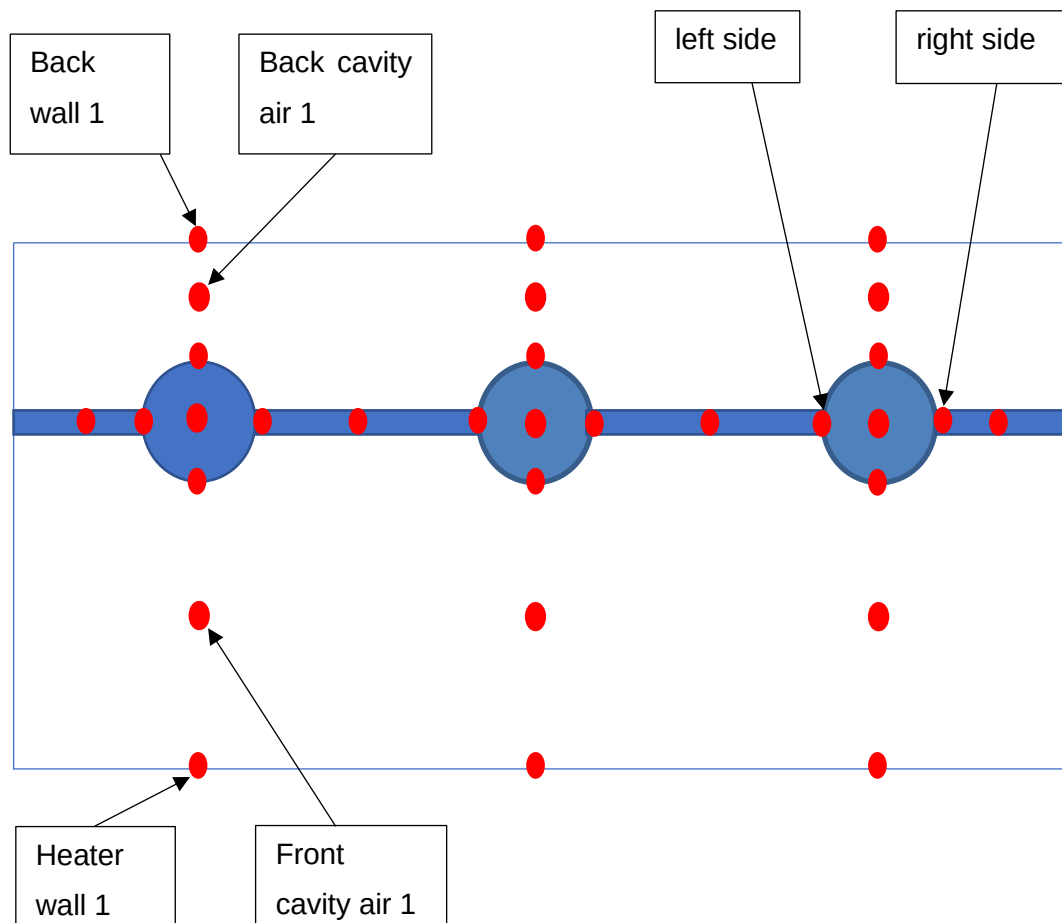


Figure 8.2-1. Locations where temperature probes were placed.

The temperature distributions were extracted and used in further calculations, of which the theory has been discussed in section 3.3. The process consisted of calculating non-dimensional parameters namely the Reynolds-, Prandtl-, Grashof-, Rayleigh-, Aicher- and Nusselt numbers. The convection heat transfer coefficients were then calculated based on the results of the non-dimensional parameters. The coefficients were calculated on the inner surfaces of the riser tubes, for the outer surfaces of the riser tubes, for the fin surfaces (where applicable), for the heater wall surface and for the back-wall surface.

8.2.2 Round tubes with fins

Figure 8.2-2 shows the geometry of the round tubes with fins and naming conventions used as referred to in the discussion of the results.

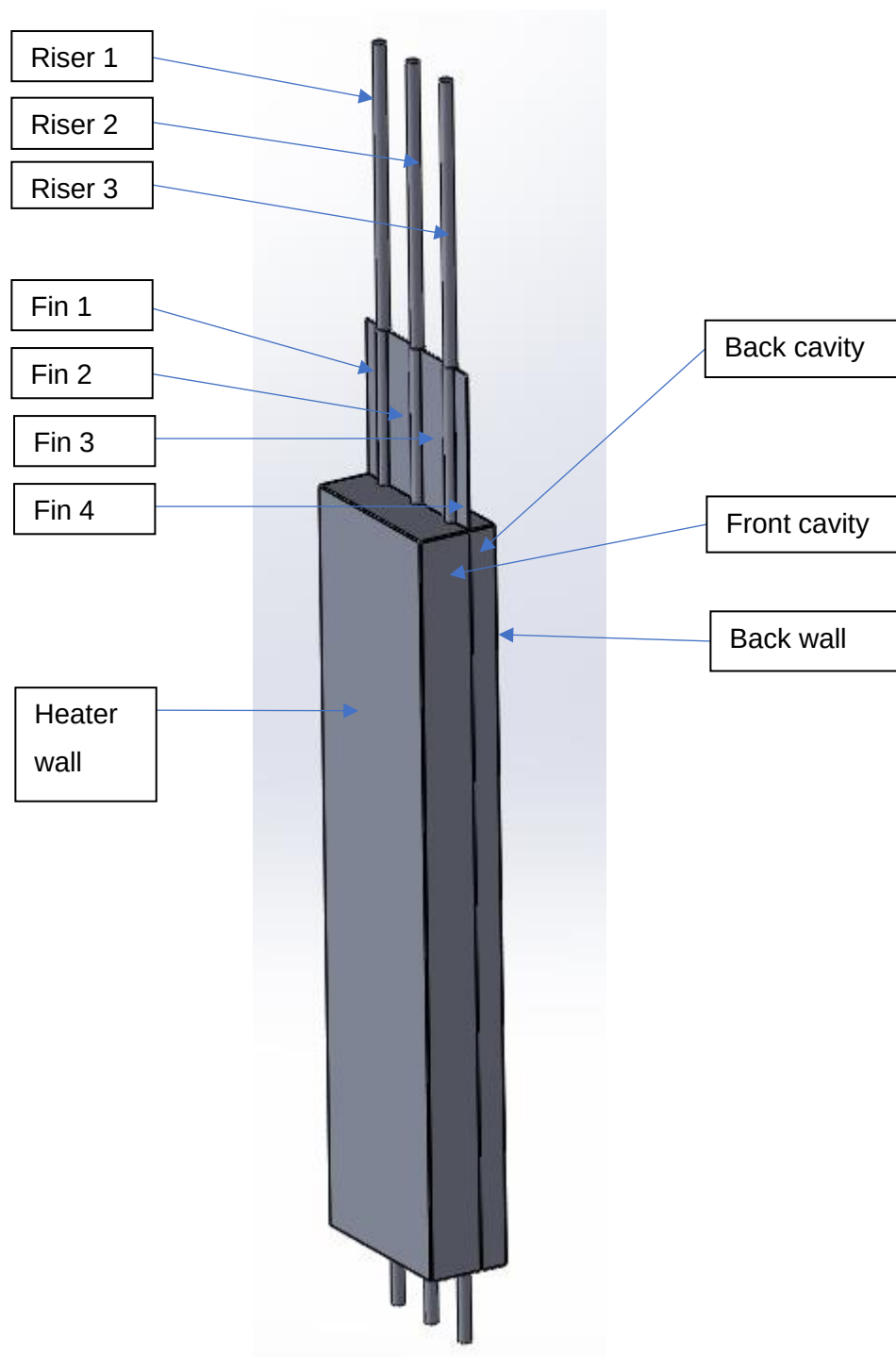


Figure 8.2-2. Naming convention used in the presentation of the results.

8.2.2.1 Heat fluxes

The surface average heat fluxes on the riser tubes' outer surfaces and fin surfaces are shown in Table 8-1. The table distinguishes between total and radiative heat flux received from the front and back cavity. The integral of the flux was also calculated to obtain the heat energy added to the given surface. The negative values on the back-cavity side indicate that the heat moved from the fin into the back cavity. Conversely, the front side shows positive values since the heat flowed from the front cavity into the fin. The outer tubes (Fin 1 & 4) had a similar heat flux and the inner tubes (Fin 2 & 3) had a similar heat flux. The heat flux of the inner fins was larger, since the area and therefore radiation view factors were larger. The radiative heat flux on the front side of the inner fins was approximately 9% larger than on the outer fins. The heat flux that reflected in the back cavity and entered the riser tubes consisted of a fraction of 16% of the heat flux that entered from the front cavity. The incident heat flux on the middle riser was approximately 5% higher than on the tubes on the side. The table also shows the fraction that the radiation heat transfer made up of the total heat transfer. The radiation heat transfer in the back cavity was 20% less than in the front cavity. The remaining percentage of the heat was transferred through convection.

Table 8-1. Heat flux on the riser and fin surfaces (For the round tubes with fins case).

Location	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Derived radiation/total %	
	Front cavity side	Back cavity side	Front cavity side	Back cavity side	Front cavity side	Back cavity side	Front cavity side	Back cavity side
Fin 1	3765.13	-350.70	3244.55	-211.80	1017.33	-94.76	86.17	60.39
Fin 2	4037.33	-428.49	3581.74	-259.75	2181.76	-231.56	88.72	60.62
Fin 3	4002.43	-420.25	3578.87	-254.01	2162.90	-227.10	89.42	60.44
Fin 4	3794.05	-360.28	3233.34	-217.06	1025.15	-97.35	85.22	60.25
Riser 1	4922.33	790.37	3782.08	413.43	1300.34	208.79	76.84	52.31
Riser 2	5190.85	799.13	3904.15	416.39	1371.28	211.11	75.21	52.11

Riser 3	4938.58	790.09	3775.17	413.32	1304.63	208.72	76.44	52.31

8.2.2.2 Temperature distributions

Figure 8.2-3 shows the temperature as a function of elevation in the riser tubes. The riser tube in the middle (Riser 2) showed a slightly higher temperature (approximately 1.3% at the maximum) than the other riser tubes.

The elevation corresponding to the first dashed line represents the lowest elevation of the heated section. Since the risers are heated from the side, the boundary layer had a higher temperature than in the middle of the riser tube. The heating effect is therefore not immediately visible, since the temperature was taken on a line monitor in the middle of the riser tubes. Only after approximately 1 m the heat had reached the middle of the riser tubes after the development of the thermal boundary layer.

The second dashed line in Figure 8.2-3 corresponds to the top of the heated section. The water temperature still increases after leaving the heated section, since the temperatures of the surrounding walls were higher than the water temperature. The third dashed line in the figure represents the top of the finned length. The temperature at these elevations rises slightly due to the redistribution of the heat in the water due to the free stream fluid being heated by the warmer boundary layer fluid. Otherwise stated, the mixing of the fluid with the temperature on the centre line becoming equal to the average temperature in the tube.

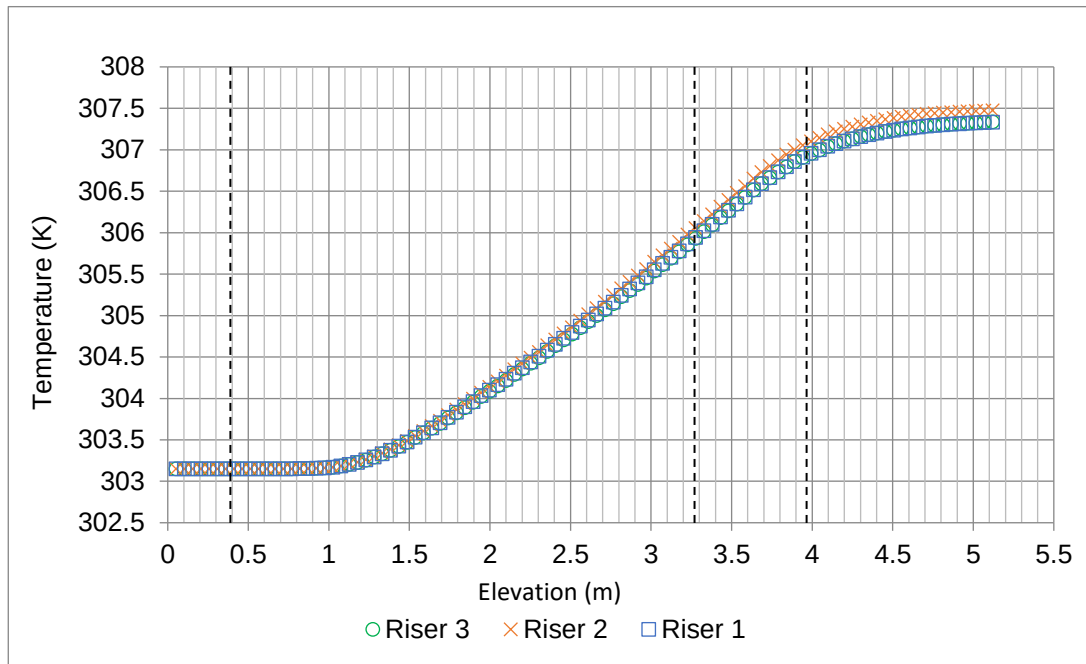


Figure 8.2-3. Riser tube water temperature distributions.

Figure 8.2-4, Figure 8.2-5 and Figure 8.2-6 show the temperature distributions in the riser tube walls. The left and right sides have the highest temperature owing to the heat transfer from the fins. The temperatures show a rapid drop after ≈ 3.3 m (owing to the risers exiting the heated section), causing an approximately uniform temperature distribution around the wall at the topmost elevation.

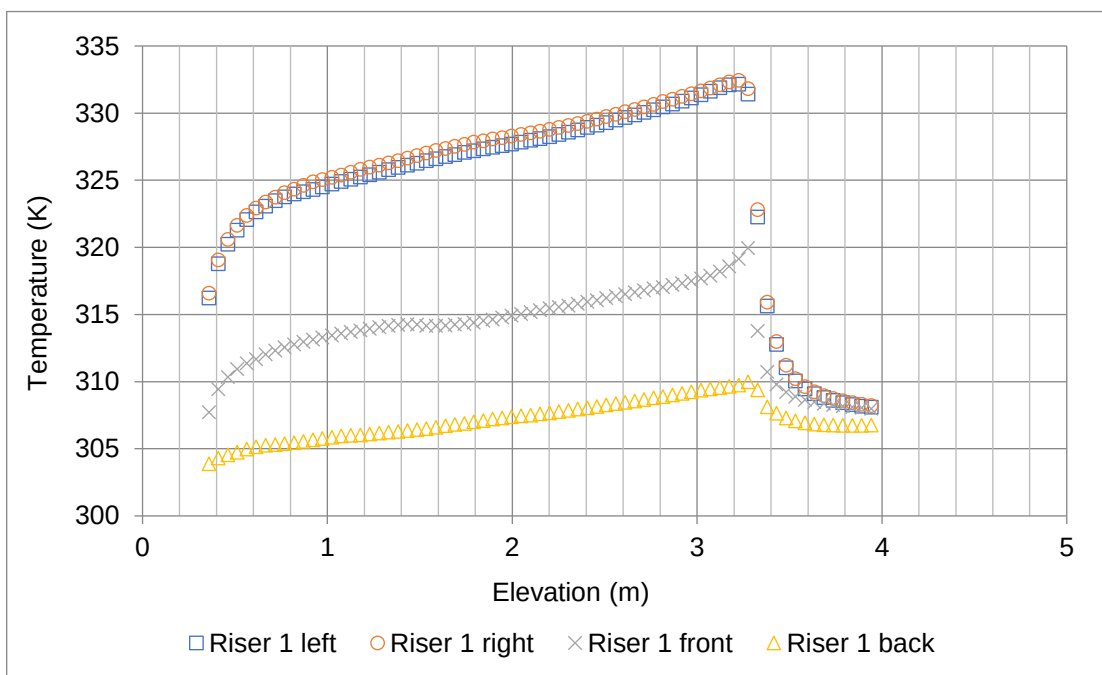


Figure 8.2-4. Riser tube wall temperature distribution for Riser 1.

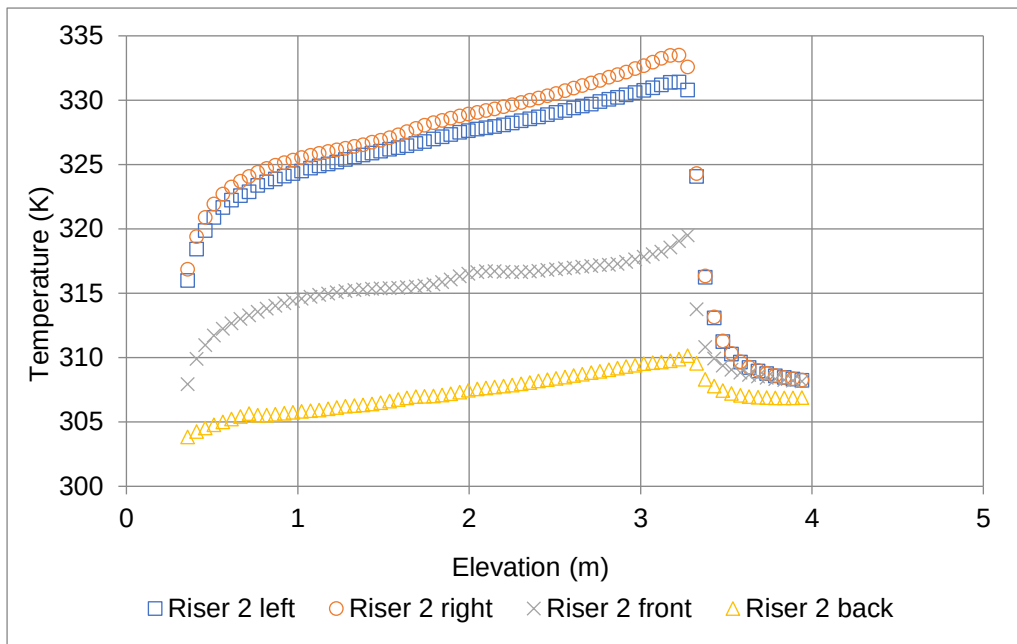


Figure 8.2-5. Riser tube wall temperature distribution for Riser 2.

Since the front walls of the riser tube had a direct view factor from the heater wall, it had higher temperatures than the back sides, which relied on the heat reflecting from the back wall for most of its heat transfer.

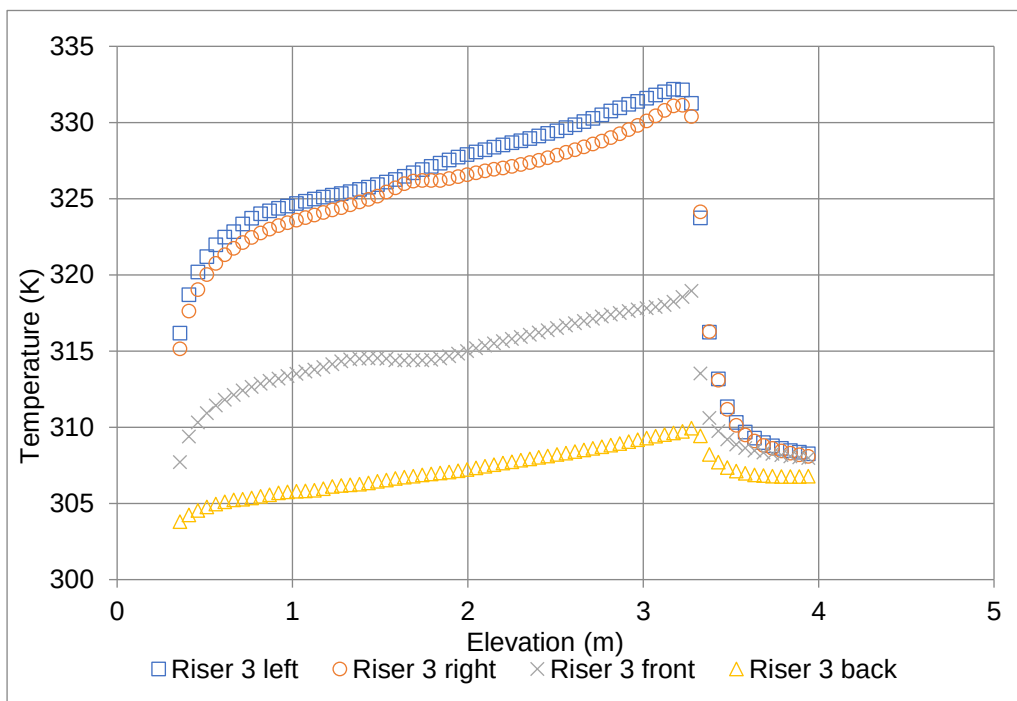


Figure 8.2-6. Temperature distributions within the walls of Riser 3.

The distributions for the three risers are quite similar in their trends and values, which means that a strong degree of symmetry exists in the phenomena. The results of the middle riser tubes could be considered as representative of the other tubes.

Figure 8.2-7 shows the temperature distribution in the fins on a probe located the middle of each fin. The inner fins have a higher temperature than the outer fins, due to the locations of the probes. The probes were taken in the middle of the fins, which was where the maximum temperature was on Fin 2 and 3 (see temperature contour in Figure 8.2-8). The maximum temperature on Fin 1 and 4 was however next to the side wall. A temperature gradient is necessary for heat transfer from the fins to the tubes. The gradient is the same and therefore the outer fins' maximum temperature is against the side wall. The temperatures in the middle of the outer fins are comparable with the temperatures on the inner fins that are half way between the middle of the fins and the tubes.

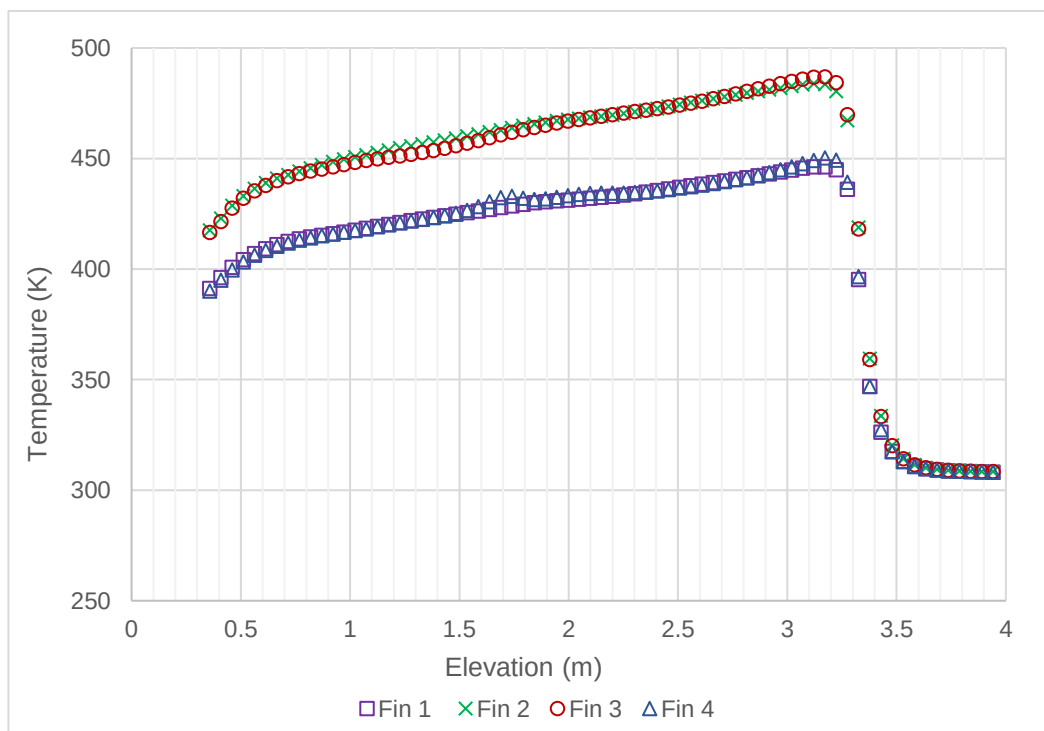


Figure 8.2-7. Fin temperature distribution.

Figure 8.2-8 shows the temperature contour on the fins and riser tube surfaces. The figure shows the locations of the hotspots on the fins and the cooling effect that was present in close proximity to the riser tubes. The temperatures in the fins decrease rapidly above the heater box, especially in the area next to the riser tubes as heat is transferred into the water in the riser tubes. The heating effect of the fins on the left and right sides of the riser tube walls persists even above the heater box.

Figure 8.2-9 shows the temperature distribution of the air in the front and back cavity. The temperatures were taken on three monitors in the middle of the cavities in line with the centres of the riser tubes. The front cavity air was 25 – 47 % warmer than the back-cavity air.

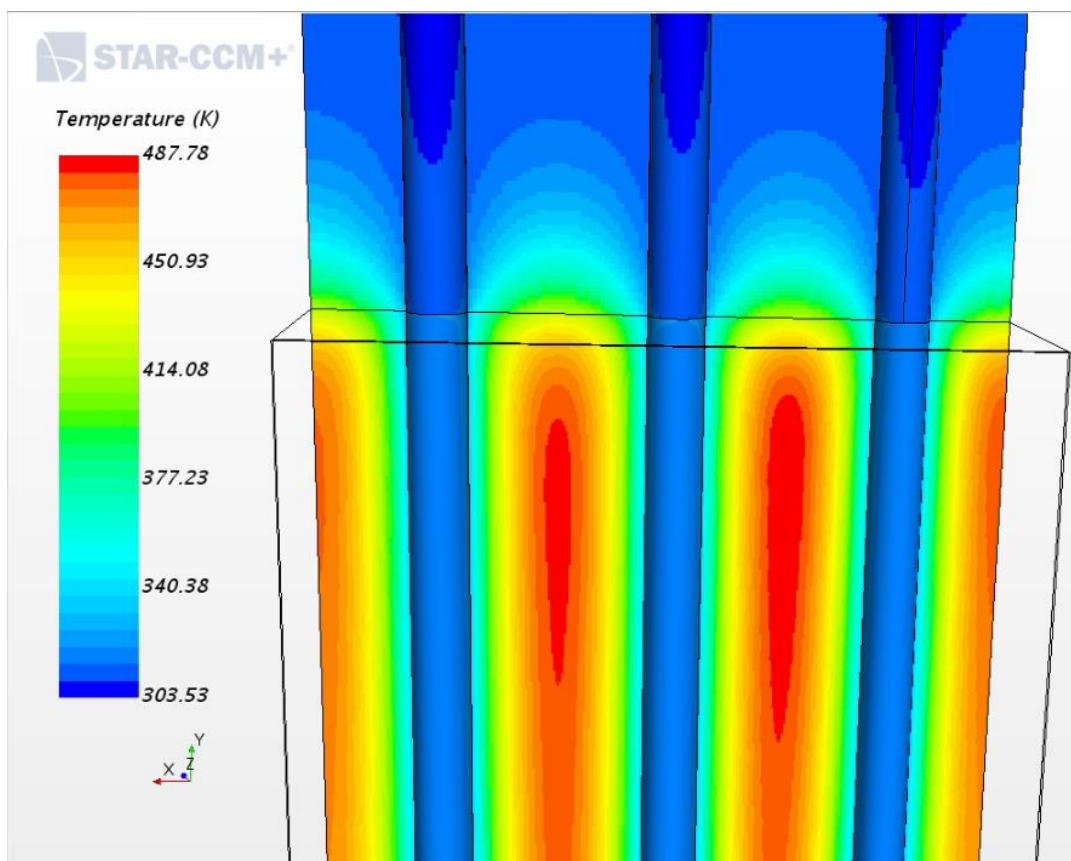


Figure 8.2-8. Temperature contour of the fins and riser tube walls.

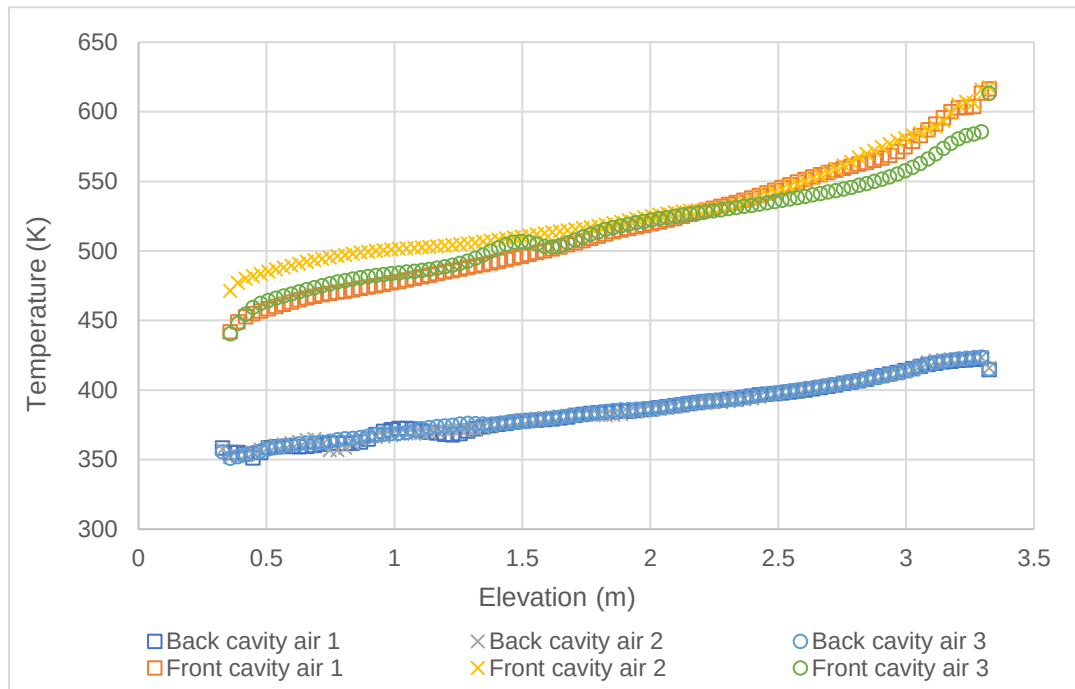


Figure 8.2-9. Air temperature distributions in the front and back cavity.

The wall temperatures on the heated and back wall are shown in Figure 8.2-10. The temperatures were taken on line monitors in line with the centres of the riser tubes. The back-wall temperature is between 64% and 73% lower than the temperatures on the heated wall.

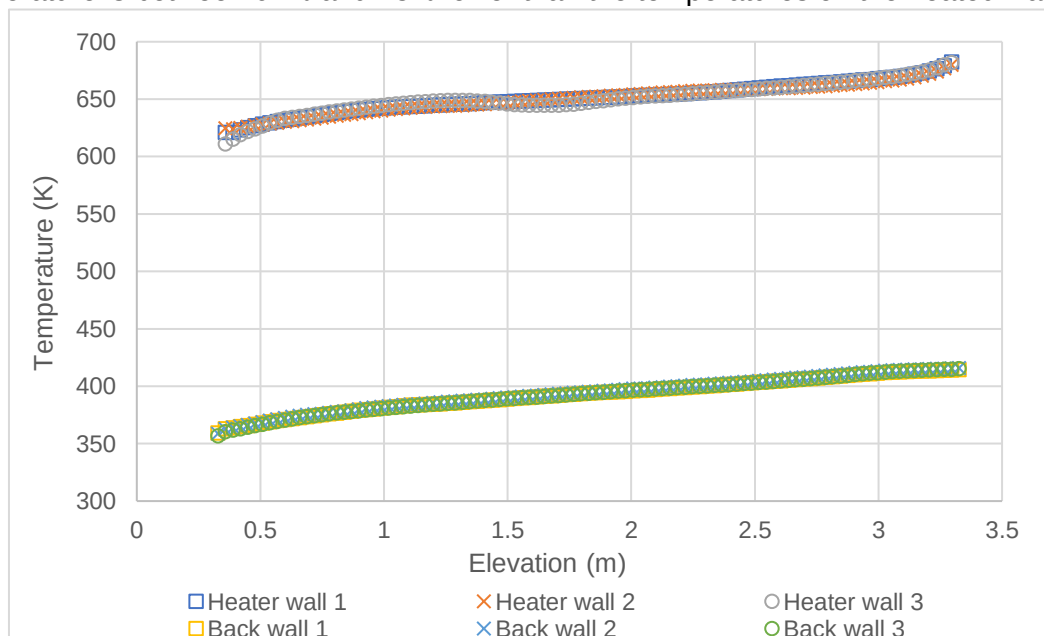


Figure 8.2-10. Heater and back wall temperature distributions.

8.2.2.3 Velocity vectors

Velocity vectors are shown to show the flow patterns and magnitude of the velocity that was found in the air in the front - and back cavity. Vectors are also shown for the flow inside of the middle riser, which gives an insight into the flow regime at four discrete points.

Figure 8.2-11 shows the velocity vectors in the heater section, with focus on the top elevations. The section plane was taken in the middle of the heated section and intersects the middle of the front cavity, middle riser and back cavity. The fluid flow in the front cavity is primarily upwards near the heater wall and downwards near the riser tubes and fins on the section plane. A recirculation cell is also shown in the figure where it occurs in the front cavity. The flow in the back cavity was mostly downwards on the section plane.

Figure 8.2-12 shows the same plane but includes a larger range at the top elevations. The flow against the back wall was driven upward but the flow direction was intermittently interrupted by recirculating flows, exacting a change in the flow direction, as shown in Figure 8.2-12. The dot in Figure 8.2-12 shows the location where the air flow was very small in the y-direction (approximated to a stagnation point and hereafter referred to as such). The turning point between upward flow against the heater wall and downward flow away from the heater wall is also visible.

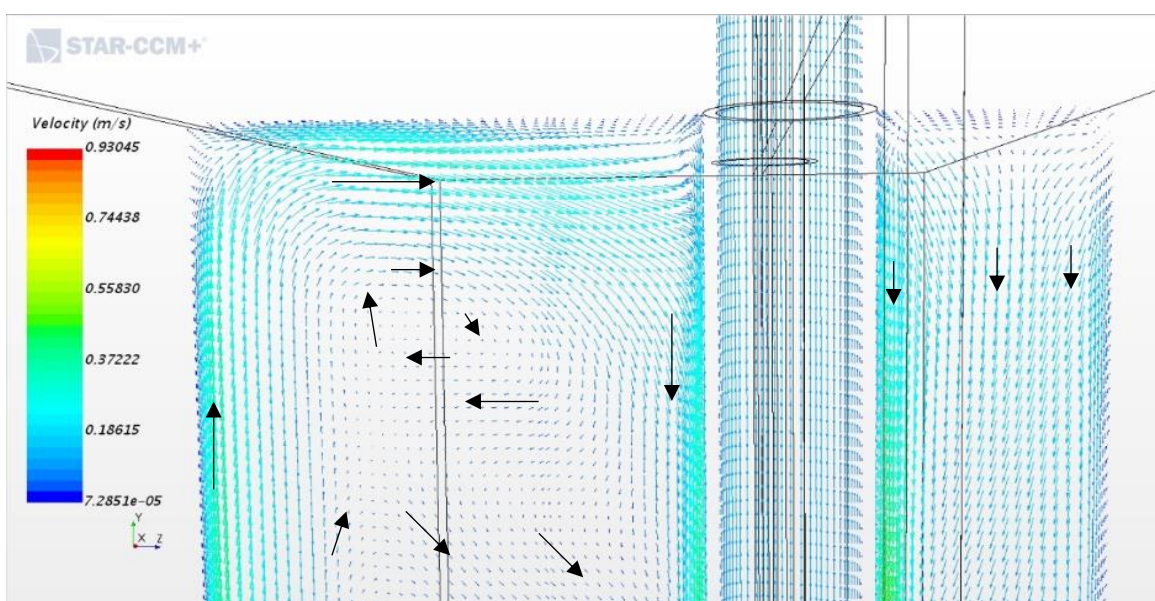


Figure 8.2-11. Velocity vectors in the heated section.

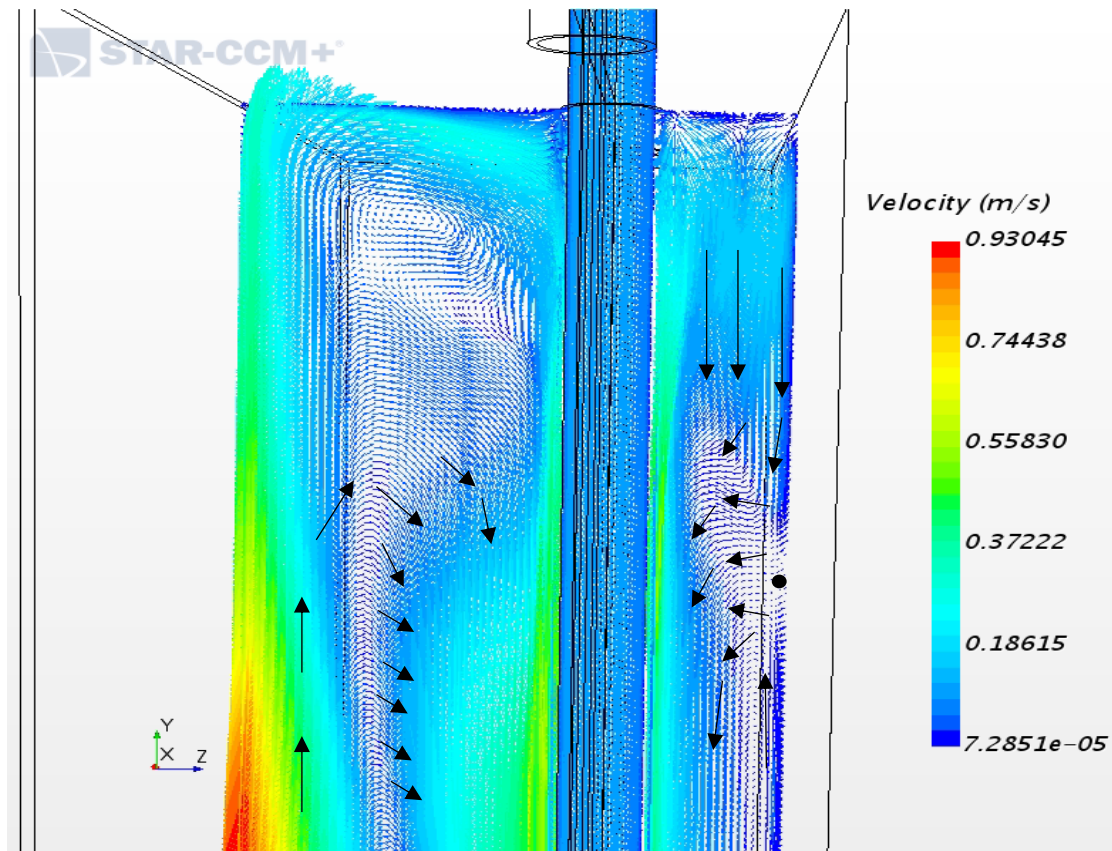


Figure 8.2-12. Vectors on a section plane in the middle of the heated section.

Figure 8.2-13 shows the velocity vectors on a section plane in the middle of the back cavity parallel to the back wall. The flow was primarily downwards near the riser tube walls and upwards near the centre of the fins because of the cooling and heating that occurs at these respective positions. The mixing of streams at the very top are also shown in Figure 8.2-13. The primary direction of the flow near the back wall was downwards. In both Figure 8.2-11 and Figure 8.2-13 the turning points are shown that occur at the top elevations of the cavity. The phenomena are similar at the bottom and therefore not shown to avoid repetition.

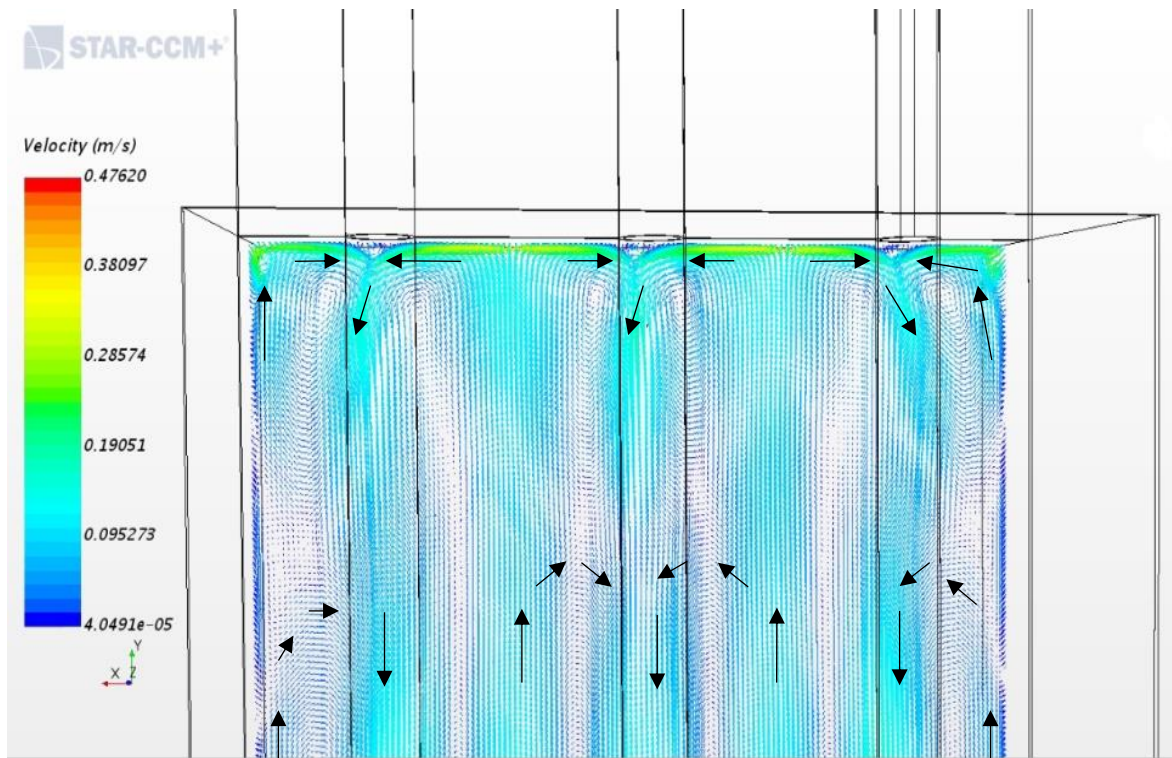


Figure 8.2-13. Velocity vector in the back cavity.

Figure 8.2-14 shows the velocity vectors (as viewed on the x-y plane) for the water inside the middle riser tube. The vectors were taken at four different locations, varying in elevation. The vectors at the lowest elevation, show a laminar profile. From the second elevation the profiles appear to be turbulent with velocity peaks closer to the riser's wall than to the centreline. These peaks were caused by the buoyancy effects having a more exaggerated effect closer to the walls, which was caused largely by the fins. At the highest elevation the profile more closely matches a pure turbulent profile since it was above the heated section and less influenced by buoyancy effects.

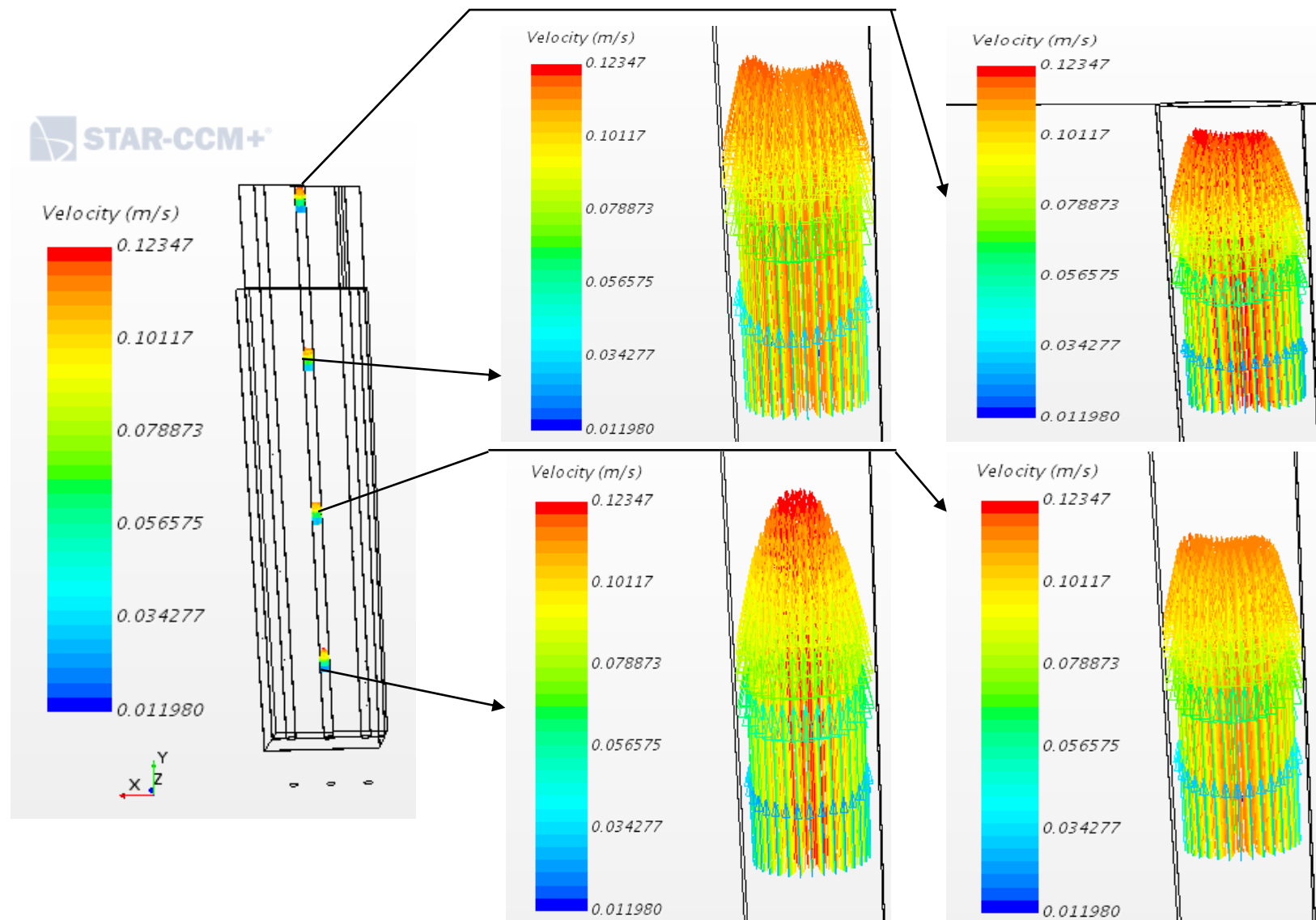


Figure 8.2-14. Velocity profiles in the middle riser.

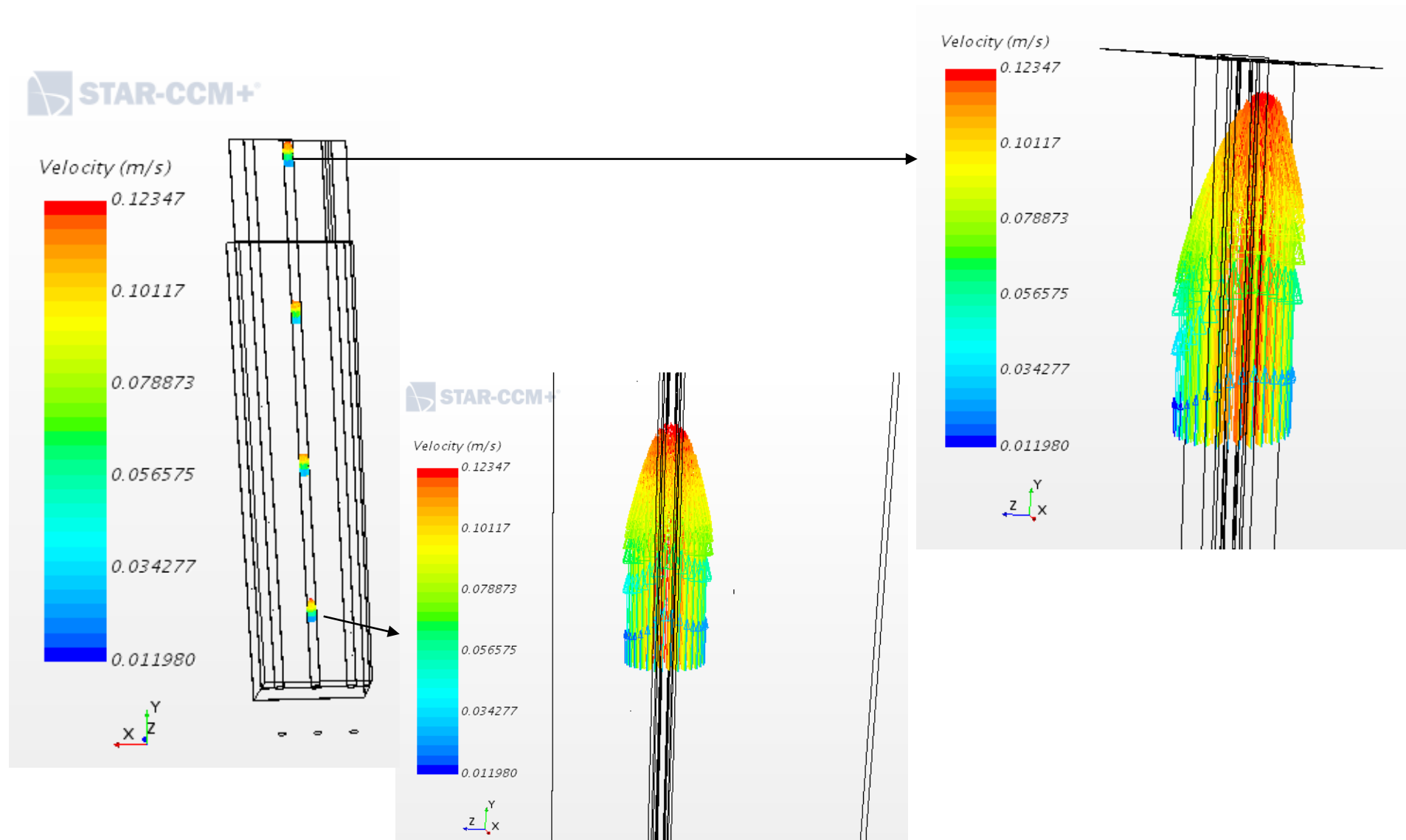


Figure 8.2-15. Velocity vectors in the middle riser (continued).

Figure 8.2-15 shows the velocity vectors as viewed from the x-axis direction (i.e. the heater is to the right hand side). The lowest elevation and highest elevation vectors are shown to indicate the change in the profiles across the elevation. At the lower elevation the maximum velocity was close to the centreline of the riser tube. The higher elevation shows an offset from the centreline, closer to the front surface. This was due to the stronger buoyancy effects from the front surface.

8.2.2.4 Heat transfer coefficients

The parameter hereafter referred to as the Aicher number, which is given by $Aicher = \frac{Ra^{1/3}}{Re^{0.8} Pr^{0.4}}$, was plotted as a function of elevation in Figure 8.2-16. The parameter can be used as an indication of the flow regime; $0.05 < Aicher \text{ number} < 0.2$ indicates mixed convection, Aicher number < 0.05 indicates forced convection and Aicher number > 0.2 indicates free convection. The graph in Figure 8.2-16 which shows the variation in the Aicher number is applicable for the inside surfaces of Riser 2 (middle riser) and is also representative of the other risers, which are therefore not shown.

The convection heat transfer coefficients for the inside surfaces of the riser tubes (water side) was plotted as a function of elevation in Figure 8.2-17. The coefficients were calculated based on the mixed convection theory described in the theory chapter (section 3.3). The coefficients on the sides of the tubes were larger than the coefficients on the front and back owing to the higher temperatures on the side of the walls. The coefficients show a gradual increase for the front and sides and a decrease for the back surface. Figure 8.2-16 can be used to explain this phenomenon. The Aicher number for the front and side surfaces increase further toward free convection as the elevation increases. The larger temperature gradient between the fluid and wall at the front and sides caused a strong free convection influence on these walls. The back surface's Aicher number also increases but the effect of free convection is not as pronounced, such that a "laminarization" as described by Celata et al. (1998) takes place that impedes the heat transfer.

Note that the Reynolds number for the water inside the riser tubes was in excess of 6 000. This means that the forced convection component was fully developed turbulent flow.

Figure 8.2-18 shows the Nusselt number ratio vs. Aicher number for the middle riser tube. The figure further illustrates the deterioration of the Nusselt number (and subsequent convection heat transfer) which occurs on the back wall of the riser tube. The figure also illustrates the recovery of the Nusselt numbers on the front and side surfaces.

The “jump” in the convection heat transfer coefficients for the side walls in Figure 8.2-17 occur where the Aicher criteria exceed 0.2. The regime above the jump was considered to be free convection.

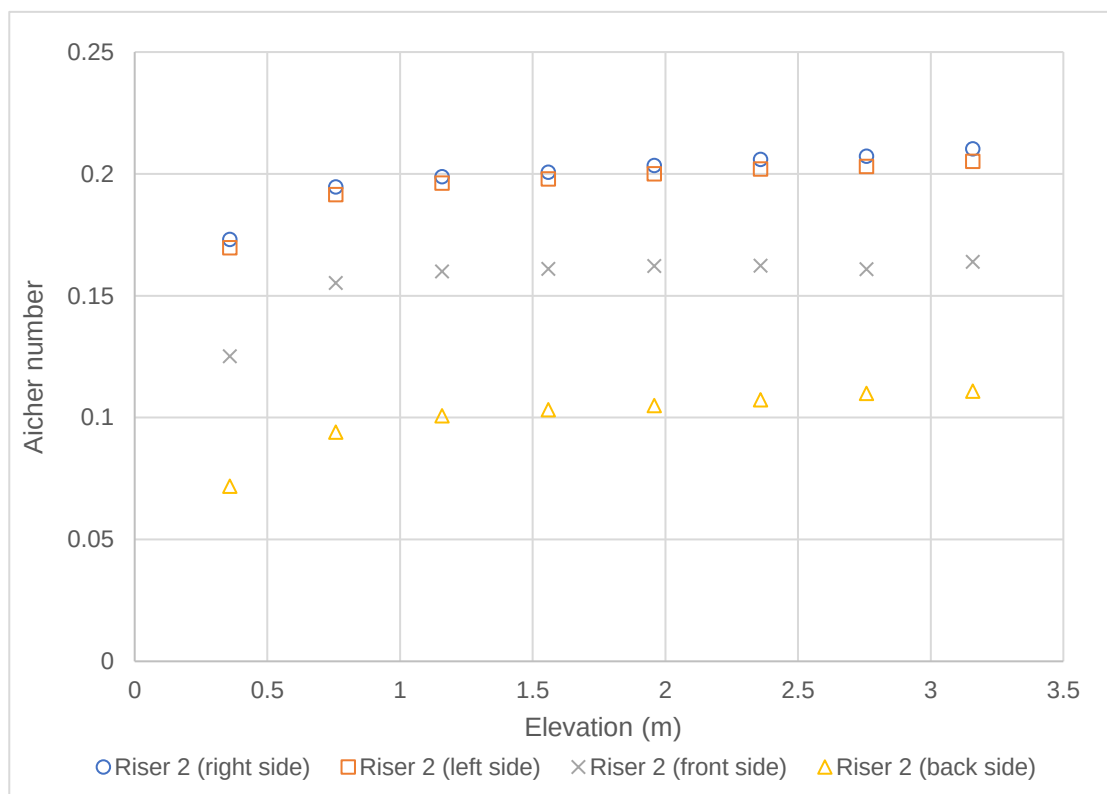


Figure 8.2-16. Aicher number as a function of elevation.

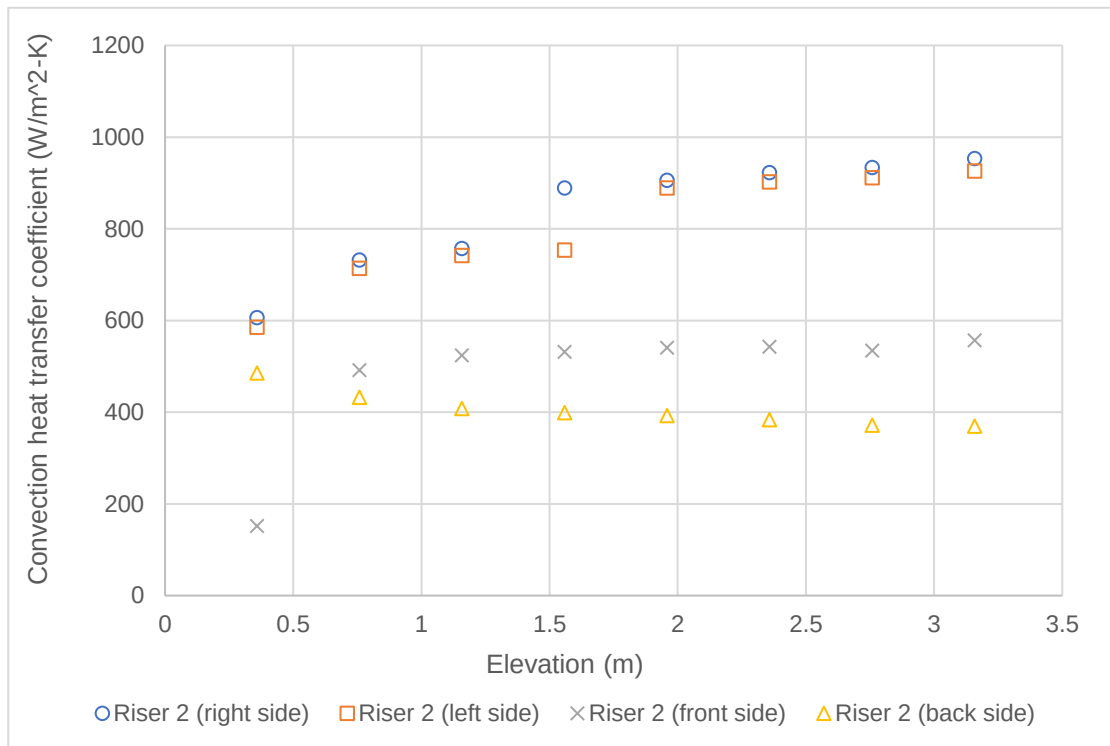


Figure 8.2-17. Convection heat transfer coefficient as a function of elevation.

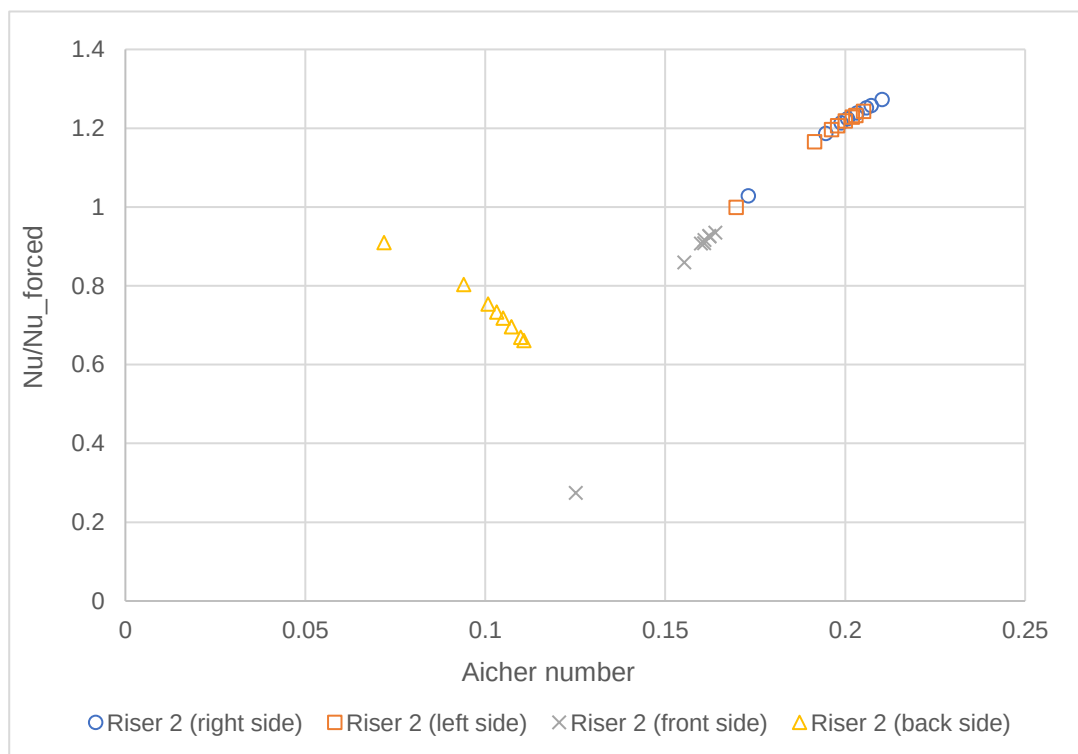


Figure 8.2-18. Nusselt number ratio vs. Aicher number.

Figure 8.2-19 shows the convection heat transfer coefficients on the outer surfaces of the fins. The coefficients in the front cavity show an overall increase, while the corresponding coefficients in the back cavity show an overall decrease. In the back cavity the difference between the fluid and surface temperatures decreased as the air was heated to a temperature closer to the surface temperature. In the front cavity the difference between the fluid and surface temperature increases, which resulted in the increase in the coefficient.

The two middle fins (#2 and #3) in the back cavity had a higher coefficient than the fins at the sides (#1 and #4). The behaviour was the opposite for the front cavity. The main driving force behind the phenomena is the difference in the fluid temperature (cavity air) and wall temperature. In the back cavity the lower surface temperature that was sampled on the side fins resulted in a smaller difference between the temperatures, and therefore convection heat transfer coefficients for these fins. In the front cavity the lower temperatures on the side fins resulted in a larger difference between the fluid and wall temperatures, resulting in larger coefficients.

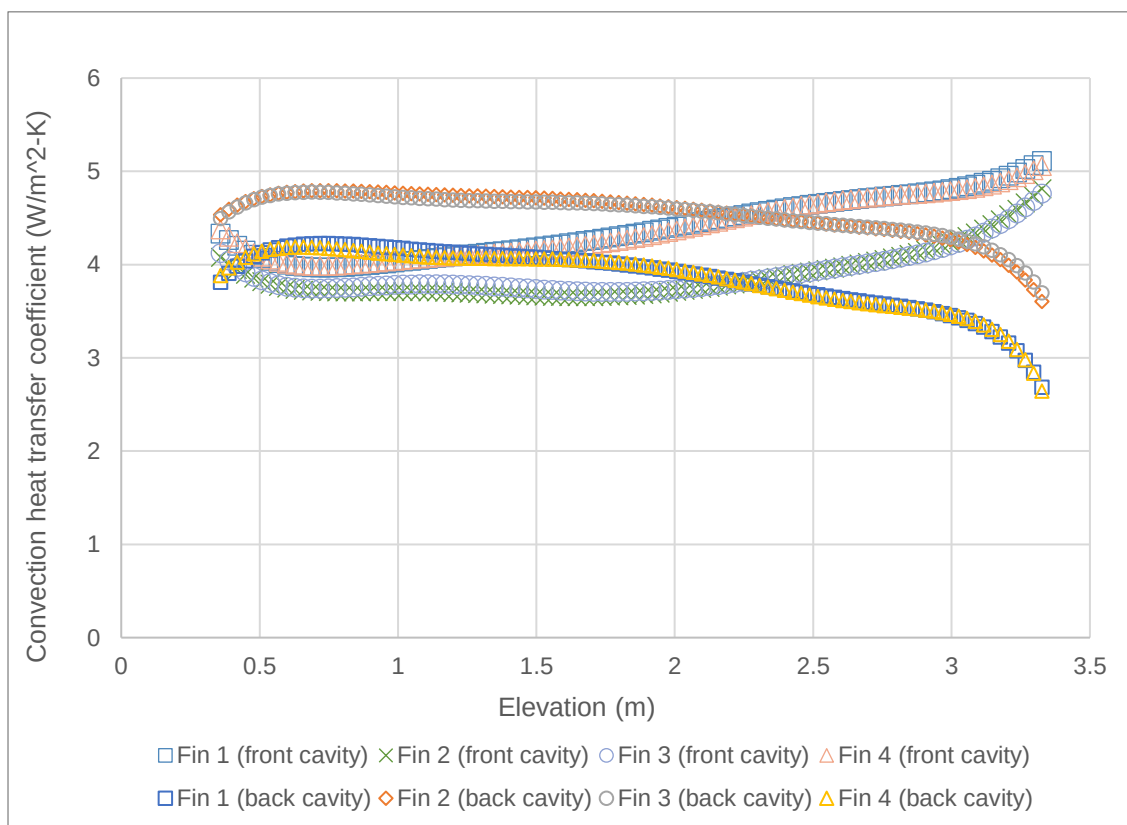


Figure 8.2-19. Convection heat transfer coefficients on the outer surfaces of the fins.

Figure 8.2-20 shows the convection heat transfer coefficients on the outer surfaces of the riser tubes for the front - and back cavity side. The coefficients show an overall increase on both sides. The surface temperatures on the riser tubes were approximately 100 K lower than the temperatures on the fin surfaces. This meant that the difference between the fluid and surface temperatures always remained high throughout the elevation of the heated section. This in turn meant higher coefficients.

A few important issues that probably should be considered are firstly, in the front cavity the air flow was downwards against the fins, transferring heat to the fins while cooling the air. Secondly, in the case of the back cavity the air flows upward against the fins, transferring heat from the fins and heating up the air. Thirdly, for both front and back cavities the air flow was downwards against the riser tubes thus cooling down the air.

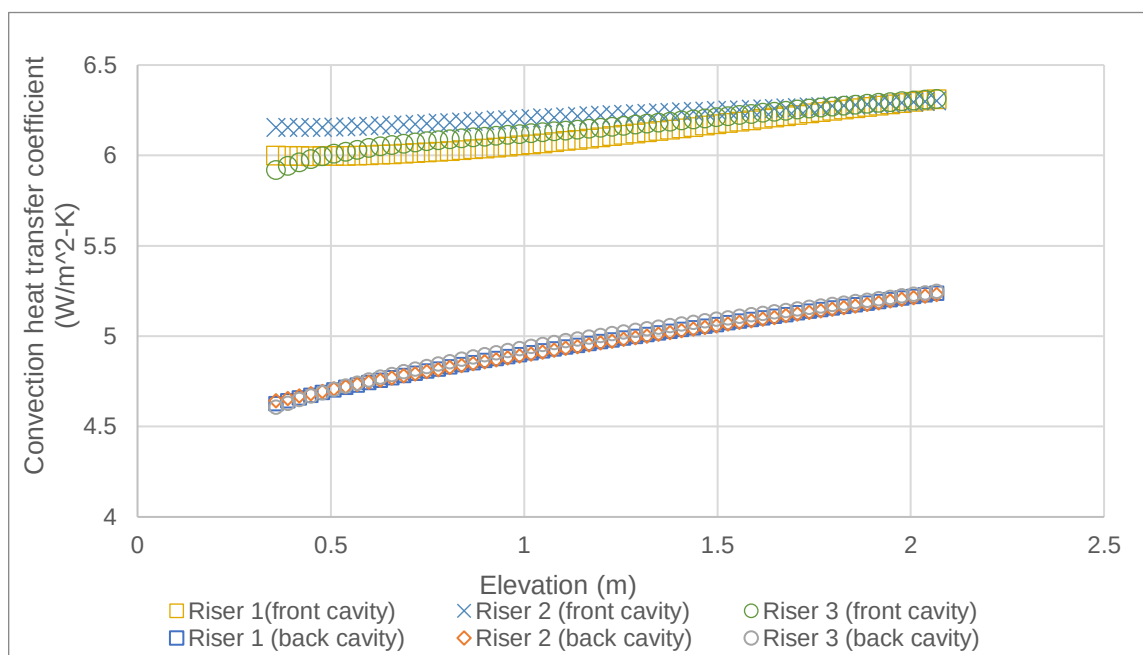


Figure 8.2-20. Convection heat transfer coefficients on the outer surfaces of the riser tubes.

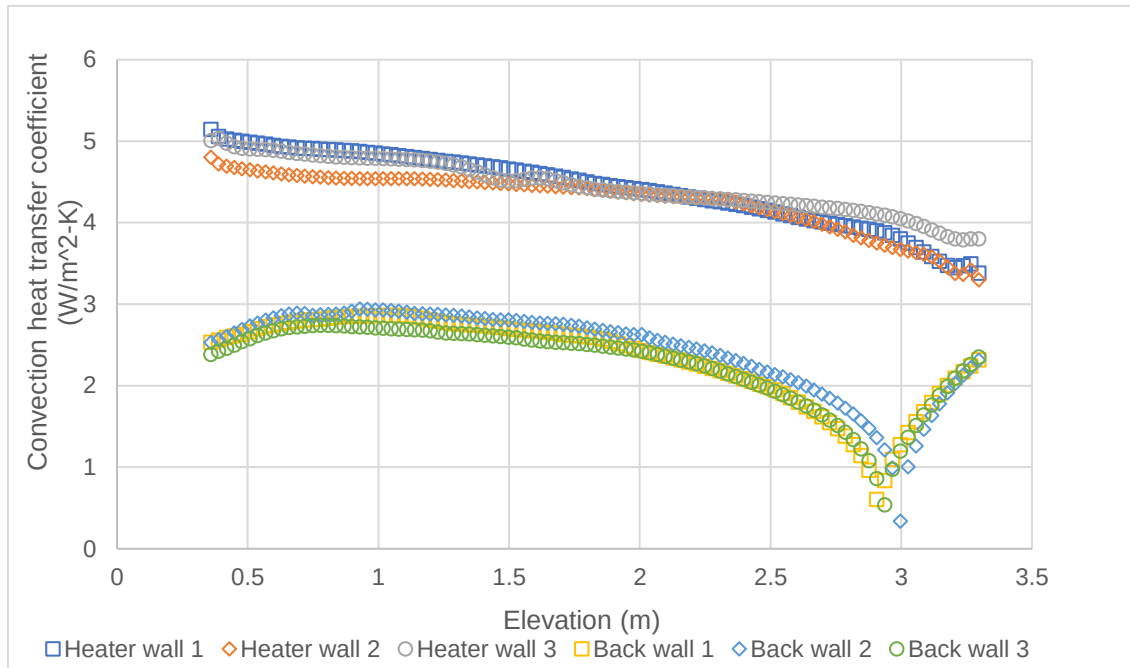


Figure 8.2-21. Convection heat transfer coefficients on the surfaces of the heater – and back wall.

Figure 8.2-21 shows the convection heat transfer coefficients as calculated for the heater – and back wall. The coefficients show an overall decrease over the elevation of the heated section. The difference in the surface – and air temperature decreased with an increase in elevation, resulting in the decrease in the coefficients. On the back wall the fluid temperature exceeded the surface temperature at approximately 3 m, causing the minimum coefficient to be calculated at this location. Additionally, the air flow direction against the back wall was primarily forced upwards for most of the elevation. Above the elevation of ≈ 3 m however, the flow direction was downwards. A ‘stagnation point’ (since the approximate velocity in the y-direction was very small) exists against the back wall where the flow direction changes and was indicated by a dot in Figure 8.2-12. Since the flow velocity was lower the convection heat transfer was diminished at this location, which explains the decrease in the convection heat transfer coefficients against the back wall at elevations of ≈ 3 m.

The parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix D.2.

8.2.2.5 Conclusion

The circular tubes with fins case forms the baseline as the geometry used in the design of the UW experimental facility. The boundary conditions were based on one of the experimental runs done by Lisowski (2013). The mesh that was used followed from a mesh dependence study, in which the method was verified by comparing results from two different codes, namely Star-CCM+ and Fluent.

The temperature profiles, temperature contours and velocity vectors were investigated. The convection heat transfer coefficients were calculated for the air-cooled surfaces in the cavity, as well as for the water-cooled surfaces inside the riser tubes. The division of the heat fluxes, between the riser tube and fin surfaces was also shown. Radiation heat transfer accounted for most of the heat transfer, with more than 75% in the front cavity. In the back cavity however the convection heat transfer had an increased influence, since the radiation heat transfer accounted for 61% (maximum) of the heat transfer.

The results expand on the study done by Lisowski (2013) by adding the results for the phenomena in the back cavity, including temperature distributions, convection heat transfer coefficients and the velocity vectors.

The mixed convection regime was present in the riser tubes and the Nusselt numbers used to calculate the convection heat transfer coefficients were therefore calculated as a combination of the free convection and forced convection Nusselt numbers.

The velocity vectors were shown for the flow in the cavity as well as for the inside of the riser tubes. The velocity vectors in the cavity showed the flow patterns that were present. The velocity vectors of the middle riser tube provided insight into the prevailing flow regime inside the riser tubes.

8.2.3 Round tubes without fins

Figure 8.2-22 shows the geometry of the round tubes without fins, with the naming convention used on the geometry as it is referred to in the discussion of results. The geometry consisted of circular tubes that are water cooled vertically rising through an air-filled heated cavity. The cross-sectional area of the riser tubes remained the same as in the finned case.

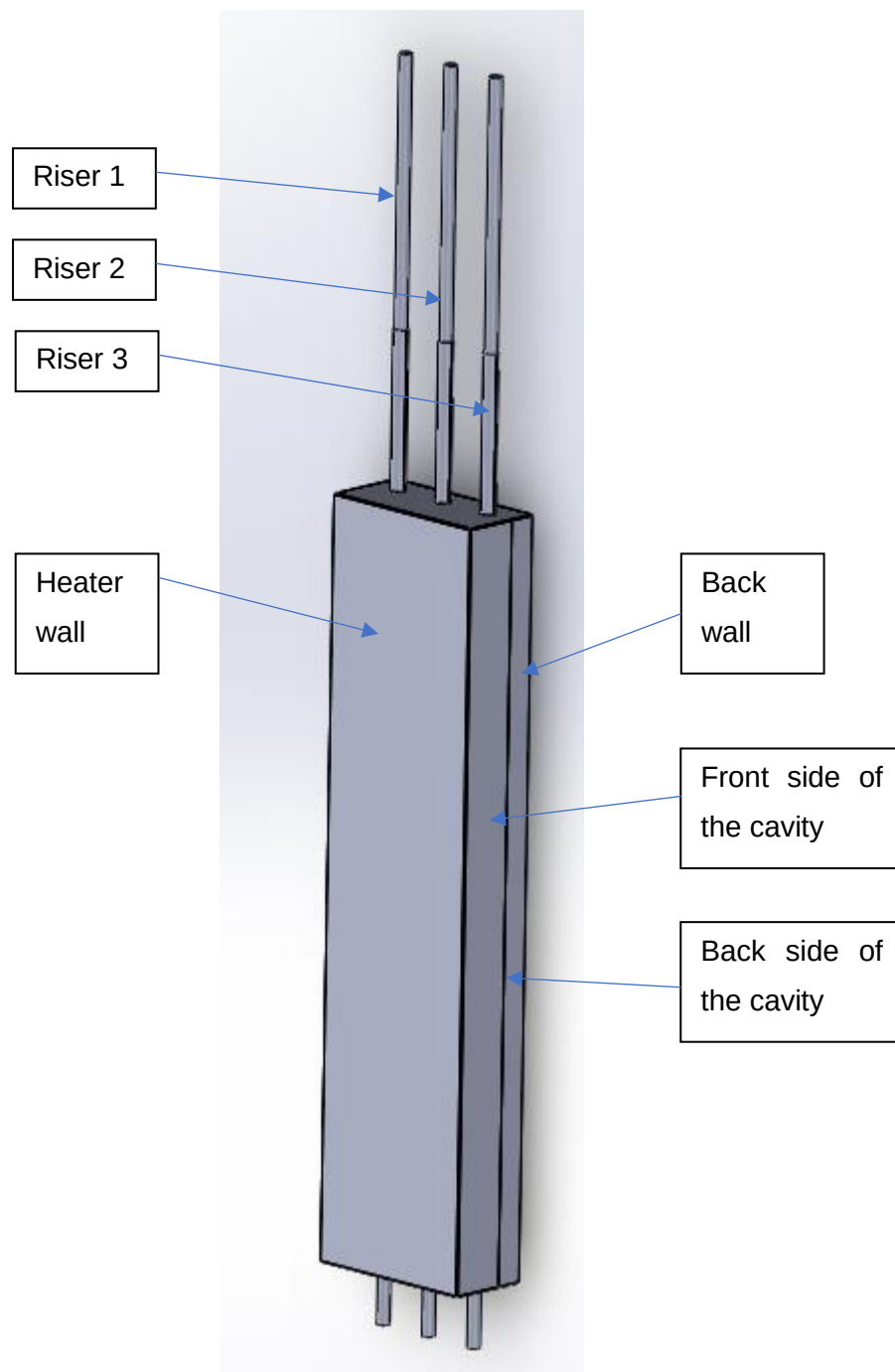


Figure 8.2-22. Geometry and naming convention as referred to in the results.

8.2.3.1 Heat fluxes

The surface average heat fluxes on the riser tubes' outer surfaces are shown in Table 8-2. Table 8-2 distinguishes between the total heat flux and radiative heat flux on the front and back of the riser tubes' outer surfaces. The integral of the heat flux was also calculated to yield the heat energy that was added to the risers from the cavity. The fraction of radiative/total heat transfer was also calculated. The convection heat transfer constitutes a larger fraction in the front part of the cavity when compared to the case with round tubes with fins. In this case the surfaces in the back part of the cavity have a non-zero view factor with respect to the heater wall. In the case described in 8.2.2 the radiative heat transfer was absorbed mostly in the front cavity, but in this case the radiative heat transfer was more dispersed between the front and back part of the cavity. The heat that the riser tubes' back cavity side received was approximately 86% of the heat received from the front cavity side. This was a noticeable increase from the corresponding finned case, where this percentage was approximately 15%. This represents a 70% increase in the heat received by the back sides of the riser tubes as a result of omitting the fins from the design.

Table 8-2. Heat flux on the riser tube surfaces (For the round tubes without fins).

	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Derived Radiation/Total heat transfer %	
	Front side	Back side	Front side	Back side	Front side	Back side	Front side	Back side
Riser 1	6664.75	5861.89	4002.95	3161.83	1880.72	1654.16	60.06	53.94
Riser 2	6796.52	5878.32	4101.49	3215.24	1917.90	1658.80	60.35	54.70
Riser 3	6683.27	5758.91	4006.84	3163.49	1885.95	1625.10	59.95	54.93

8.2.3.2 Temperature distributions

The water temperature in the three riser tubes is shown as a function of elevation in Figure 8.2-23. The temperature distributions are very similar for the three tubes, however temperatures in Riser 2 are slightly higher than Riser 3 (0.018% maximum), which in turn was slightly higher than the temperatures in Riser 1 (0.007% maximum). The first dashed line (lowest elevation) indicates the lowest elevation of the heated section. The second dashed line indicates the top of the heated section and the third dashed line indicates the top elevation of the finned length. The temperatures in the figure show a delay in increasing, since the heat didn't spread immediately to the middle of the tubes where the line probe was placed. Similarly, the temperatures kept increasing after leaving the heated section, since the middle was still being heated by the warmer fluid in the boundary layer. A redistribution therefore occurs that endeavours to obtain a uniform temperature across each riser.

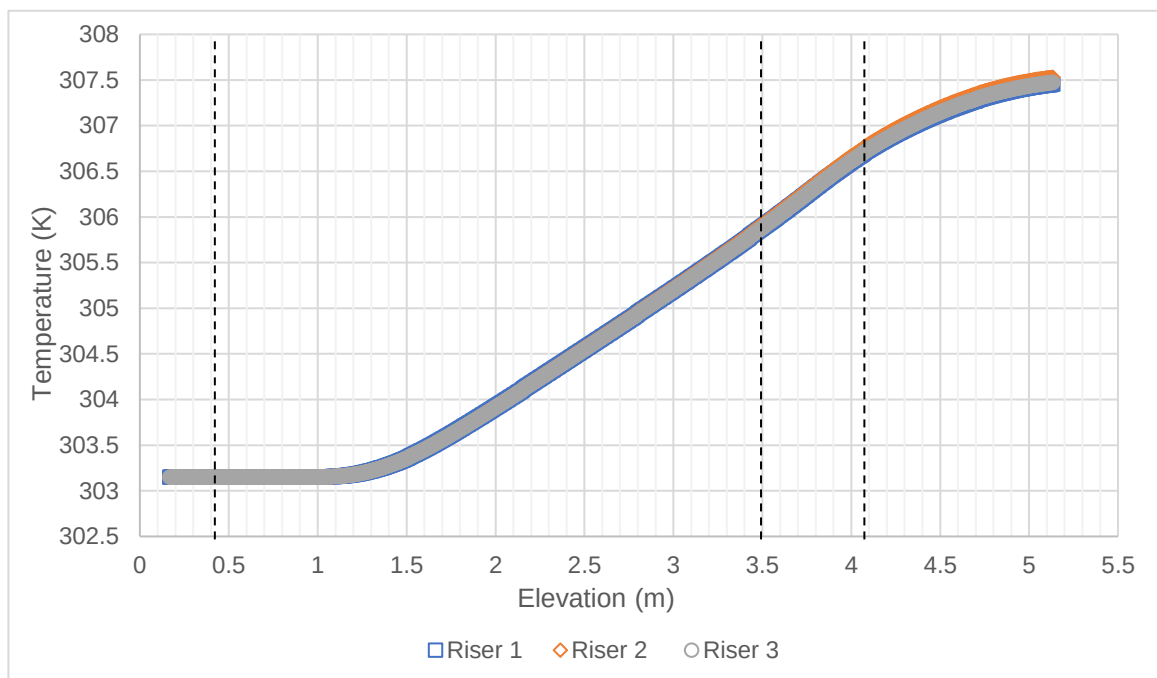


Figure 8.2-23. Riser tube temperature distributions.

Figure 8.2-24, Figure 8.2-25 and Figure 8.2-26 show the temperature distribution in the walls of the riser tubes and represent localized temperatures as found on line probes. The temperatures at the highest elevations show oscillatory behaviour due to the circulation that occurs from the front to the back side of the cavity. At the elevation of approximately 3.2 m, the back walls experienced a temperature rise, while simultaneously the sides experienced a

temperature drop. The heat was therefore removed from the side walls of the riser tubes as air moved into the back part of the cavity. A stagnation point exists at the location on the back surface of the riser tube near where the temperature probe was placed (refer to Figure 8.2-1), which created a hot pocket resulting in the temperature rise. The dramatic drop in the surface temperatures above 3.4 m coincides with the risers' exit from the heated section.

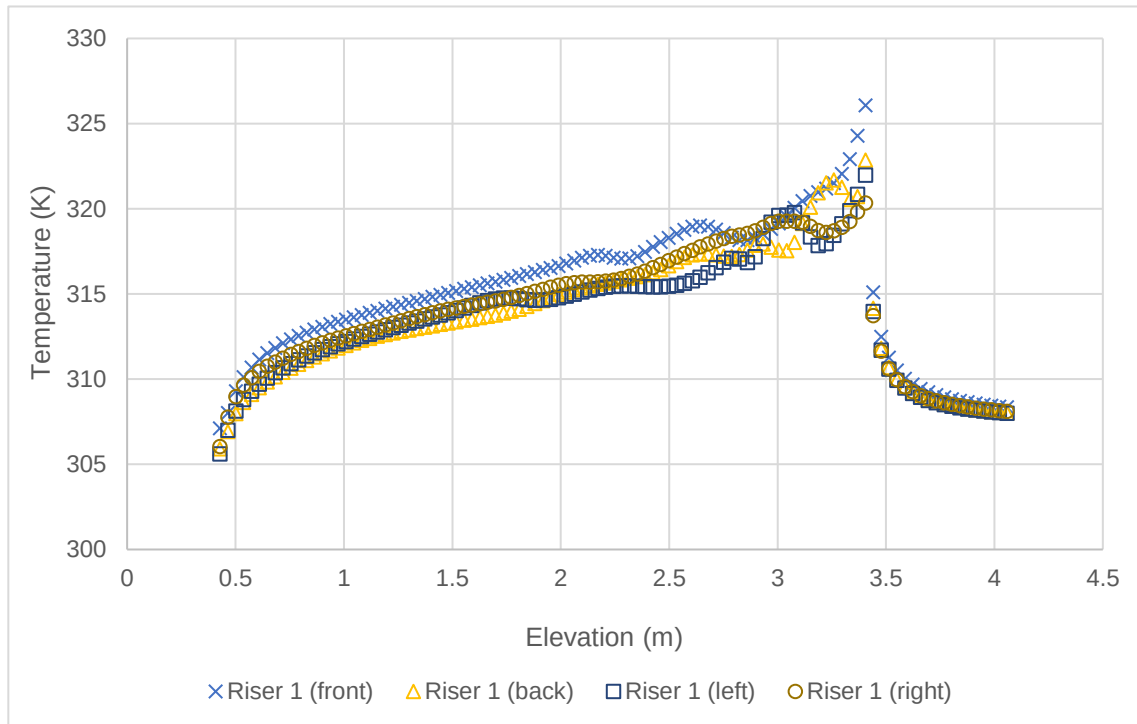


Figure 8.2-24. Wall temperature distributions for Riser 1.

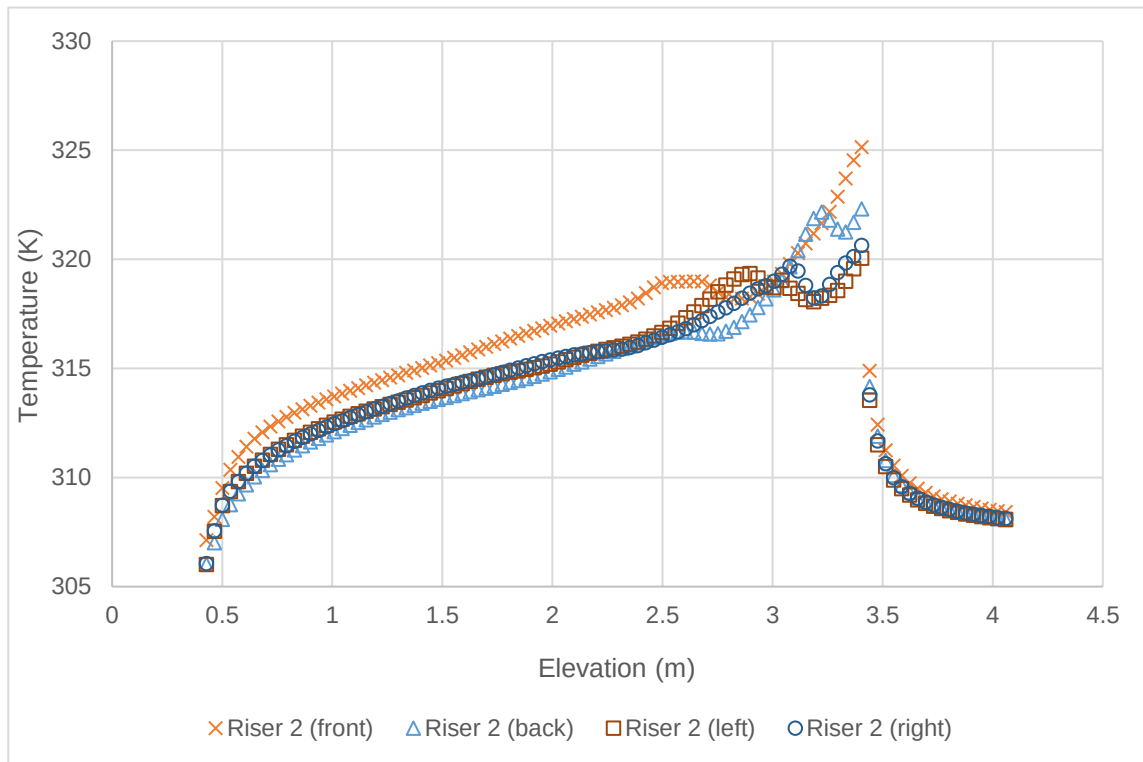


Figure 8.2-25. Wall temperature distributions for Riser 2.

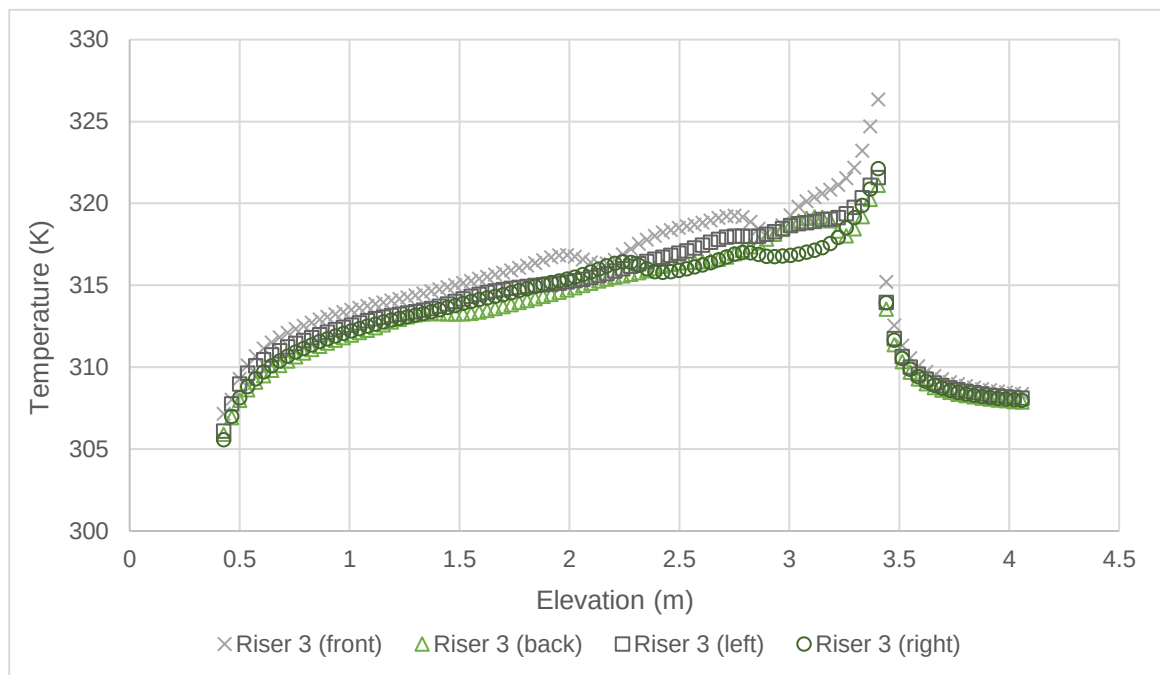


Figure 8.2-26. Wall temperature distributions for Riser 3.

Figure 8.2-27 shows the cavity air temperature distributions. The temperatures were taken on line probes situated at the middle of the front and back sides, in line with the centrelines of the

riser tubes. The graph shows localized temperature peaks for the back cavity where the heated air from the front cavity circulated to the back cavity. The temperature distribution can be seen to be more uniform between the front cavity and back cavity air than in the corresponding finned case (see Figure 8.2-9).

Figure 8.2-28 shows the temperature distribution on the heater and back walls. The temperatures were taken on a line probe that was in line with the centrelines of the riser tubes. The heater wall was found to be approximately 11 - 23% warmer than the back wall. As can be expected the back wall temperature was significantly higher than in the finned case. The temperature gradient within both sides of the cavity was also larger than in the finned case, since the air in the cavity could distribute the heat throughout the front and back part of the heated section. A hot spot formed at the top elevations on the heater wall, since the rising hot air converged at this point. The cavity's top wall also slowed the flow down that contributed to the formation of the hotspot. In contrast, the back wall temperatures show a plateau at the top elevations since the flow patterns effectively removed the heat from this spot and didn't allow for transport of the hot air from the front cavity to reach this location.

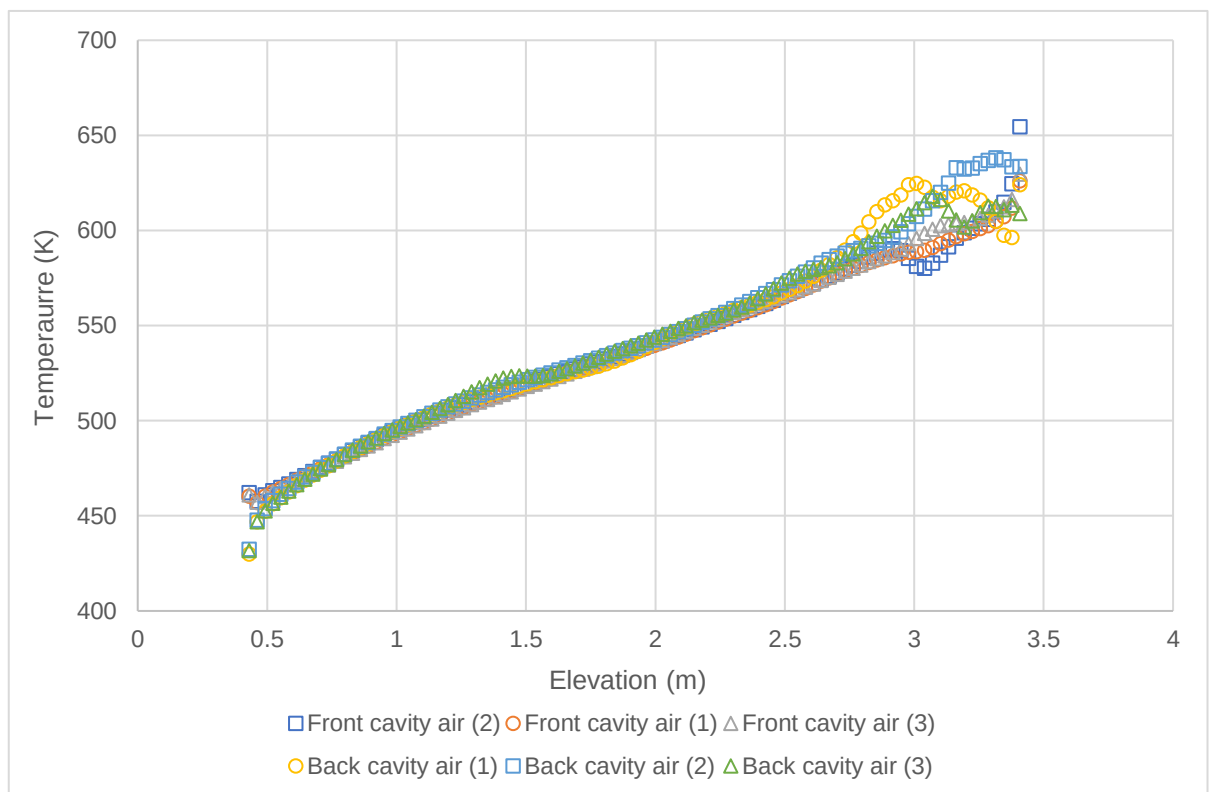


Figure 8.2-27. Cavity air temperature distributions.

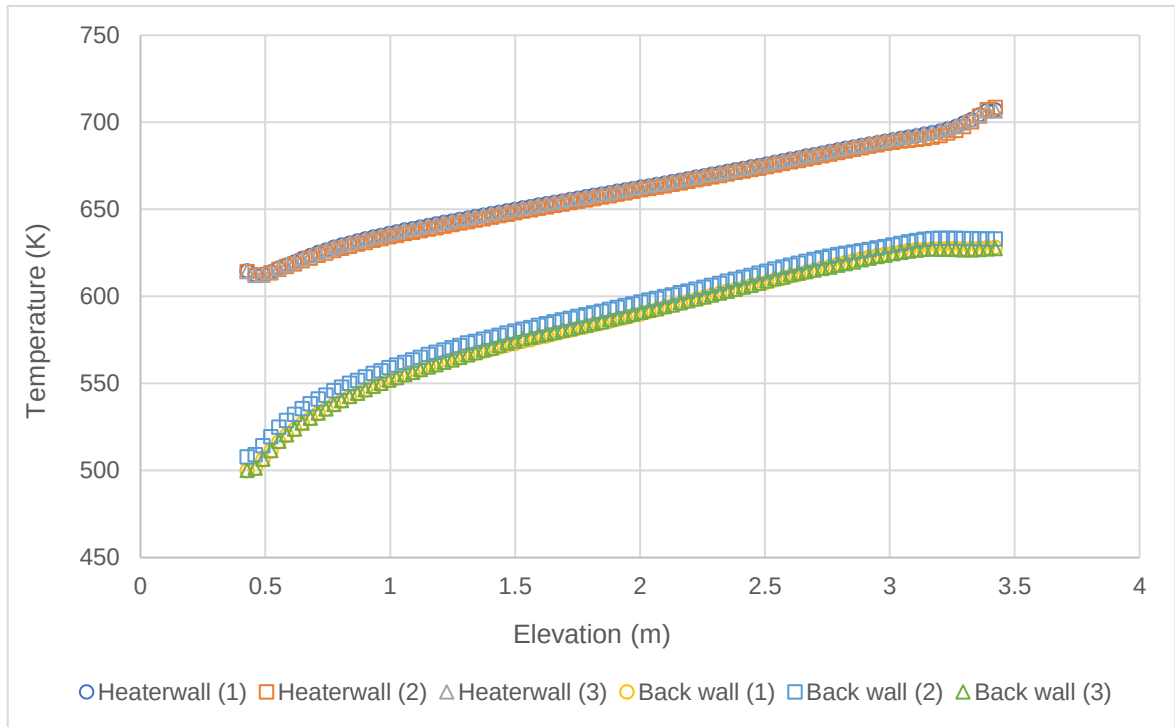


Figure 8.2-28. Heater – and back wall temperature distributions.

8.2.3.3 Velocity vectors

A plane section was placed in the middle of the heated section and intersected the front cavity, middle riser tube and back cavity. The velocity vectors on the section plane are shown in Figure 8.2-29. The arrows were added to enhance clarity by showing the main directions of flow. The flow direction on the plane was upwards in the vicinity of the heater wall and downwards next to the riser tube walls. The effects seen in the back part of the cavity was due the heated air from the front cavity being reflected by the back wall towards the risers. In addition, direct radiation from the heater wall reaching the back wall also participated in forming the flow patterns.

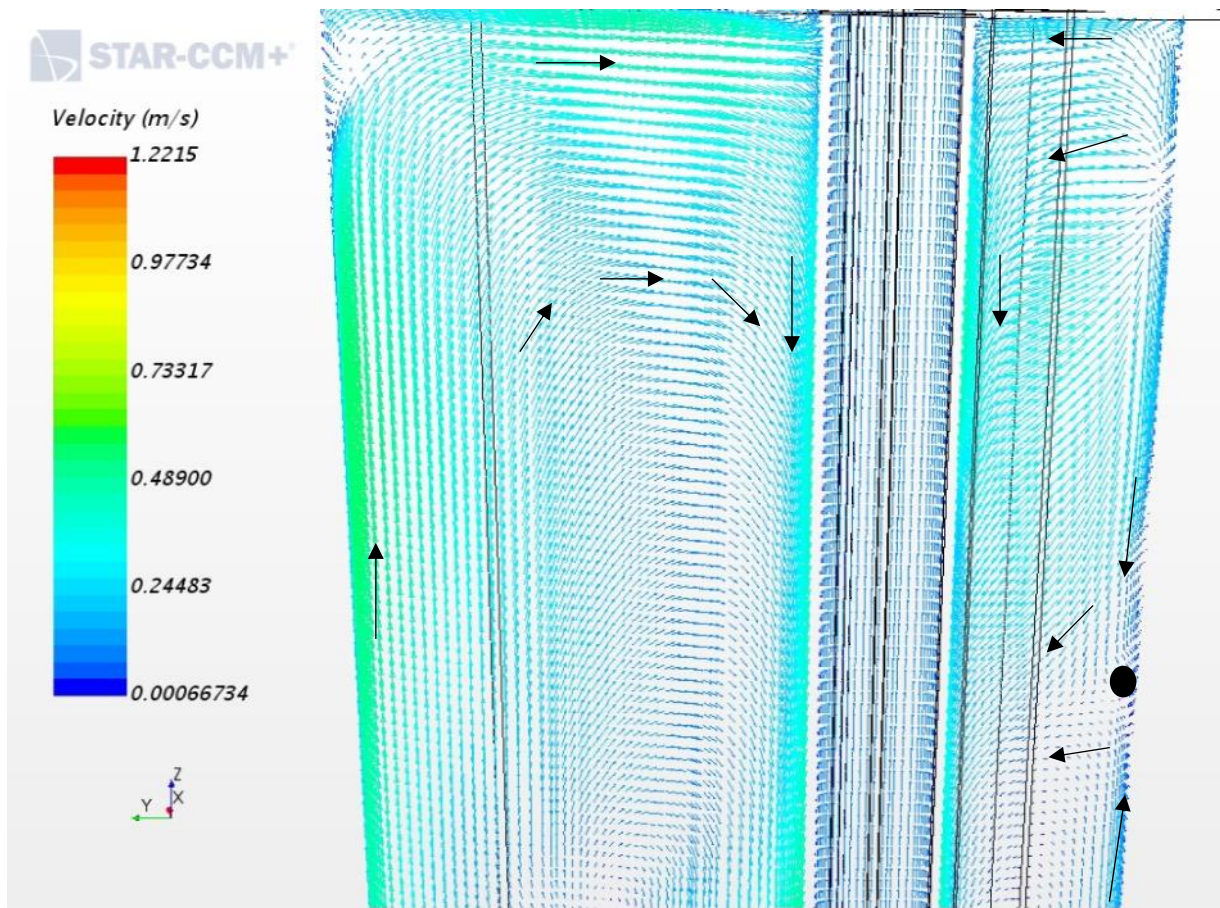


Figure 8.2-29. Velocity vectors of the air in the top part of the heated section.

Figure 8.2-30 show the velocity vectors on a plane that intersects the heated section between Riser 2 and 3. The figure focusses especially on the top part of the heated section as the bottom part was similar. The figure shows the flow path of the heated air as it passes by the riser tubes. The space between the riser tubes created a strong down draft, forcing the streams downwards as the fluid was transported between the front and back cavity. This was because of the cooling effect of the riser tubes causing the air density to rise, causing downward flow. The downward flow next to the risers persists for most of the elevation of the heated section.

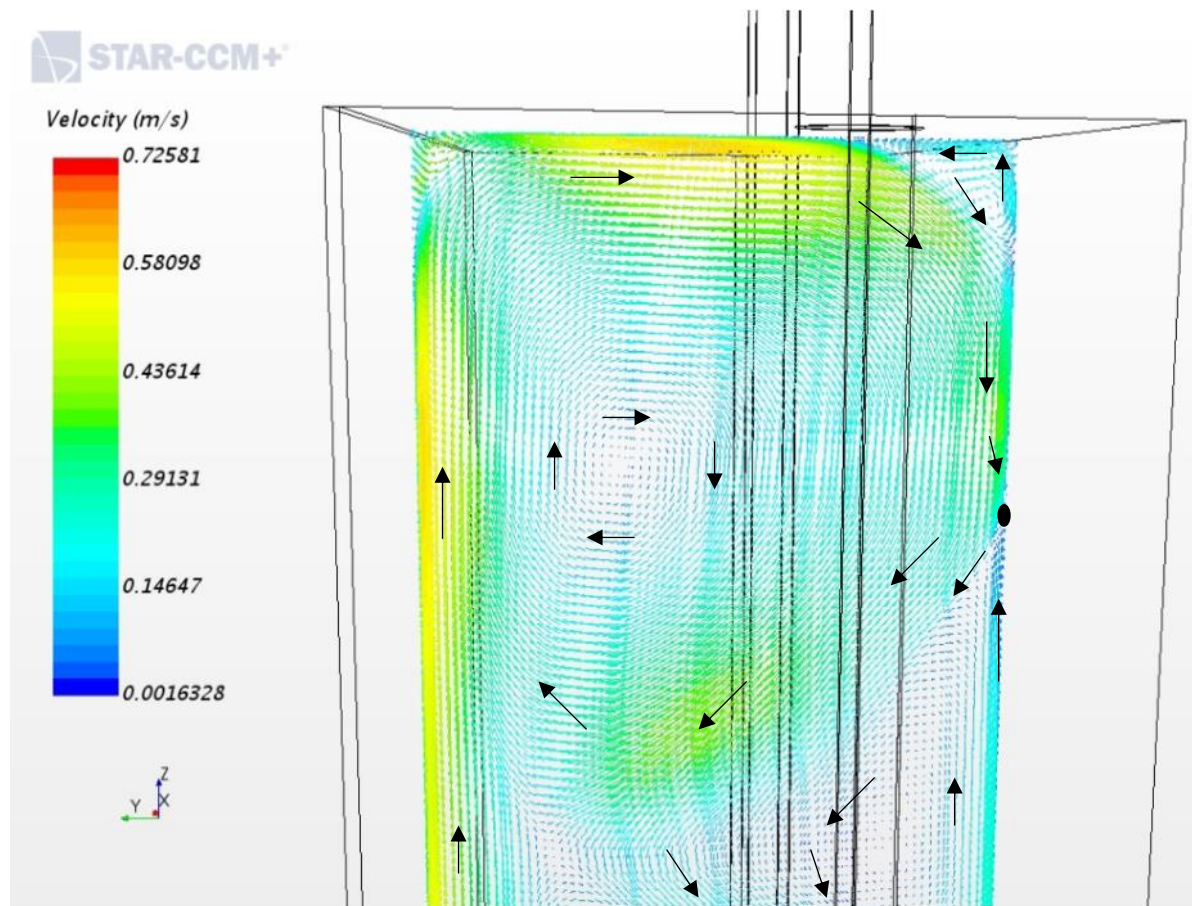


Figure 8.2-30. Velocity vectors in the top part of the front and back cavity.

The dot in Figure 8.2-30 shows a 'stagnation point' (approximate, since flow velocity in the z direction was very small) where the direction of the flow changed against the back wall. The flow direction was primarily upward due to the strong buoyancy forces against the back wall. At the location of the dot, however the direction changes due to recirculating flows that drive the air in the top part downwards.

Figure 8.2-31 shows the velocity vectors in the middle riser at four different elevations, as viewed on the x-z plane. The lowest elevation profile matches a laminar flow parabolic form, from which the subsequent profiles broadened out more across the cross-sectional area.

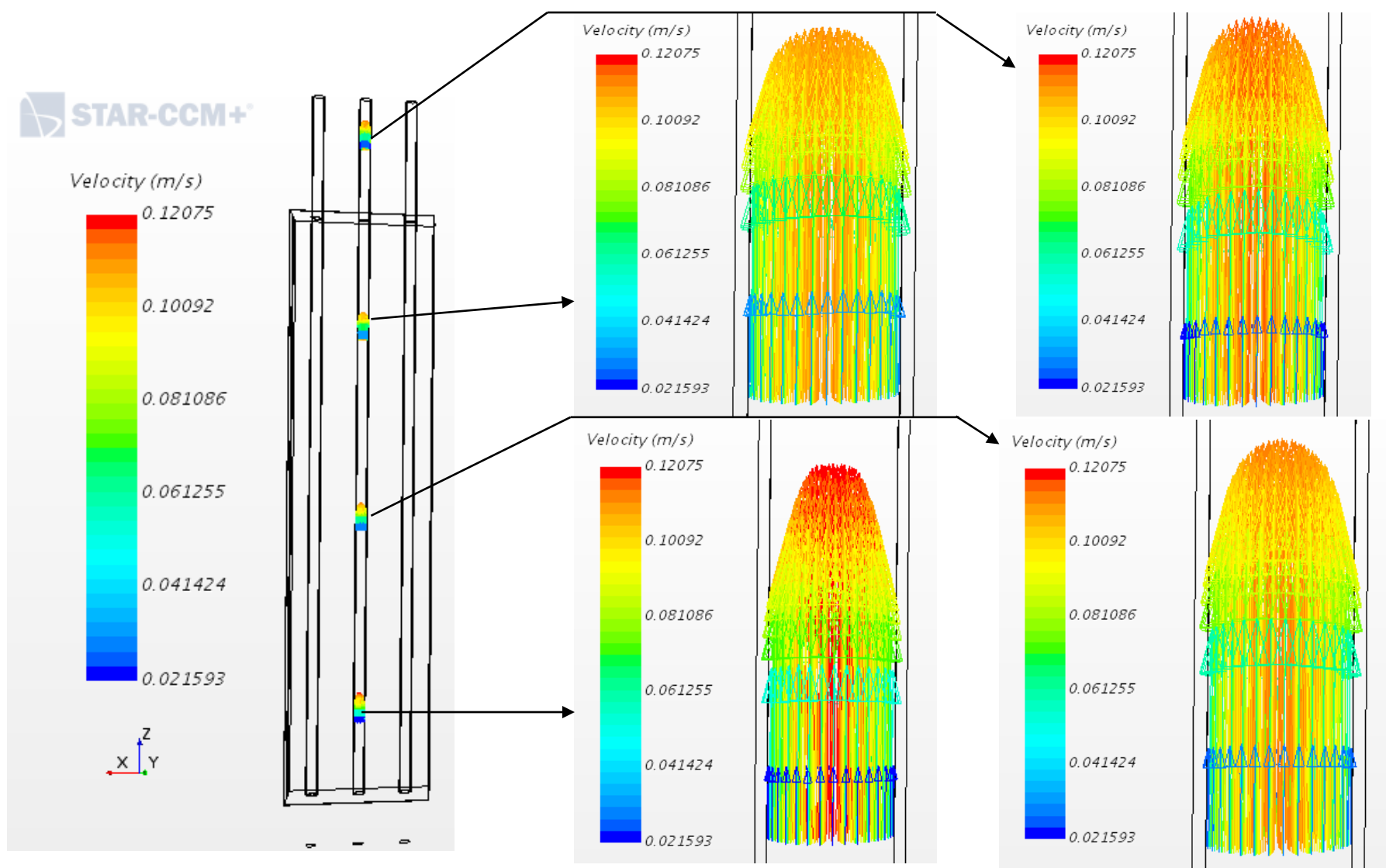


Figure 8.2-31. Velocity vectors at four elevations in the middle riser tube.

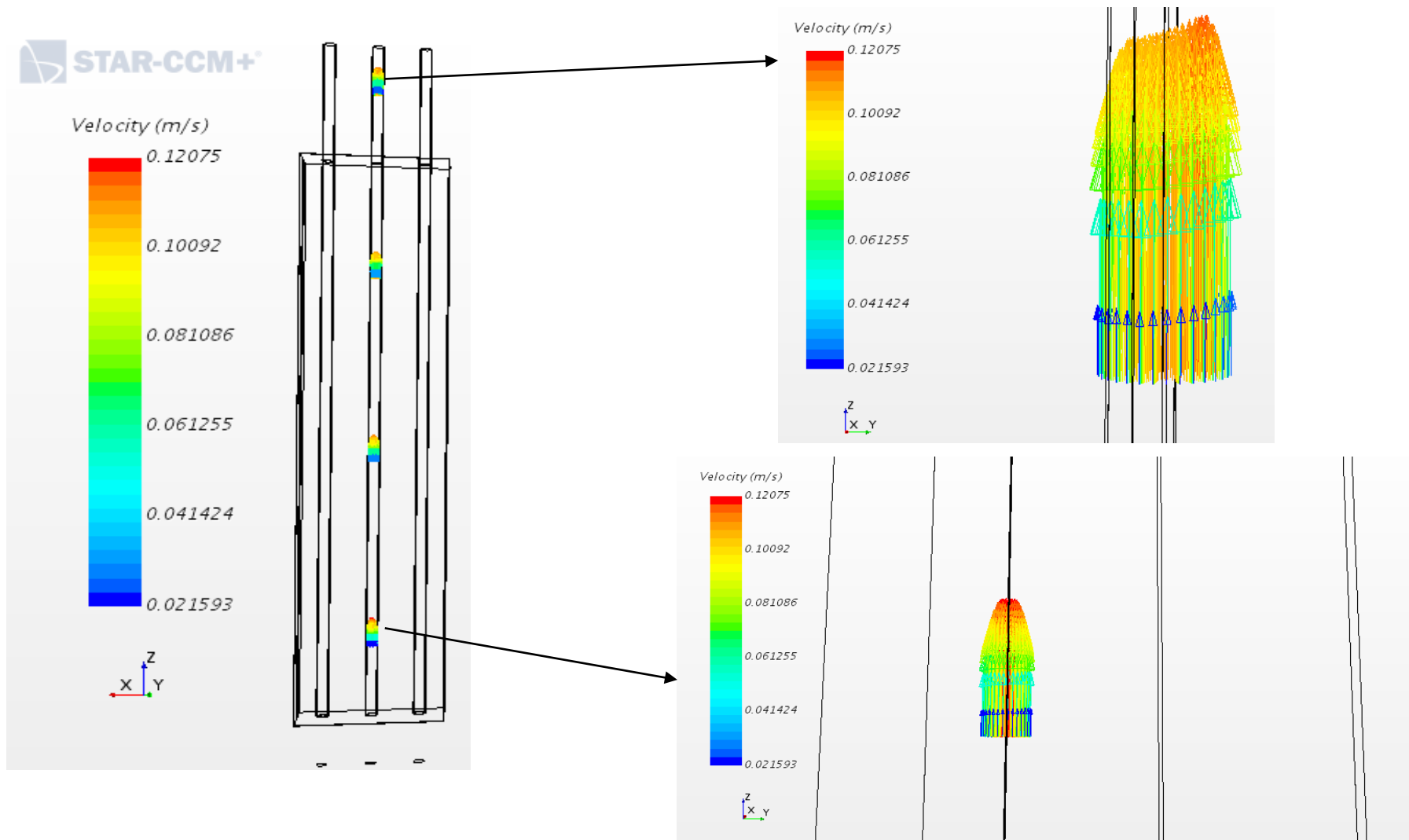


Figure 8.2-32. Velocity profiles in the middle riser (continued).

The fourth elevation's profile appeared "narrower" than the third, owing to the heating being present at the third elevation but absent at the fourth. It is also evident that the peaks that were caused by the fins in the previous configuration (see Figure 8.2-14) are now absent.

Figure 8.2-32 shows the velocity profiles of the lowest and highest elevations as viewed from the x-direction (on the y-z plane, i.e. the heater wall located to the right). The lowest elevation shows a velocity profile that was approximately symmetric. This was because the elevation was primarily affected by the inlet boundary conditions. The velocity profile at the highest elevation was asymmetric since it was heavily influenced by the buoyancy driven heating which was concentrated on the front side of the riser walls.

8.2.3.4 Heat transfer coefficients

The convection heat transfer coefficients were calculated for the inner surfaces of the riser tubes. The coefficients for the middle riser tube are shown in Figure 8.2-33 and is also representative of the coefficients of the other riser tubes which are therefore not shown. In contrast with the case with fins, the coefficients of the front side are higher than the other sides. The differences between the coefficients on the different sides are not as large as in the finned case (see Figure 8.2-17) owing to the more uniform temperature distributions across the cross-section of the walls.

Figure 8.2-34 shows the Aicher number as a function of elevation. The Aicher number falls well within the criteria for mixed convection and none of the walls show transition to free convection as with the case with fins (Figure 8.2-16).

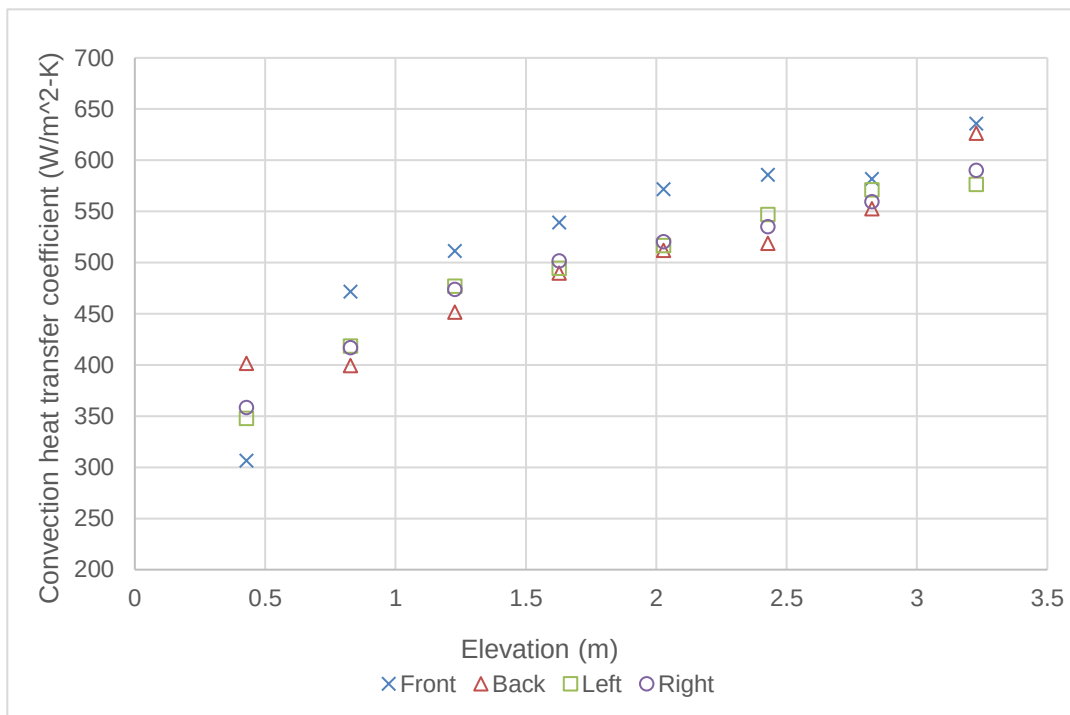


Figure 8.2-33. Convection heat transfer coefficient on the riser tubes' inner surfaces as a function of elevation.

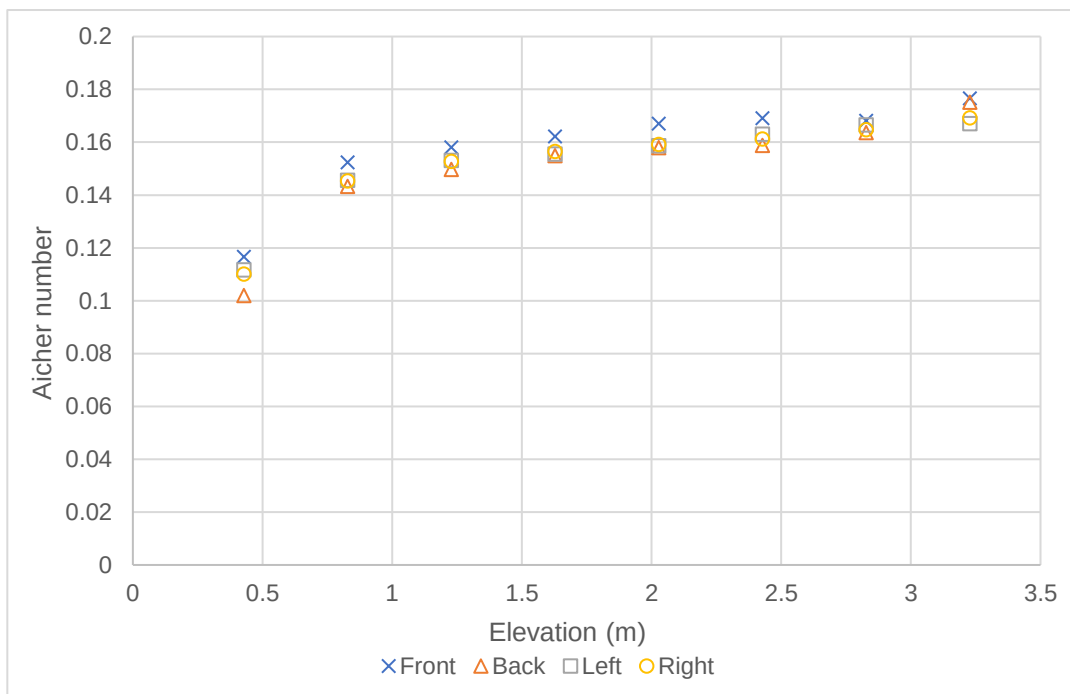


Figure 8.2-34. Aicher number as a function of elevation for the middle riser.

Figure 8.2-35 shows the Nusselt number ratio vs the Aicher number for Riser 2. The lowest elevation point on all four walls show the deterioration of the Nusselt number and subsequent points show the recovery of the Nusselt number towards the free convection regime.

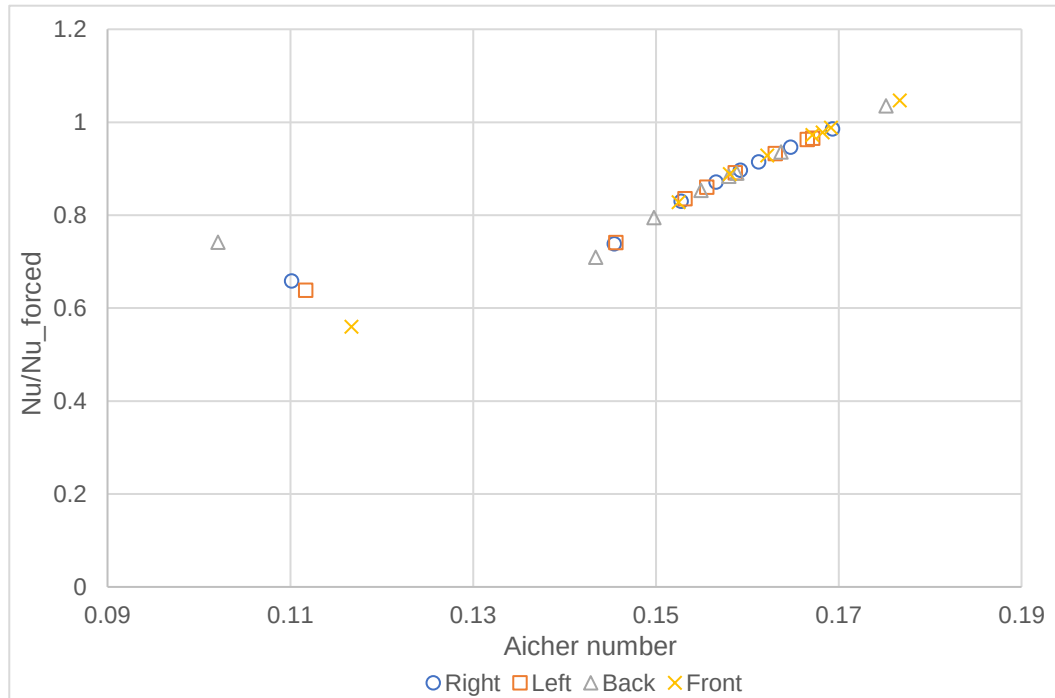


Figure 8.2-35. Nusselt number ratio vs. Aicher number for the middle riser.

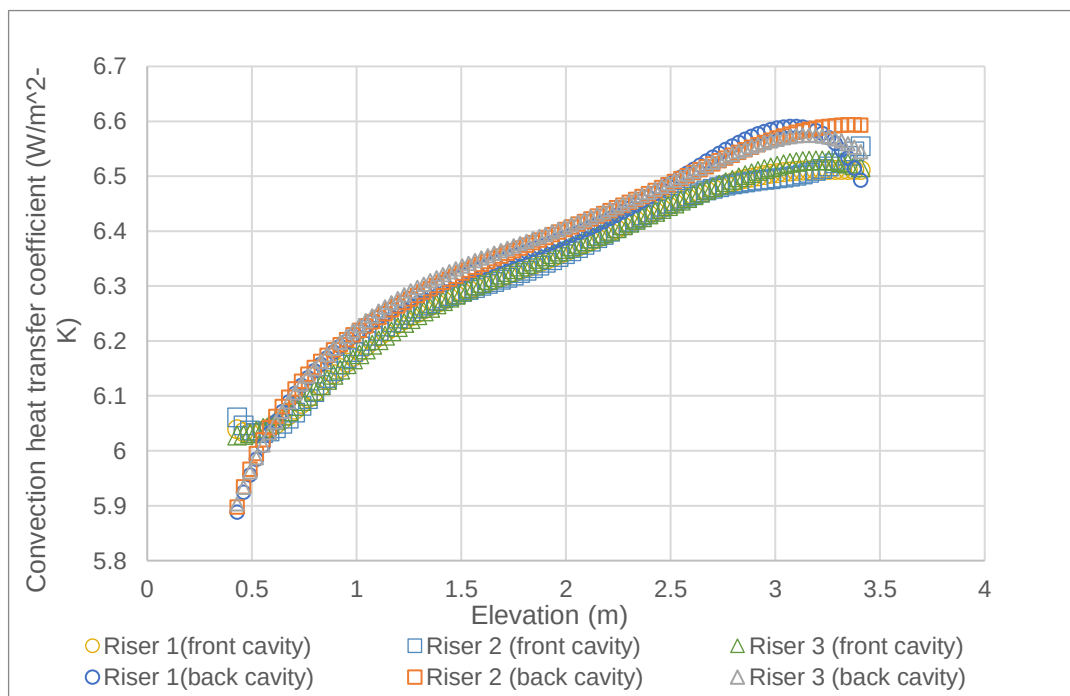


Figure 8.2-36. Convection heat transfer coefficients on the outer surfaces of the riser tubes as a function of elevation.

Figure 8.2-36 shows the calculated convection heat transfer coefficient on the riser tubes' outer surfaces in the cavity. The corresponding coefficients from the different sides were more uniform in value than the corresponding case with fins (see Figure 8.2-20). That was due to the surface and fluid temperatures around the cross section of the tubes' walls being more uniform than the case with fins. The coefficients at the bottom elevation on the back sides of the riser tubes were noticeably lower than the corresponding coefficients on the front sides. This was because both the surface and fluid temperatures were lower at this point. The front surfaces at the same elevation, in contrast, received direct heat from the heater wall by radiation, causing slightly higher wall temperatures and subsequently higher coefficients.

Figure 8.2-37 shows the convection heat transfer coefficients for the heater- and back wall. The coefficients show a gradual decrease with increasing elevation, since the fluid temperature approaches the wall temperature with increasing elevation resulting in lower coefficients. The surface and fluid temperatures are almost the same value at the back wall at 3.1 m to 3.3 m which resulted in the minimum values of the coefficients to be calculated at that location. Additionally, the switch from upward to downward flow against the back wall at these elevations resulted in a 'stagnation point', which was indicated by the dot in Figure 8.2-30.

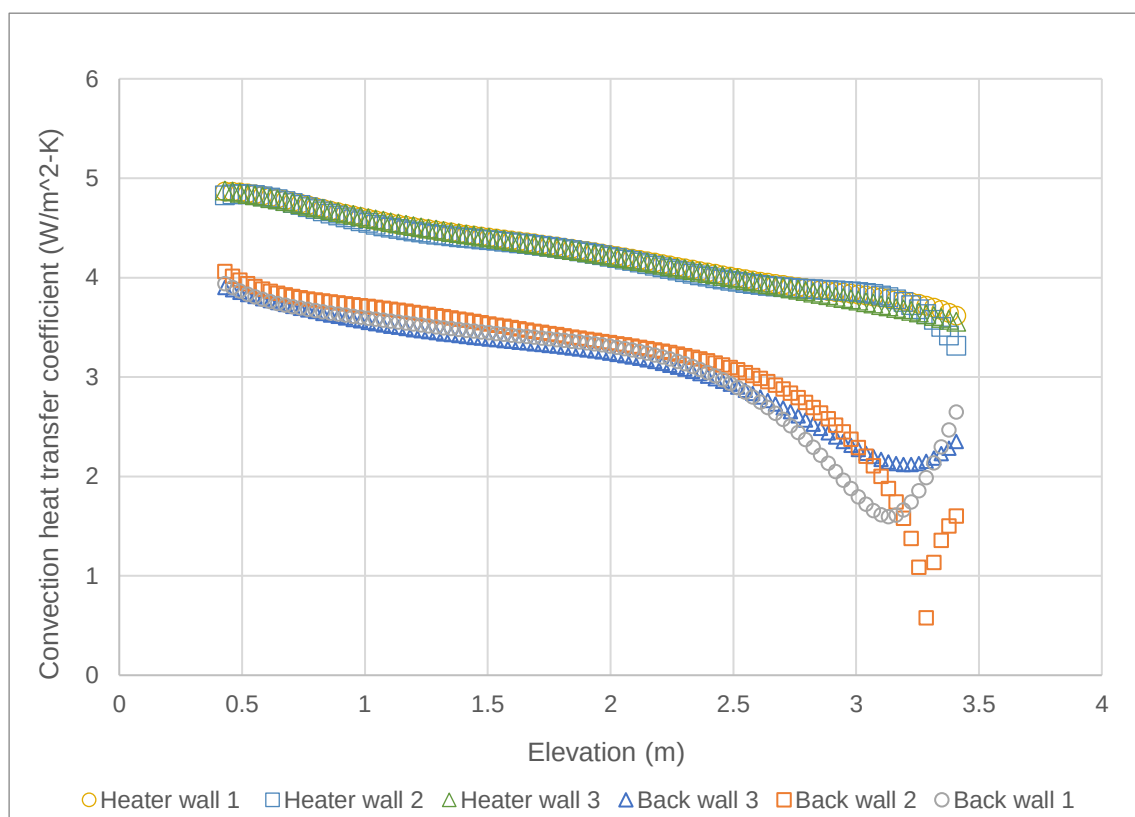


Figure 8.2-37. Convection heat transfer coefficient as a function of elevation on the heater- and back wall.

The 'stagnation point' in turn resulted in diminished convection heat transfer which explains the decrease in the convection heat transfer coefficients in this area.

The parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix D.3.

8.2.3.5 Conclusion

The case with circular riser tubes without fins was run using the same boundary conditions as the corresponding case with fins.

The temperature profiles and velocity vectors were investigated along with the distribution of heat transfer to each surface in the cavity. The convection heat transfer coefficients were calculated for the important surfaces in the cavity and for the inside surfaces of the riser tubes.

The velocity vectors were shown for the middle riser tube and provided some insight into which flow regime dominated the flow inside the riser tubes.

The omission of the fins from the geometry meant that the back part of the cavity was unshielded from direct heat transfer from the heater wall. The temperatures around the cross-sectional area of the riser tubes was more uniform between the different sides where the temperature probes were placed.

The radiation heat transfer to the riser tube walls in the back cavity increased notably, being more than seven-fold the transfer in the corresponding case with fins (see Table 8-1). The radiation heat transfer to the front of the riser tube walls was approximately 16% lower than in the corresponding case with fins (see Table 8-1).

The Aicher numbers calculated for each side of the inner surfaces of the riser tubes indicated that mixed convection occurred throughout the elevation of the riser tubes in the heater box.

8.2.4 Square tubes with fins

Figure 8.2-38 shows the geometry of the square tubes with fins variation, with the naming convention as referred to in the discussion of the results. The cross-sectional area of the tubes was the same as the circular tubes and the same boundary conditions were used.

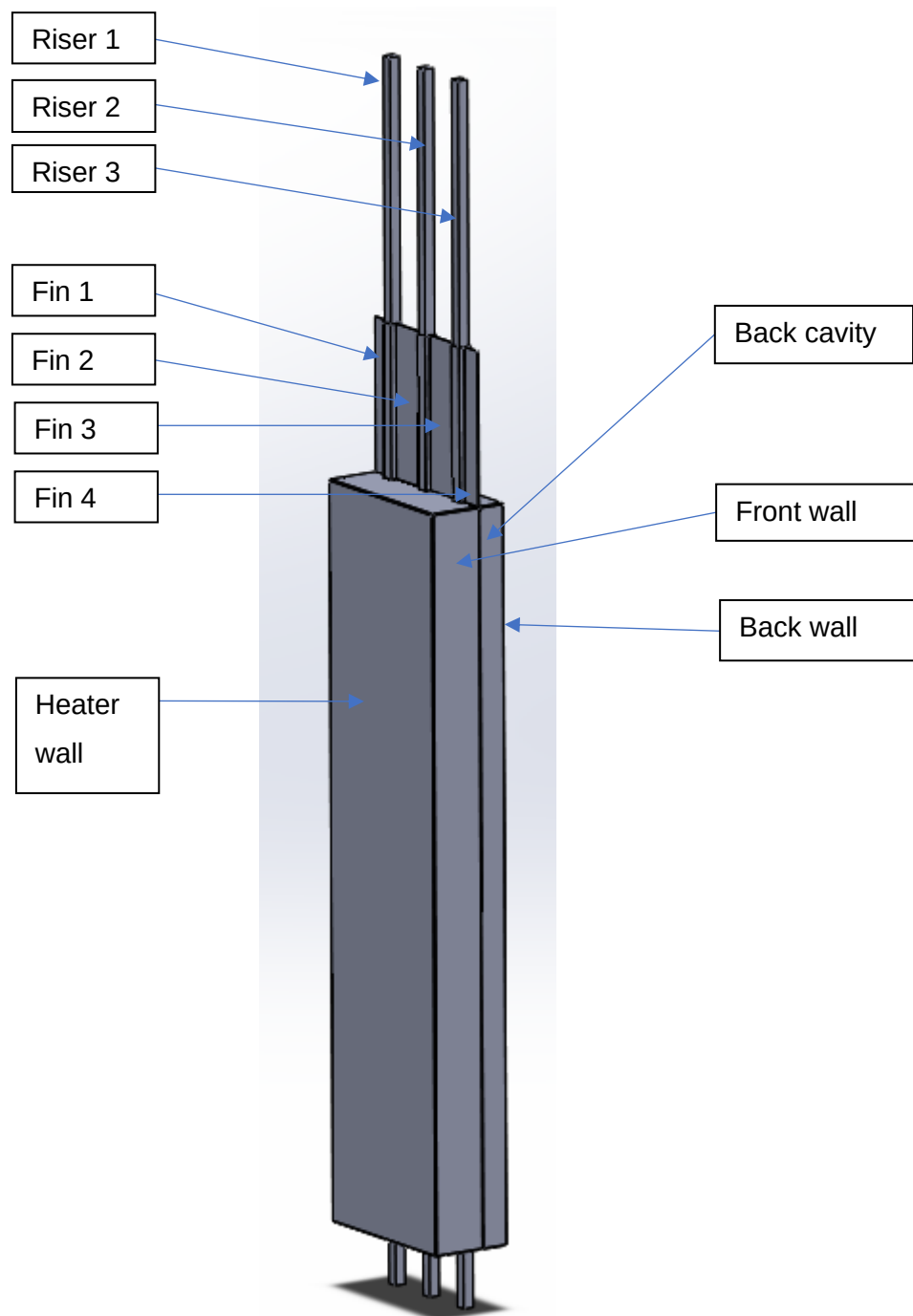


Figure 8.2-38. Geometry and naming convention as referred to in the discussion of the results.

8.2.4.1 Heat fluxes

Table 8-3 shows selected results for the outer surfaces of the riser tubes. Table 8-3 distinguishes between the surface average of the total and radiative heat flux entering via the front and back cavity. The ratio of radiative/total heat transfer was also calculated and is shown in Table 8-3. The negative signs indicate heat transfer from the fin into the back cavity. The positive sign at the back side of the riser indicates a nett transfer from the back cavity to the tubes. The middle riser tube received 2.7% higher heat flux than the tubes on the sides, which is slightly lower than the case with circular tubes ($\approx 5\%$).

The geometry of the square tubes increased the surface area and was 15% larger which resulted in a higher radiative heat flux on the outer surfaces of the riser tubes. The surface area exposed to the cavities for the circular tubes case was 1.59 m^2 compared to 1.84 m^2 for the square riser tubes. The area of the fins was also larger by approximately 3%. The integral of the heat flux on the outer surfaces of the riser tubes was larger than in the corresponding circular tubes case (see Table 8-1) due to the larger surface area of the square tubes.

The average heat flux, as well as the radiative heat flux, was however lower than the circular tubes case. This was because the sides of the riser tubes received less radiative heat since they run perpendicular to the heater wall. The average heat flux on the total outer surface was therefore decreased by the lower flux values on the sides. In the case with circular tubes the distribution was more even, leading to higher average heat fluxes.

Note that for example the front cavity side includes the front surface of the riser tube and roughly $\frac{1}{2}$ of the left and right sides that are exposed to heat transfer in the front cavity.

Table 8-3. Selected results for the riser tubes.

	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Derived radiation/total heat transfer %	
Location	Front cavity side	Back cavity side	Front cavity side	Back cavity side	Front cavity side	Back cavity side	Front cavity side	Back cavity side
Fin 1	3544.47	-377.13	2968.97	-220.87	986.81	-105.00	83.76	58.57
Fin 2	3849.47	-463.08	3318.99	-275.21	2143.46	-257.85	86.22	59.43
Fin 3	3838.00	-459.16	3324.19	-272.90	2137.07	-255.67	86.61	59.43
Fin 4	3571.29	-380.21	2971.62	-222.88	994.28	-105.85	83.21	58.62
Riser 1	4719.02	784.00	3489.31	404.96	1448.90	240.72	73.94	51.65
Riser 2	4847.84	791.89	3609.64	409.05	1488.46	243.14	74.46	51.65
Riser 3	4723.50	783.30	3492.49	405.25	1450.28	240.50	73.94	51.74

8.2.4.2 Temperature distributions

Figure 8.2-39 shows the water temperatures as a function of elevation taken on probes in the centres of the tubes' cross sections. The temperatures show a linear increase with elevation within the heated section. The temperature increases slightly above the heated section owing to heat transfer from the riser walls as well as from the fluid in the warmer boundary layer.

Figure 8.2-40 shows the wall temperatures as a function of elevation for the middle riser tube, which is representative of the other riser tubes. Similar to the corresponding circular tubes case (for reference see Figure 8.2-5), the sides have a higher temperature, owing to the heat transfer from the fins. The temperatures drop significantly after exiting the heated section at ≈ 3.1 m.

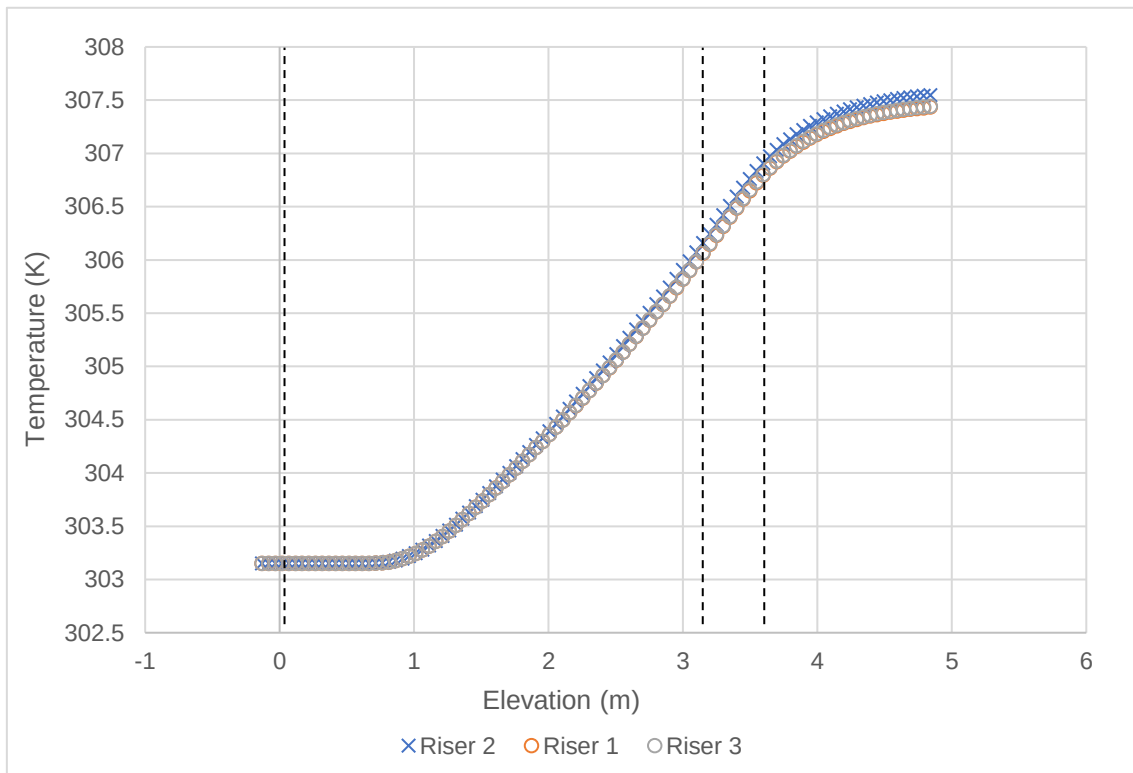


Figure 8.2-39. Riser water temperature distributions.

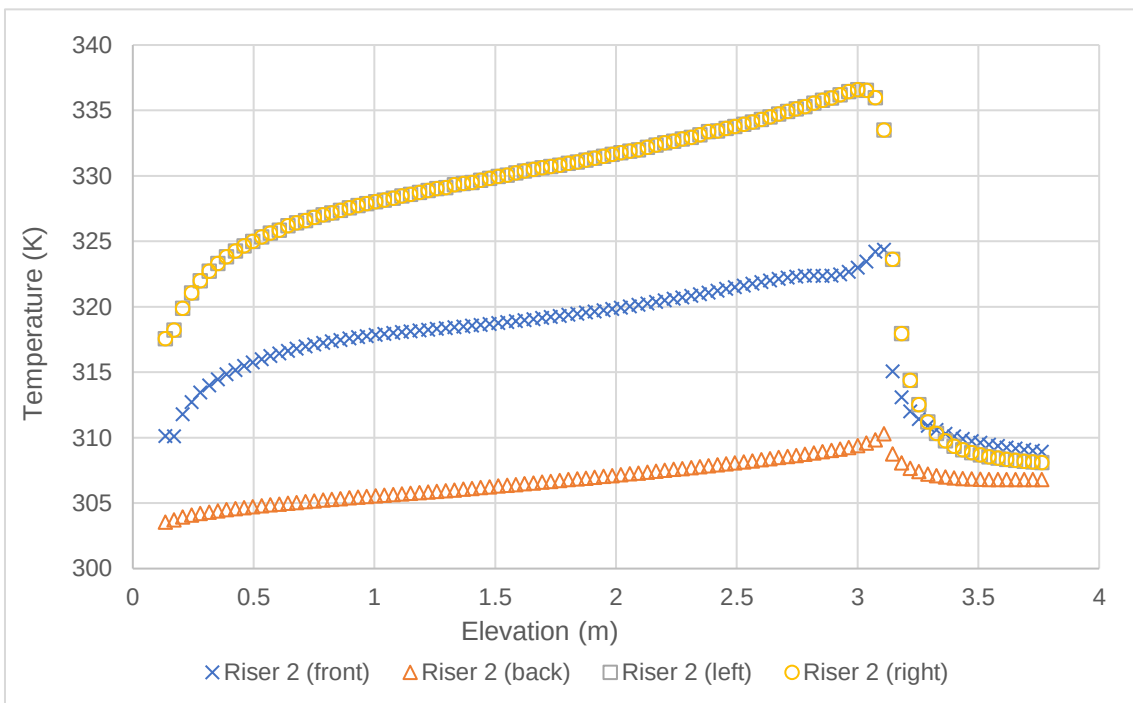


Figure 8.2-40. Riser 2's surface temperatures as a function of elevation.

Figure 8.2-41 shows the temperature distribution within the walls of the fins. The temperatures in the inner fins were higher, since the line probes used to obtain the temperature distributions were placed in the middle of the fins. The maximum temperatures for the side fins were against the side walls, while the maximum temperatures for the middle fins were in the middle of these fins.

Figure 8.2-42 shows the temperature contours at the top elevation of the heated section. The figure shows the maximum temperatures occurred in the middle of Fins 2 and 3 (inner fins). The cooling effect of the riser tubes is also visible. The heat also propagated upwards in the walls above the heated section but was swiftly cooled down due to the cooling effect of the risers. Above the heater box the temperatures within the left and right sides of the riser tube wall remained higher than the front wall. This was due to the heating effect of the fins through conduction.

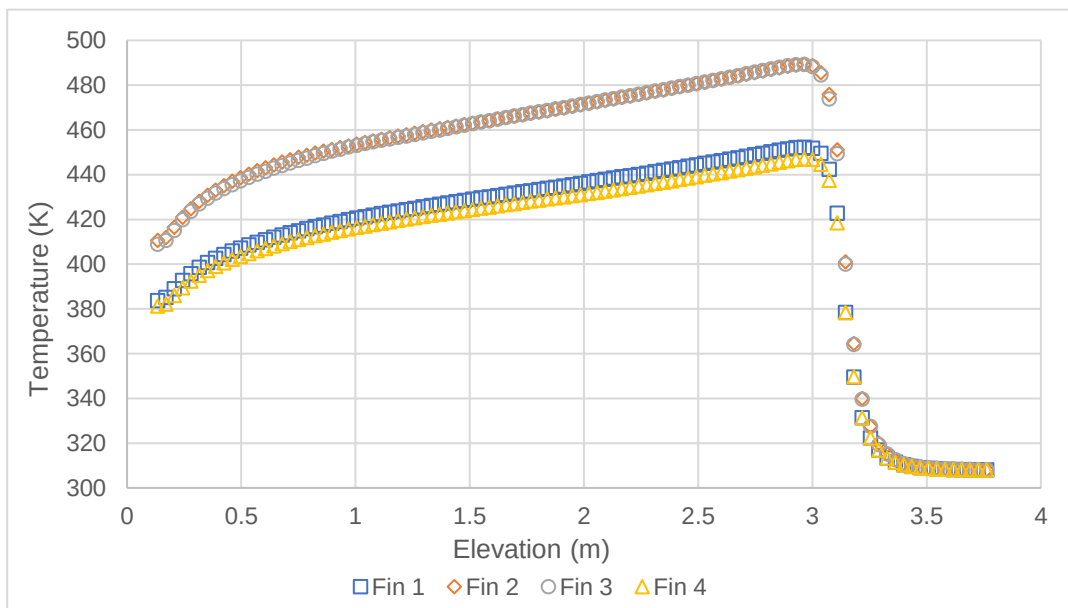


Figure 8.2-41. The temperature distributions in the fin walls.

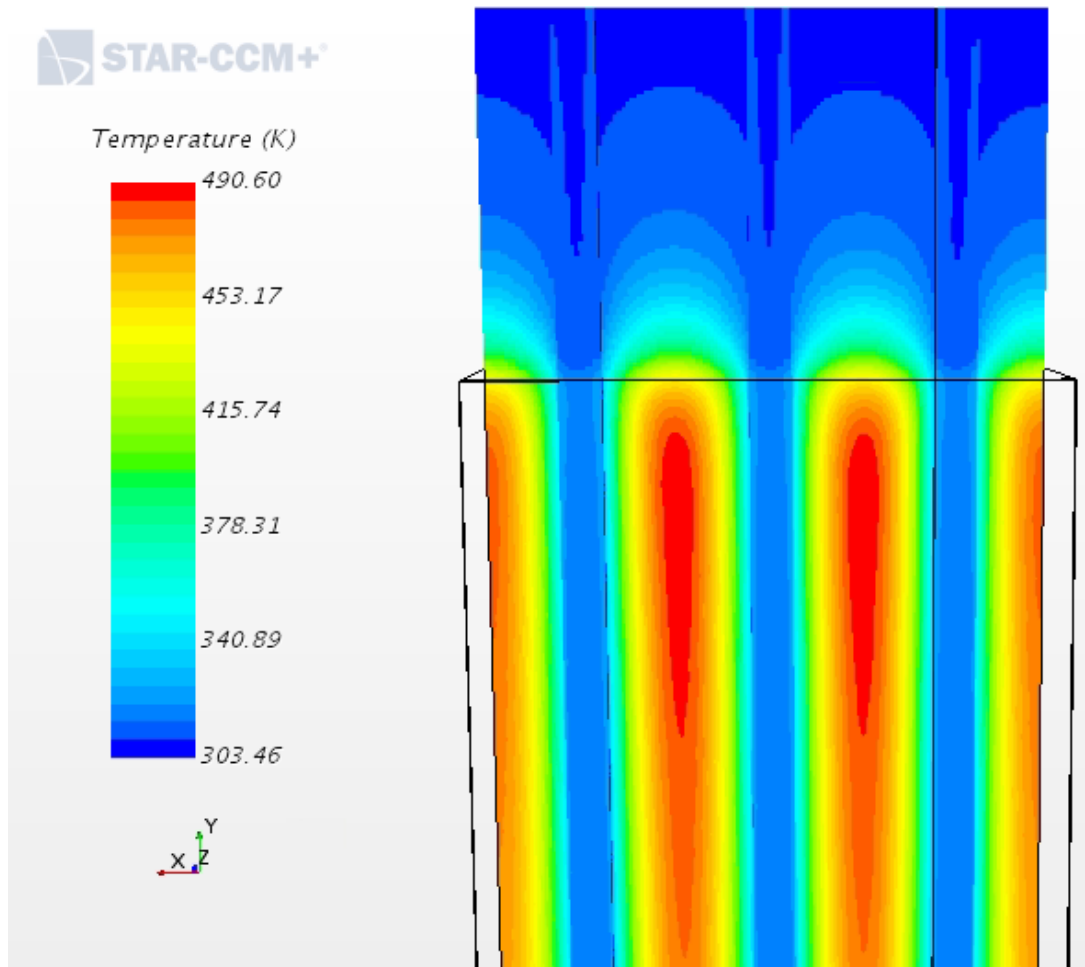


Figure 8.2-42. Temperature contour on the front surfaces of the riser tubes and fins.

Figure 8.2-43 shows the temperatures on the heater- and back wall as a function of elevation. The temperatures show an overall increase over the elevation. The temperatures in the heater wall were 64 – 74 % higher than the corresponding temperatures on the back wall. This is comparable with the 64 – 73 % of the corresponding circular tubes case.

Figure 8.2-44 shows the air temperature distribution as a function of elevation for the front and back cavity. The front cavity air had temperatures that were 20 – 40 % higher than the back-cavity air, which is similar to the corresponding circular tubes case (25 – 47 %). The very top elevation of Figure 8.2-43 and Figure 8.2-44 show similar behaviour. In the front cavity the rising heated air collects at the top of the cavity, resulting in the hotspot in the air and wall temperatures that were observed. In the back-cavity the fluid flow patterns guided the air downwards, transporting the heat away from the top elevation causing the plateau on the back wall in Figure 8.2-43 and the cold spot in Figure 8.2-44.

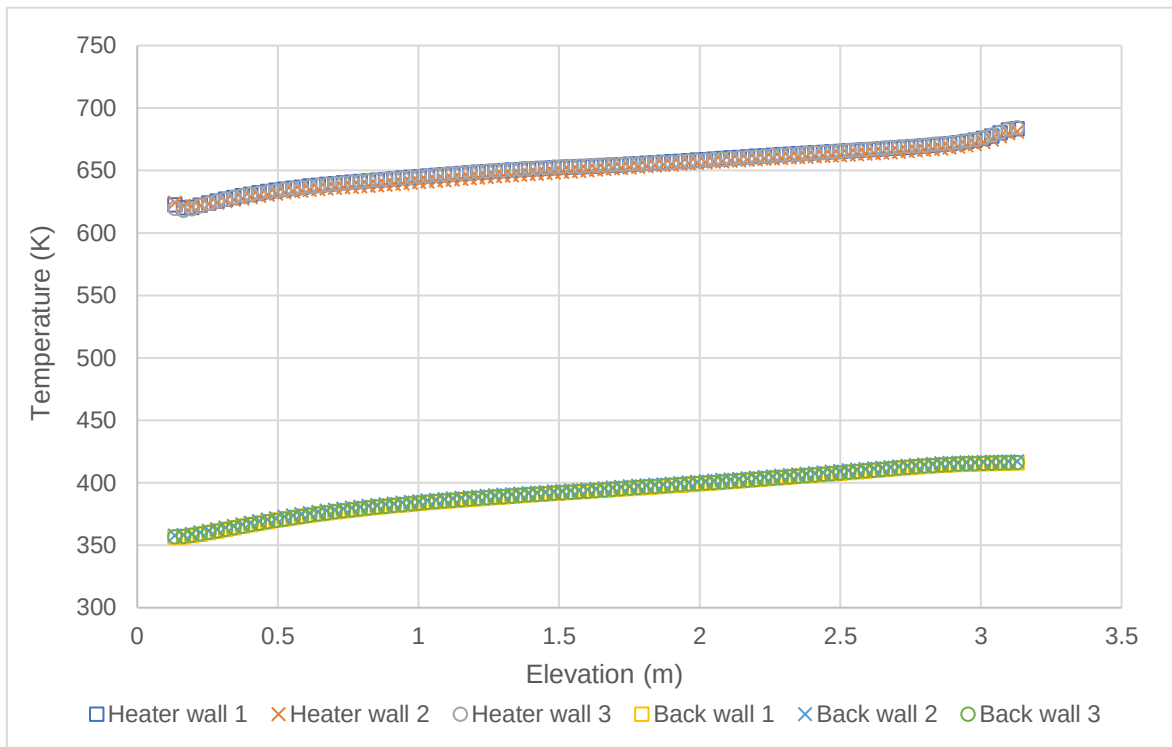


Figure 8.2-43. Heater- and back wall temperature distributions.

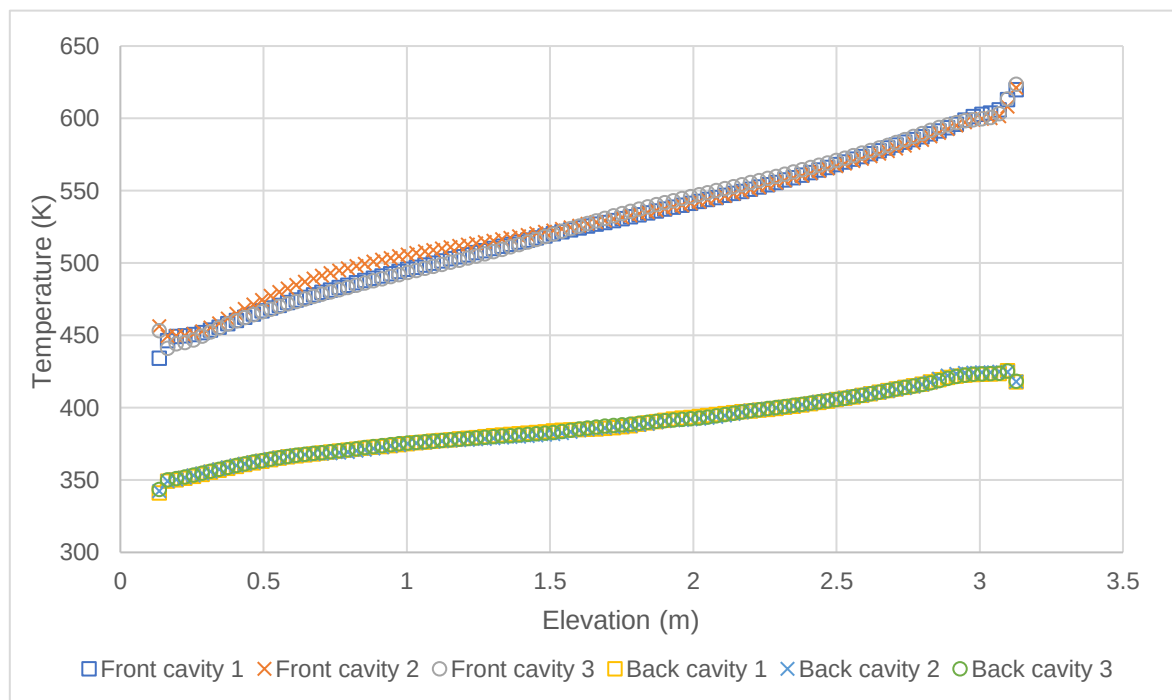


Figure 8.2-44. Air temperature distributions in the front and back part of the cavity.

8.2.4.3 Velocity vectors

Vector scenes were extracted from the results to investigate the dominating flow patterns that occurred in the cavities.

Figure 8.2-45 shows the velocity vectors on a section plane taken through the centreline of the second riser, perpendicular to the heater wall. Arrows were added to indicate the dominant flow direction on the plane for the front and back cavity. The flow was upwards in the vicinity of the heater wall and downwards in the vicinity of the riser tubes in the front cavity. The dominant flow direction was downwards in the back cavity on the section plane. The heated air flowing from the heater wall to the riser tubes separated at the topmost elevation against the riser tube, forming a recirculation cell as indicated.

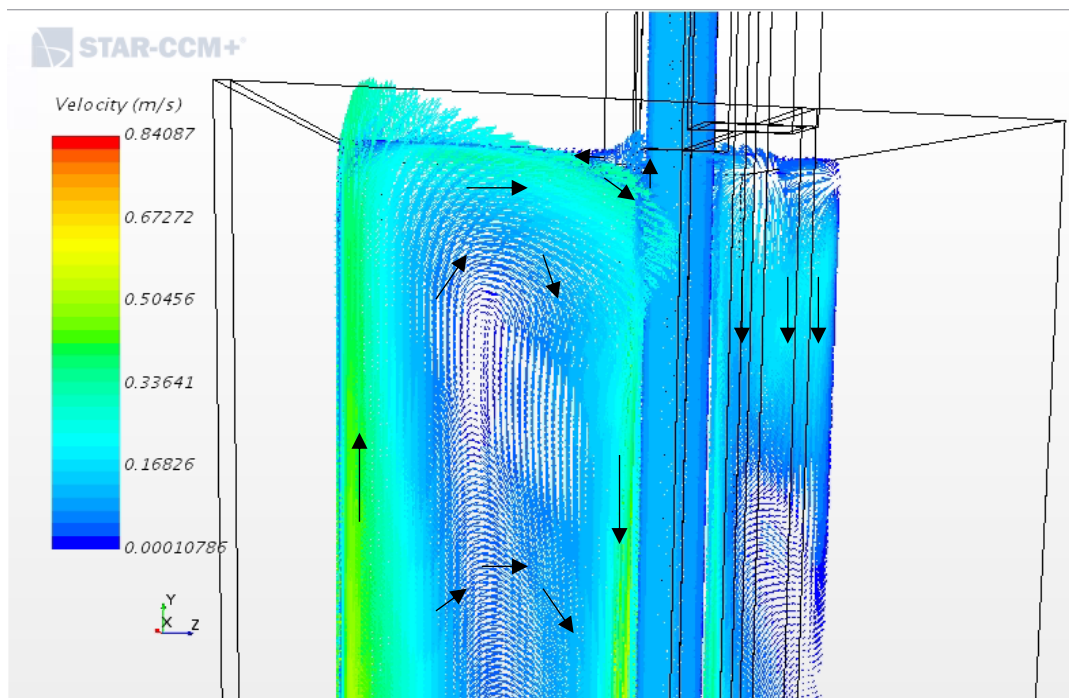


Figure 8.2-45. Velocity vectors in the front and back cavity for the case with square tubes with fins.

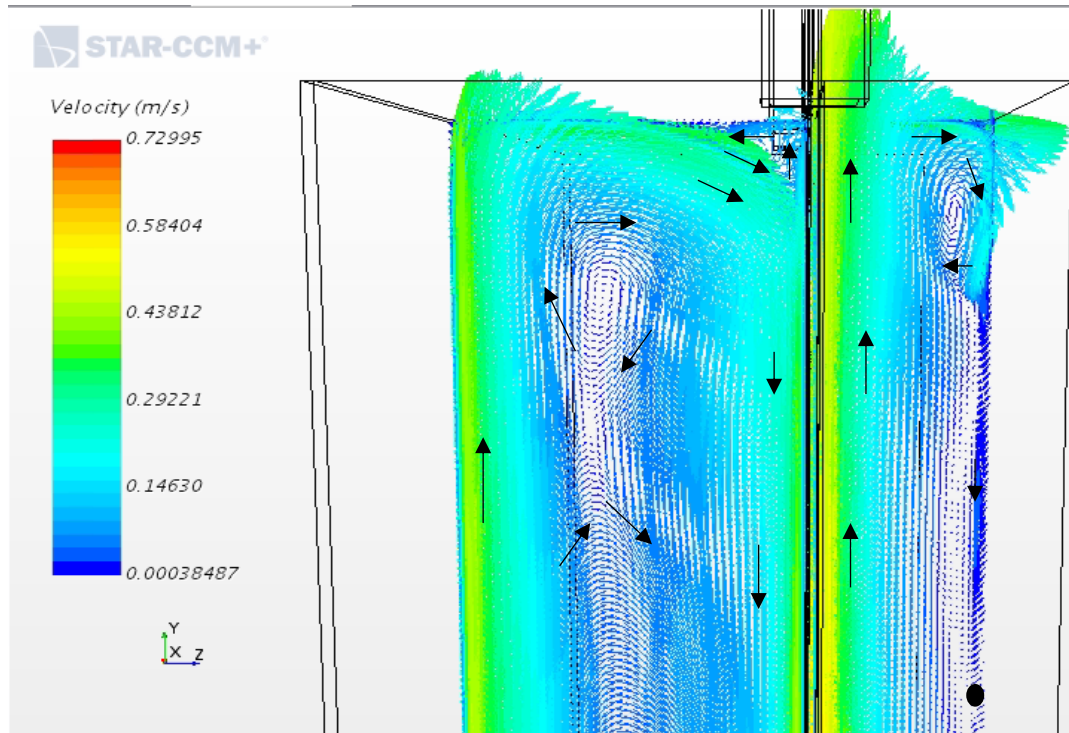


Figure 8.2-46. Velocity vectors in the front and back cavity (continued).

Figure 8.2-46 shows the velocity vectors taken on another section plane that was placed in between Riser 2 and 3, perpendicular to the heater wall. The figure shows dominating upward flow next to the heater wall, downward flow on the front surface of the fin and upward flow on the back surface of the fin. Three recirculation cells are visible on the plane. Two in the front cavity and one in the back cavity. The recirculation cell formed at the topmost elevation against the fin similarly to the cell next to the riser tube in Figure 8.2-45.

The dot in Figure 8.2-46 shows the location where the flow direction changed from upwards to downwards flow, forming a 'stagnation point' (approximated since the velocity in the y-direction was very small) with respect to the y-direction.

Figure 8.2-47 shows the velocity vectors at four different elevations in the middle riser as viewed on the x-y plane. The velocity profile at the lower elevation showed a profile that resembled a laminar flow profile. The profile broadened across the cross-sectional area to the second elevation and resembled a more turbulent profile. At the third and fourth elevation the buoyancy effects from the hot side walls, caused by the presence of the fins, caused the velocity peaks.

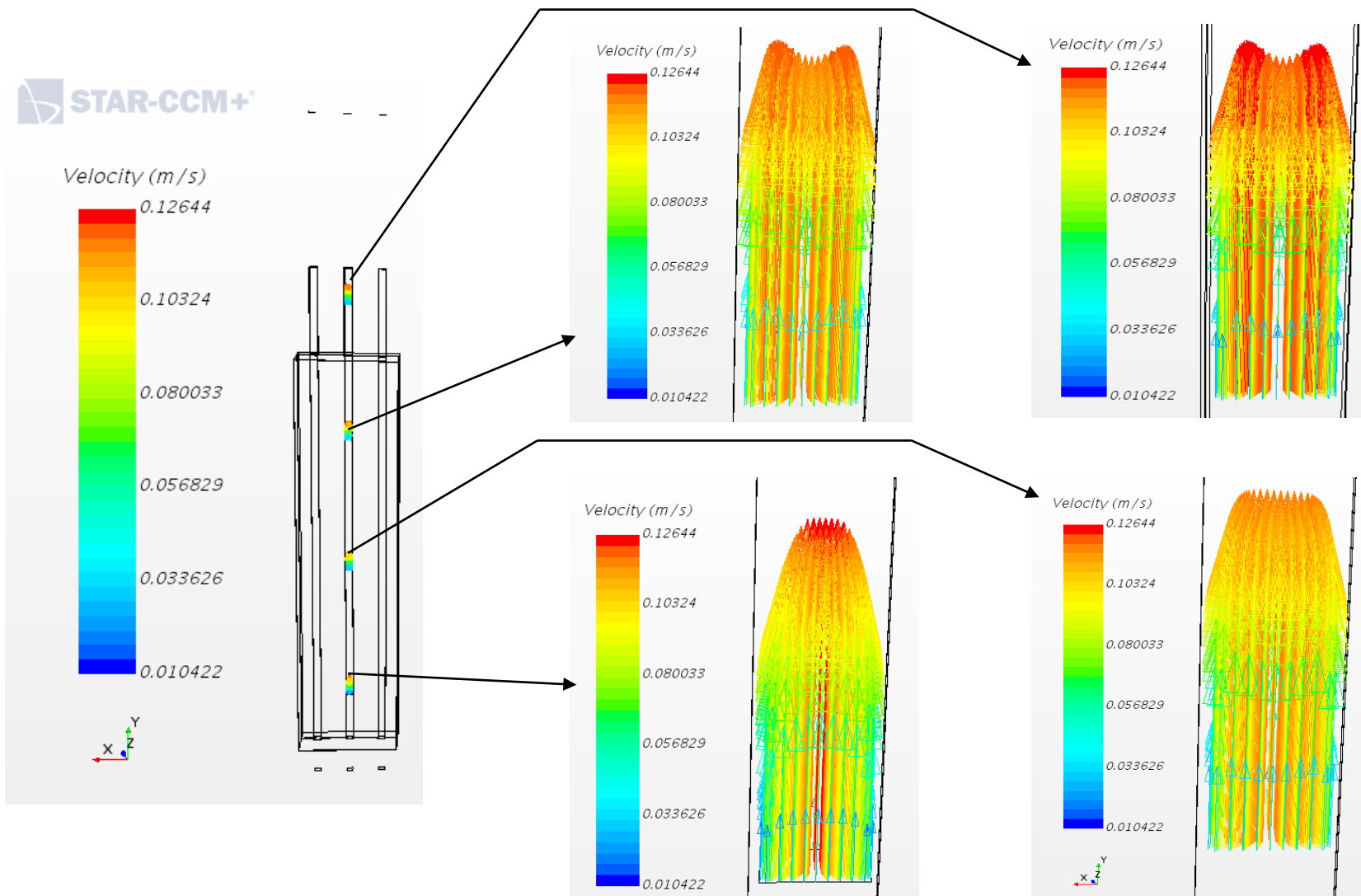


Figure 8.2-47. Velocity vectors at four elevations in the middle riser.

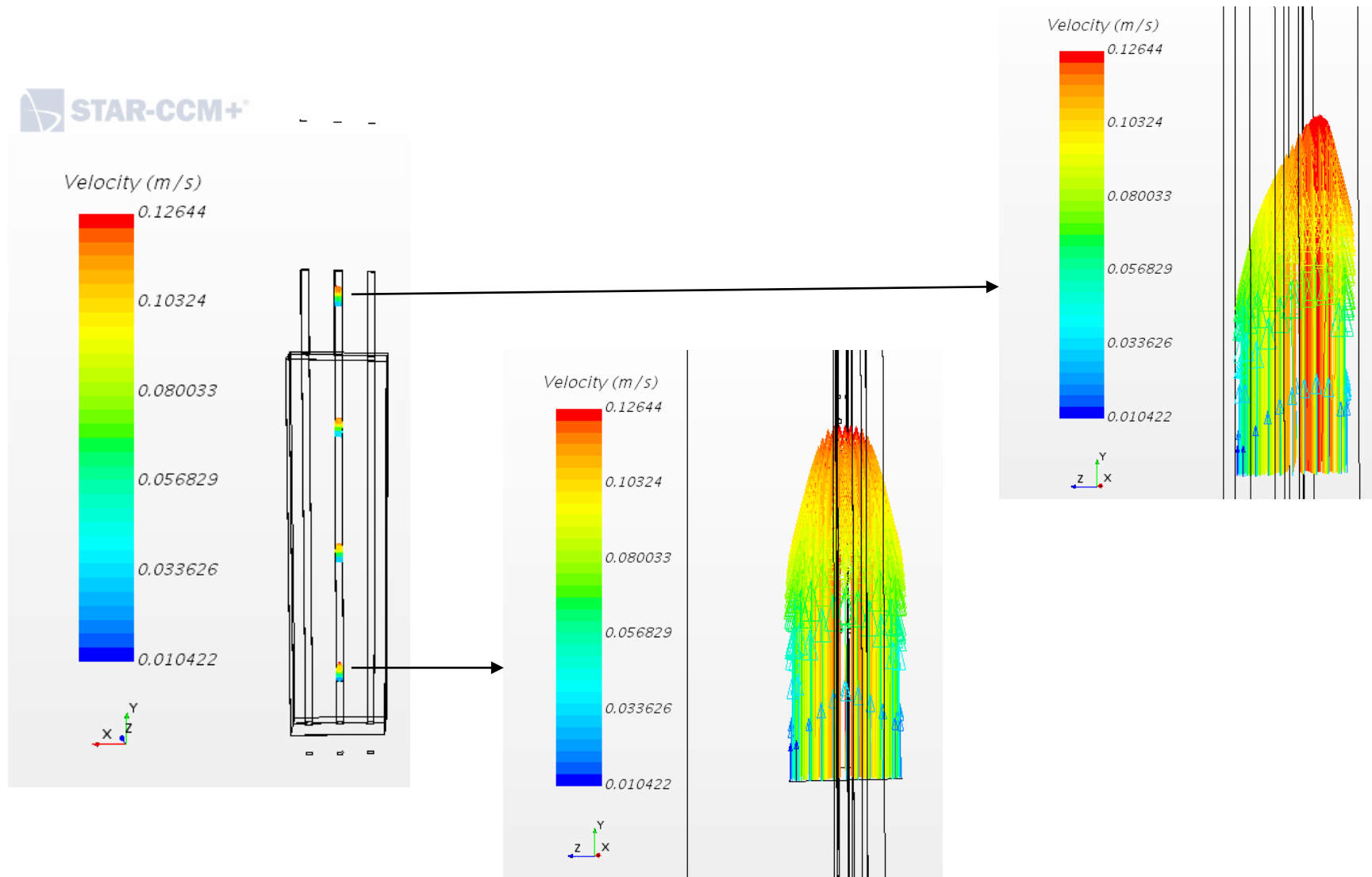


Figure 8.2-48. Velocity vectors in the middle riser (continued).

Figure 8.2-48 shows the velocity profiles in the middle riser tube as seen from the x-direction (i.e. on the y-z plane). At the lower elevation the profile was symmetric as it was primarily influenced by the inlet boundary condition. At the highest elevation the profile was asymmetric, since the buoyancy effects were strongly influenced by the asymmetric heating. The peaks seem to be near the fins and front and are thus associated with the hot corners mentioned in the discussion of Figure 8.2-57.

8.2.4.4 Heat transfer coefficients

Figure 8.2-49 shows the calculated convection heat transfer coefficient for the inner surface of the middle riser tube. The coefficients show the same trends as the corresponding heat transfer coefficients in the case of circular tubes, with the sides having the highest coefficients and the back having the lowest. Also similar to the corresponding circular case the coefficients show an overall increase over the elevation for the front and sides and a decrease for the back side.

Figure 8.2-50 shows the Aicher number calculated as a function of elevation. The graph shows that the front and back sides of the inner surface of the riser tube were within the mixed convection regime. The left and right sides however transitioned to the free convection regime at 0.9 m which caused the increase in the convection heat transfer coefficients at that elevation in Figure 8.2-49.

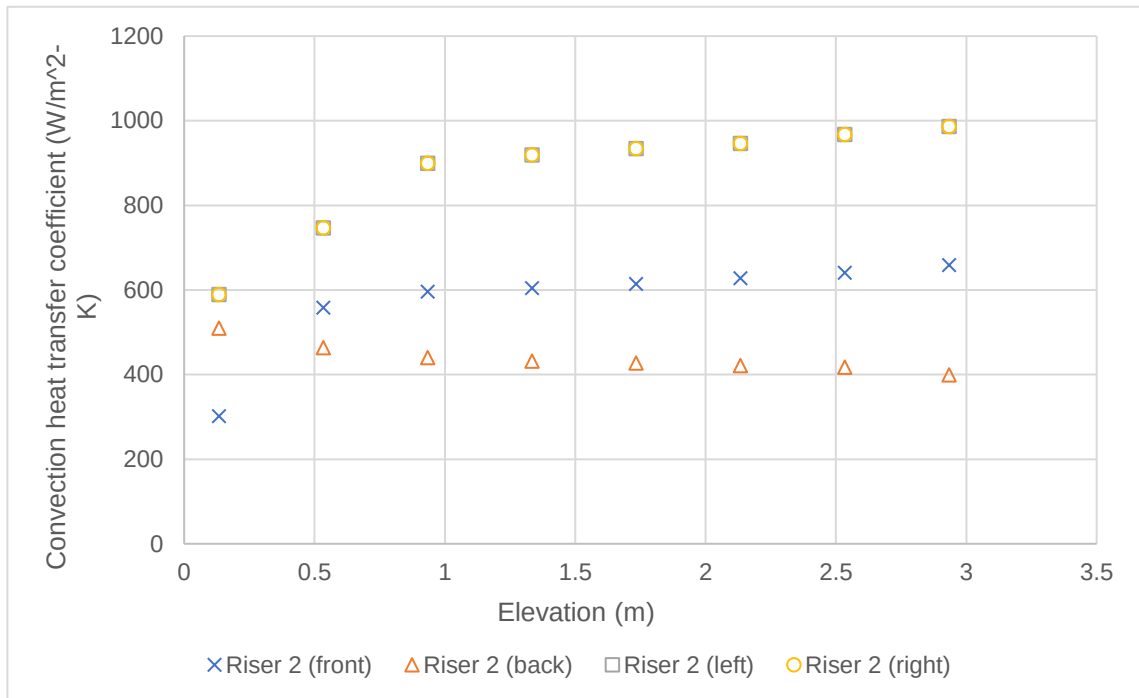


Figure 8.2-49. Convection heat transfer coefficient for the inner surfaces of the inner riser tube.

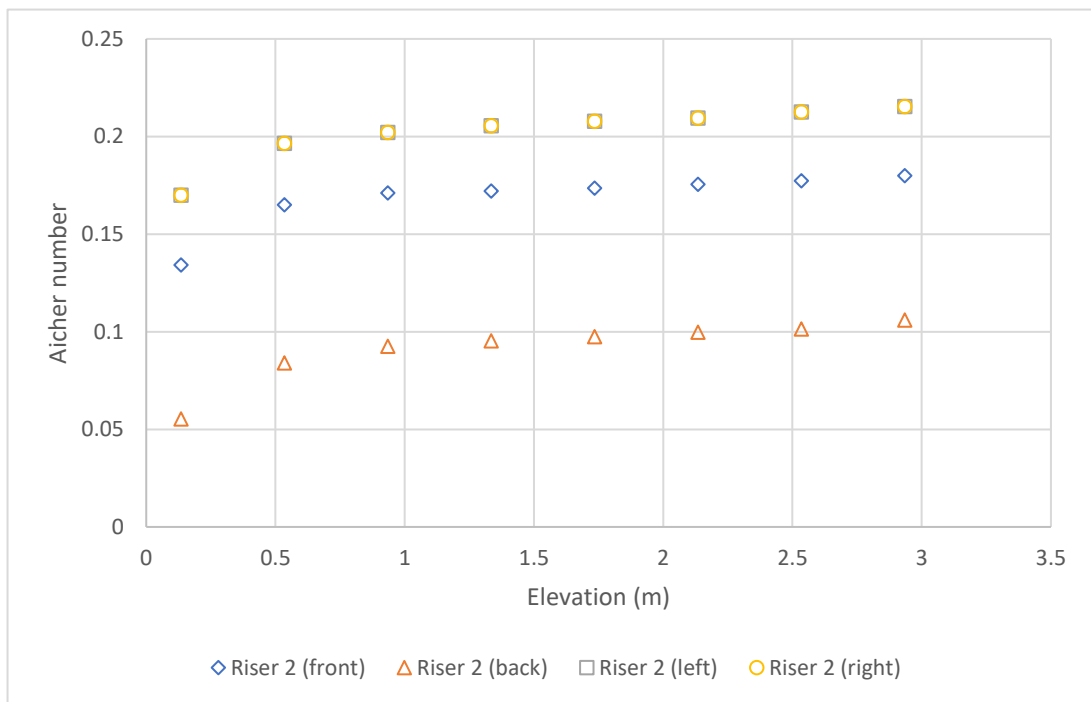


Figure 8.2-50. Aicher number as a function of elevation.

Figure 8.2-51 shows the convection heat transfer coefficient for the heater- and back wall as a function of elevation. The coefficients show an overall decrease with elevation, similar to the corresponding values in the case with circular tubes. The elevation at approximately 2.6 m showed lower coefficients for the back wall, since the fluid and surface temperatures were almost the same value. Additionally, a 'stagnation point' (approximated) existed at this location (as indicated by the dot in Figure 8.2-46) where the flow direction shifted between upward and downward flow. The lower velocity at this location diminished the convection heat transfer which explains the decrease in the convection heat transfer coefficients against the back wall.

Figure 8.2-52 shows the convection heat transfer coefficients on the outer surfaces of the riser tubes. The coefficients for both sides show an overall increase over the elevation. On the back-cavity side, the coefficients at the bottom elevation are especially low since both surface and fluid temperatures at this location were low. At the topmost elevations on the back-cavity side the temperature shows a plateau. This was possibly due to the decrease in the temperature of the riser tube walls as heat escaped upwards out of the heated section.

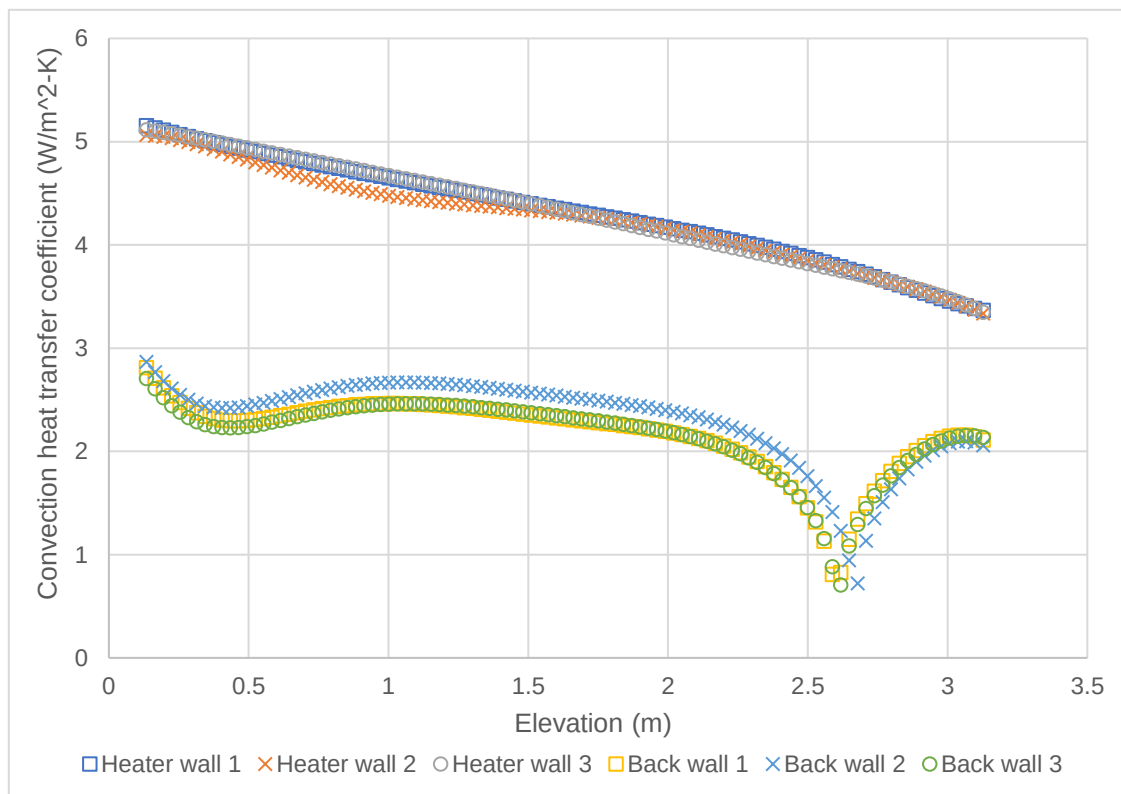


Figure 8.2-51. Convection heat transfer coefficients for the heater- and back wall.

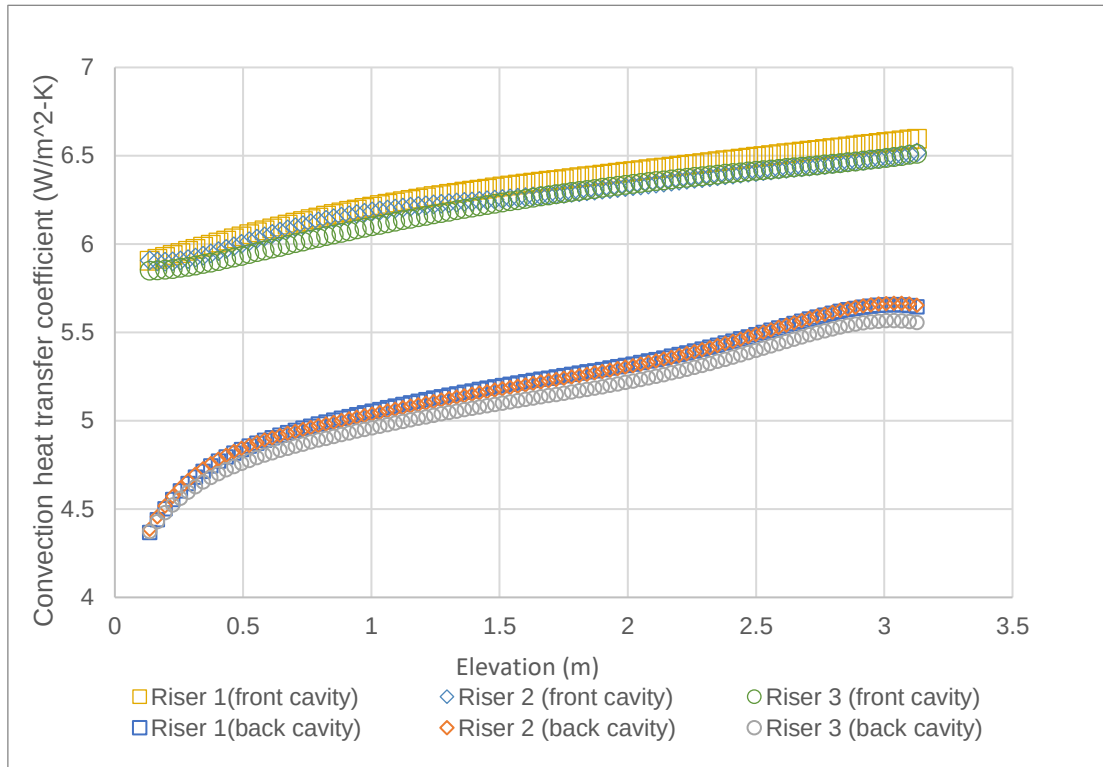


Figure 8.2-52. Convection heat transfer coefficient on the outer surfaces of the riser tubes.

Figure 8.2-53 shows the convection heat transfer coefficients on the outer surfaces of the fins. The coefficients in the front cavity showed an overall increase over the elevation, while the back-cavity side coefficients show an overall decrease. This was due to the difference in fluid and surface temperature increasing in the front cavity and decreasing in the back cavity with an increase in elevation. The heat was transferred upward and out of the heated section within the walls of the fins and tubes. The coefficients in the back cavity were more susceptible to this loss, since the difference between the fluid and surface temperatures were smaller than for the front sides. The smaller difference in fluid and surface temperature resulted in lower Grashof numbers, Rayleigh numbers, Nusselt numbers and subsequently convection heat transfer coefficients.

Similar to the corresponding circular tubes case, the outer fins (#1 and #4) show a larger coefficient than the inner fins (#2 and #3) for the front cavity and a smaller coefficient in the back cavity. The lower temperatures found by the line probes on the outer fins resulted in a larger difference in the fluid and wall temperatures in the front cavity. This resulted in a higher Grashof number, which subsequently resulted in a higher convection heat transfer coefficient. In the back cavity, conversely, the lower temperatures that were sampled from the fins resulted

in a smaller difference between the fluid and wall temperatures, resulting in the lower coefficients for the side fins.

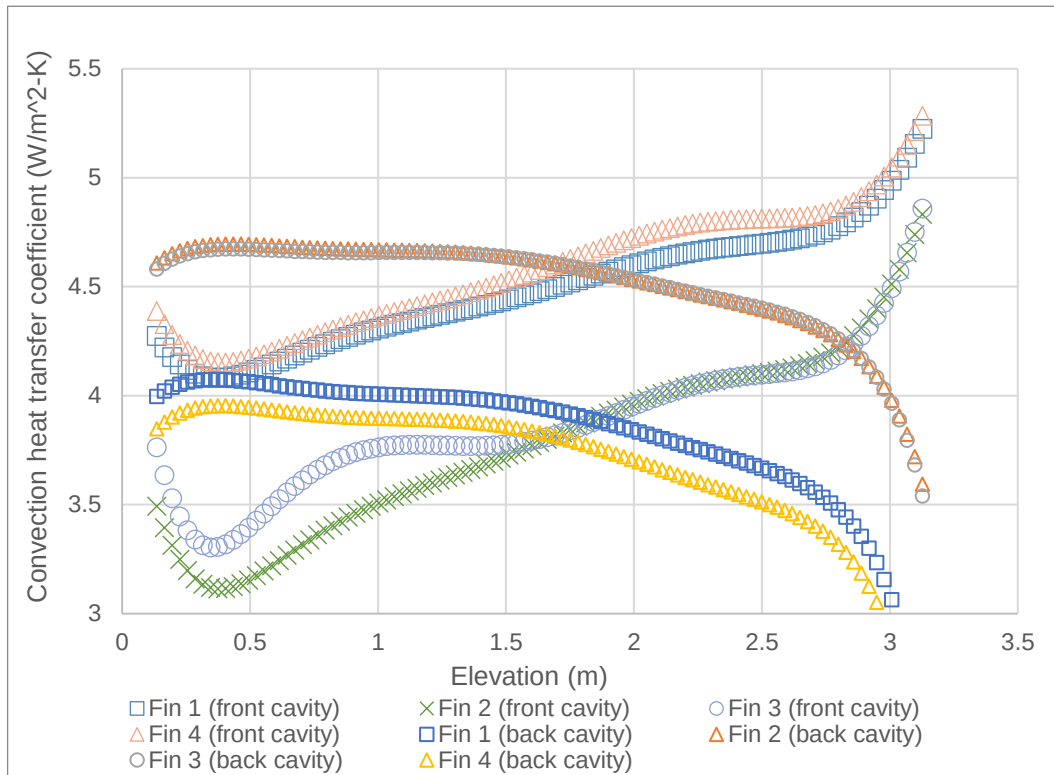


Figure 8.2-53. Convection heat transfer coefficients on the outer surfaces on the fins.

Similarly, as mentioned in the case with the round tubes, the air flows downwards against the fins in the front cavity (while cooling down) and upwards against the fins in the back cavity (while heating up). This also contributed to the behaviour of the coefficients shown in Figure 8.2-53.

The parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix D.4.

8.2.4.5 Conclusion

The geometry of the UW RCCS facility was modified to contain square tubes instead of circular tubes. The cross-sectional area of the tubes was kept the same as for the circular tubes case and the same boundary conditions were used.

The temperature distributions and velocity fields were investigated. The convection heat transfer coefficients were calculated for the important surfaces in the cavity and for the inner surfaces of the riser tubes.

The velocity vectors inside the middle riser were shown and provided insight into the prevailing flow regime that was present in the riser tube.

The heat flux distributions on the outer surfaces of the riser and fin surfaces were also investigated. The square tube geometry increased the surface area of the tubes that was exposed to the cavity. The integral of the heat flux on the riser tubes' outer surfaces were higher as a result. The square geometry also meant that the sides of the tubes received less radiation heat transfer, resulting in lower average heat fluxes transferred to the tubes' outer surfaces.

The Aicher numbers calculated for the inside of the tubes indicated the flow to be mixed convection. The left and right inner surfaces however, transitioned to the free convection regime. The behaviour was therefore similar to the corresponding circular tubes case in terms of the flow regime inside the riser tubes.

8.2.5 Square tubes without fins

Figure 8.2-54 shows the CAD model of the geometry of the square tubes without fins that was used in the CFD modelling. The figure also shows the naming convention as used when referring to the results.

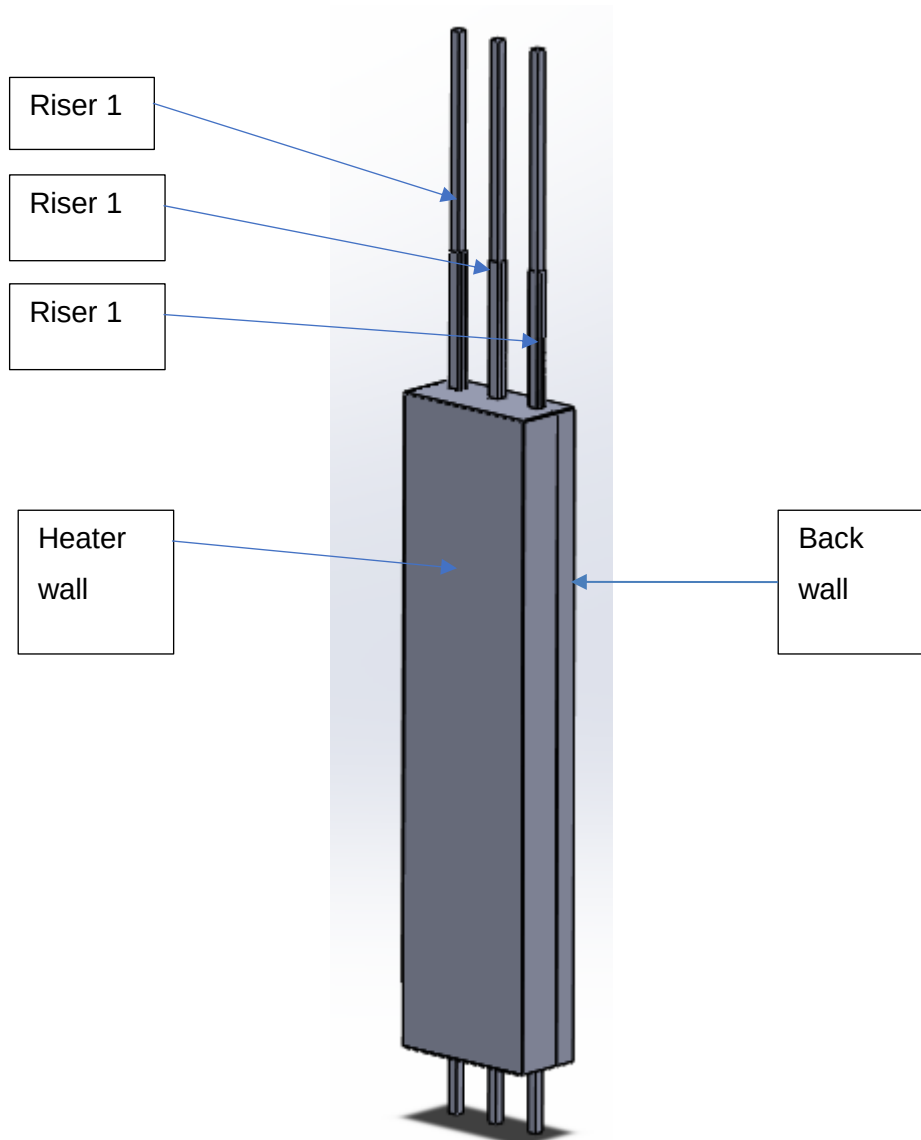


Figure 8.2-54.
Geometry used in the
CFD model.

8.2.5.1 Heat fluxes

Table 8-4 shows the surface average of the radiative - and total heat flux measured on the front - and back cavity sides of the riser tubes. The integral of the heat flux was also calculated to obtain the heat energy added to the riser tube surfaces. The percentage of the radiation heat transfer of the total heat transfer was also calculated. The convection heat transfer can then be inferred by the calculating the remaining percentage. The radiative heat transfer was approximately 16 % higher than in the corresponding circular tubes case (for reference see Table 8-2). The square geometry resulted in a larger surface area that was exposed to direct thermal radiative heat transfer. The disparity between the total heat flux on the middle riser and the riser tubes on the sides was 1.4 % (compared to 1.6-1.9 % for the circular tubes case). The surface area of the riser tubes was 15.8 % larger than the case with the circular tubes. The surface average of the heat flux on the outer surfaces of the riser tubes was smaller than the fluxes of the corresponding circular tubes case (see Table 8-2). The integral of the heat flux on the outer surfaces of the riser tubes was comparable to the circular tubes case. Note that for example the front side includes the front surface and $\frac{1}{2}$ of the left and right-side walls on the front side of the cavity.

Table 8-4. Selected results for the riser tubes.

	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Derived radiation/total heat transfer %	
	Front side	Back side	Front side	Back side	Front side	Back side	Front side	Back side
Riser 1	5837.94	5025.61	4440.17	3630.27	1897.44	1633.41	76.06	72.24
Riser 2	5920.14	5103.33	4539.47	3723.15	1924.15	1658.67	76.68	72.96
Riser 3	5838.41	5026.60	4439.71	3630.61	1897.59	1633.74	76.04	72.23

8.2.5.2 Temperature distributions

Figure 8.2-55 shows the water temperature at the middle of the riser tubes. The distribution was similar to the corresponding circular tubes case (see Figure 8.2-23), with a linear increase in temperature within the heated section. The temperature increase above the heated section

was due to the heat distribution equalising, as heat was transferred from the riser tube walls and boundary layer to the middle of the tube (where the probe was located).

Figure 8.2-56 shows the temperature distribution within the wall of the middle riser tube, which was also representative of the other riser tubes. The temperatures on the front wall were the highest, with the corresponding temperatures on the other three surfaces being almost the same.

Figure 8.2-57 shows the temperature contour on the outer surfaces of the riser tubes. The maximum temperatures were obtained in the corners of the tubes which was approximately 10 K warmer than the middle of the surfaces at the elevation shown. The possible reason is the boundary layers of the two sides overlapping at the corners inside of the riser tube. The lower velocity in the boundary layer caused the heat to build up in the corners. Another possibility is that the corners were heated by the hot air flowing around these corners while moving from the front to the back cavity (see Figure 8.2-61). Note that the corresponding corners in the corresponding finned case also contained a hotspot (see Figure 8.2-42).

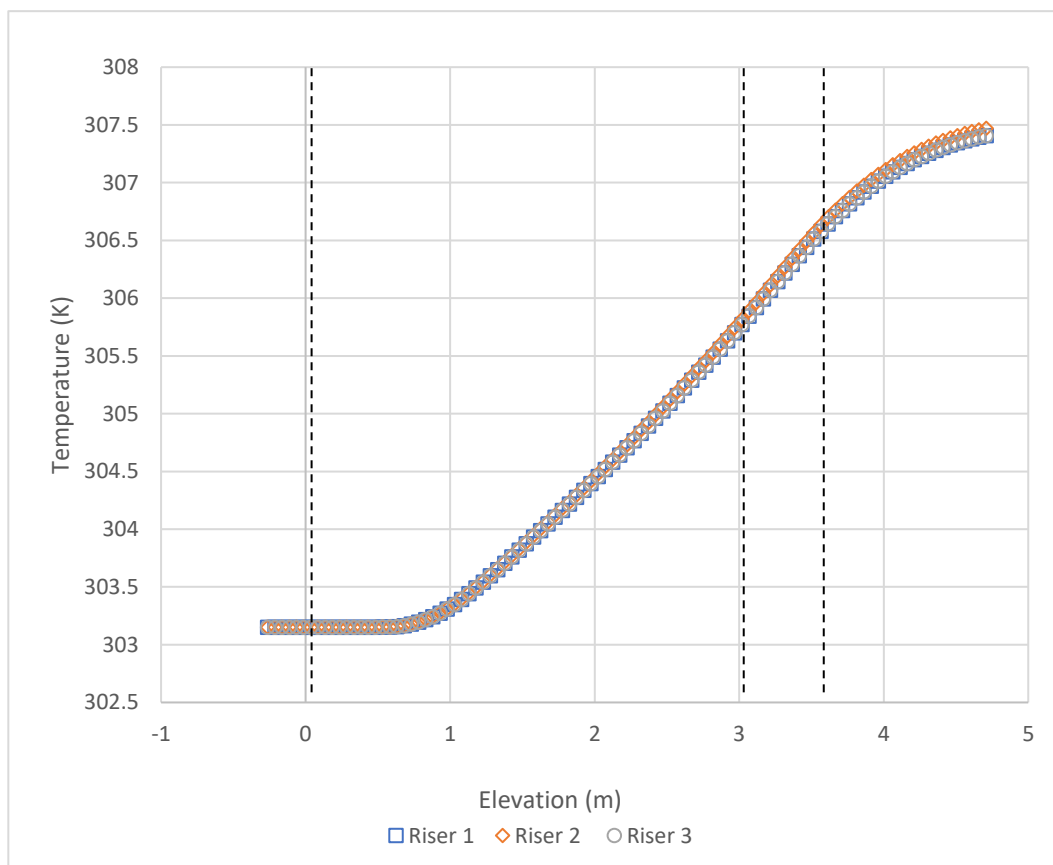


Figure 8.2-55. Water temperature distributions in the middle of the risers.

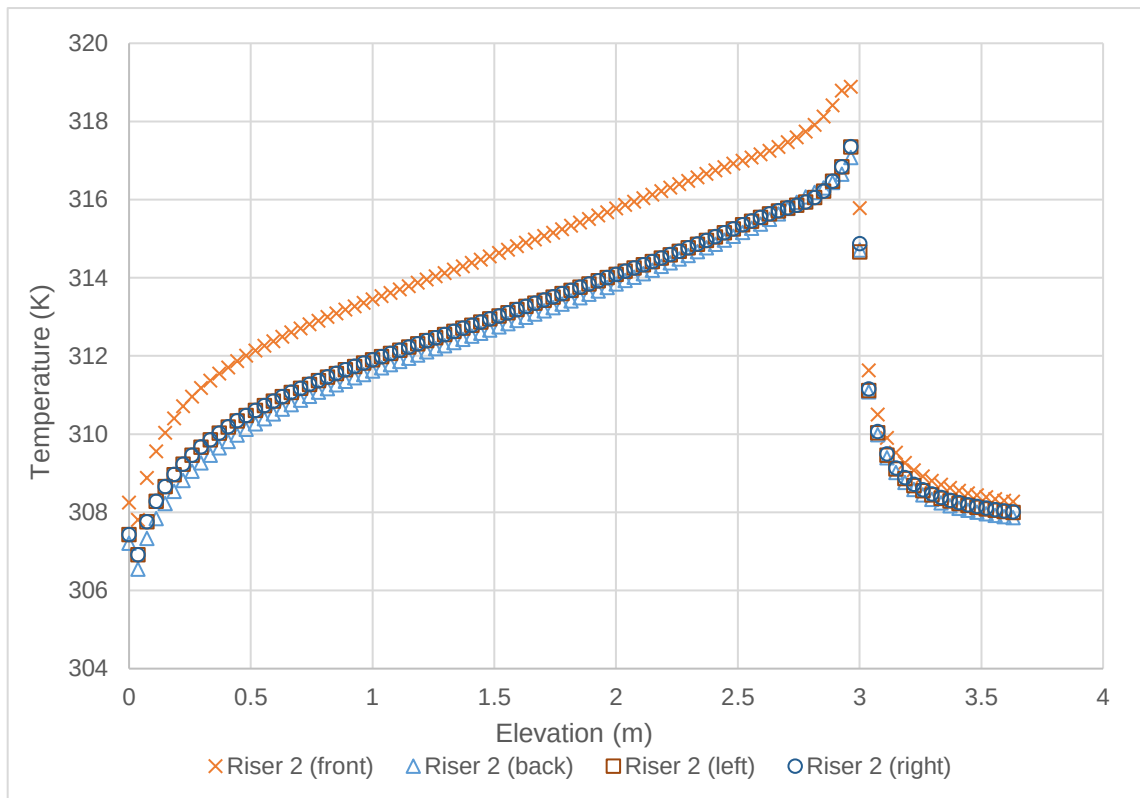


Figure 8.2-56. Riser tube wall temperatures for the middle riser tube.

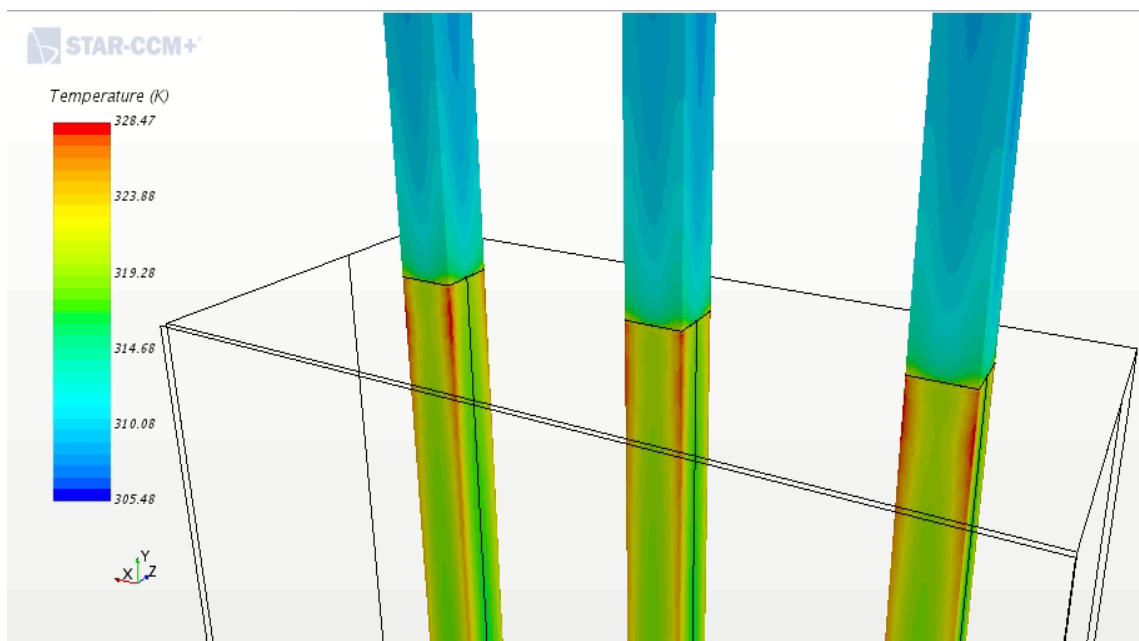


Figure 8.2-57. Temperature contour on the outer surfaces of the riser tubes.

Figure 8.2-58 shows the air temperatures as a function of elevation. The temperatures were taken on three line probes at a location that is half way between the centrelines of the tubes and the heater or back wall for the front cavity air and back cavity air respectively. The line probes were also placed across from the riser tubes, in line with each riser tube's centreline. The front and back sides of the cavity had a similar temperature distribution, since the air temperature could equalize by moving between the front and back parts of the cavity. At the top elevation in the front cavity a hotspot was observed, because of the hot air being slowed down by the cavity's top wall. In contrast, on the back-cavity side, the fluid flow paths transported the heat downwards and away from the probe at the top elevation. At the bottom elevation the temperature in the back cavity the air was unheated, since the heated fluid moved upwards and away from this position due to buoyancy forces.

Figure 8.2-59 shows the temperatures on the heater- and back wall. The temperatures of the back wall were significantly higher than the corresponding square tubes with fins case. This was because of the direct radiation heat transfer from the heater wall that was possible with the omission of the fins from the design. On the back wall the very bottom and top elevations show a deviation from the slope that was maintained throughout the middle elevations. The distributions infer a stronger cooling effect at the top and bottom elevations since the flow patterns remove heat more effectively from these locations. The buoyancy effects at the bottom elevation removes heat upwards resulting in the cold spot at the bottom. At the top elevation the recirculation cell (see Figure 8.2-61) prevented the flow of the heated air from the front cavity to reach the wall at the top elevation.

Another observation that can be made in Figure 8.2-59 is that the temperature profiles on the heater wall are the same, while the temperatures at the second probe position are slightly higher (approx. 1.3%) than the temperatures at the other probe positions. Since the heater wall was the position at which the heat flux boundary condition was placed, dispersion could not occur as much as on the back wall. The probe line on the back wall at which the higher temperatures were found is located in the middle of the heater box. The temperatures in the middle were generally higher (temperatures on the surfaces of Riser 2 for example) causing the higher temperatures to be found in the middle of the back wall as well.

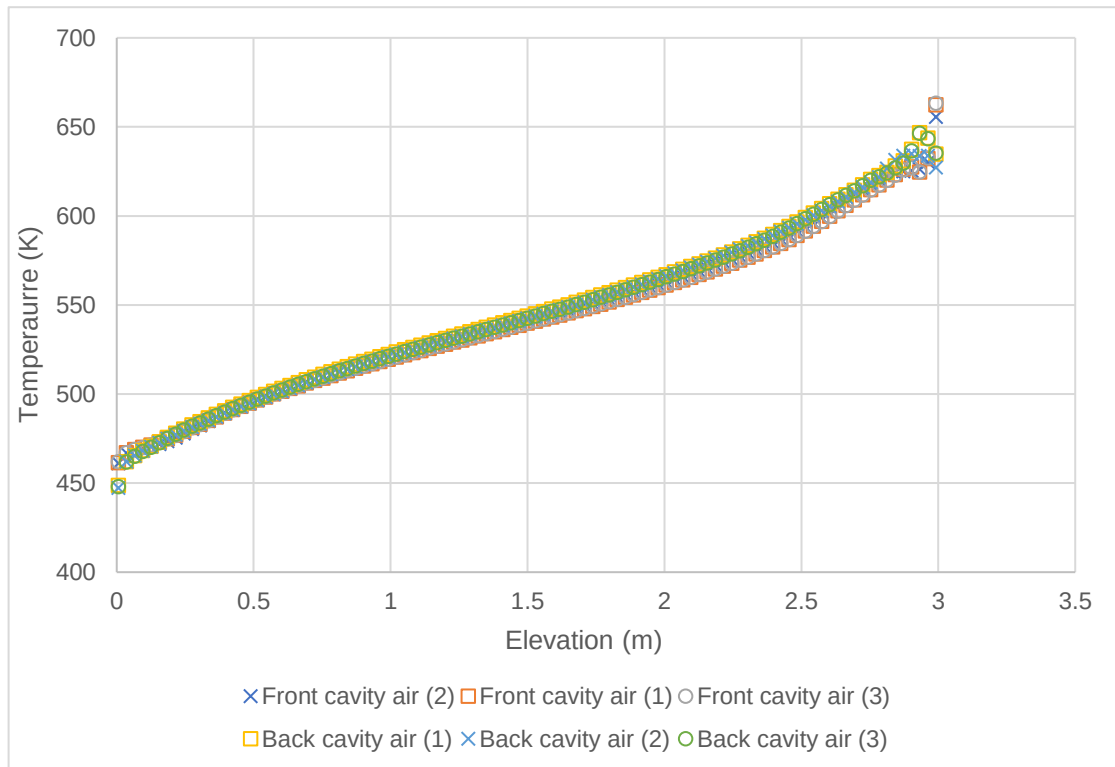


Figure 8.2-58. Air temperatures in the front and back side of the cavity plotted as a function of elevation.

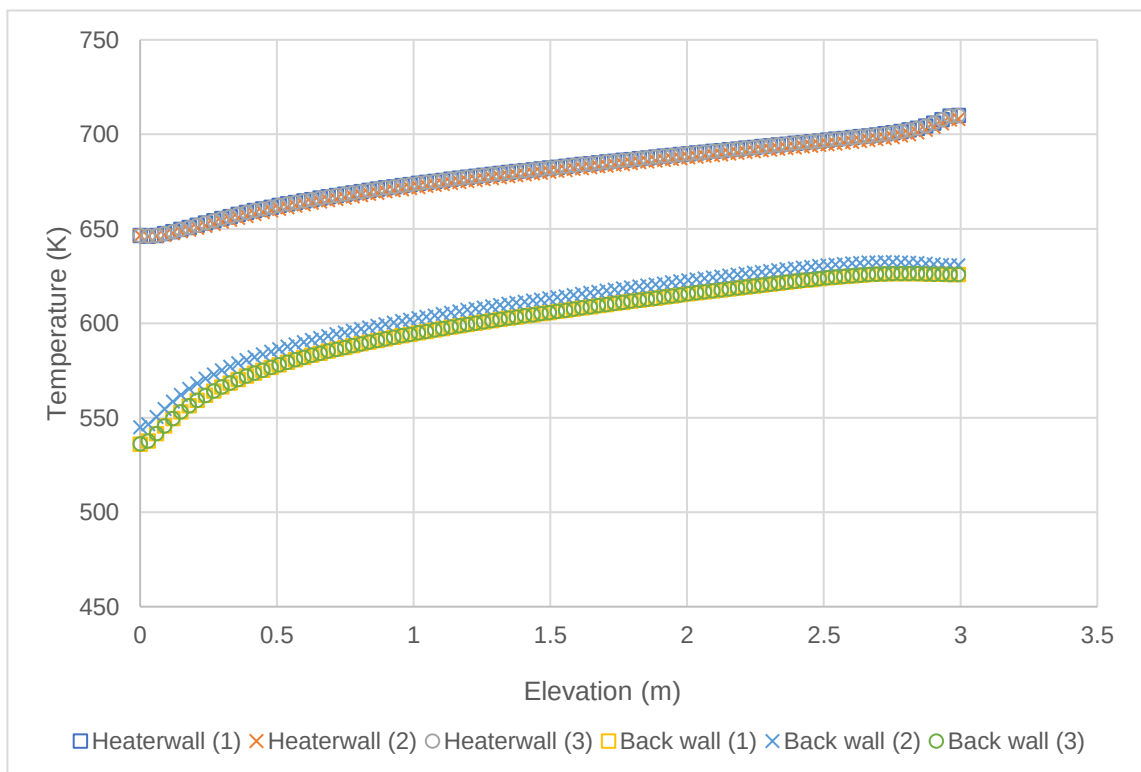


Figure 8.2-59. Heater- and back wall temperatures as a function of elevation.

8.2.5.3 Velocity vectors

Figure 8.2-60 shows the velocity vectors in the front and back part of the cavity. The vectors are shown on a section plane that intersects the middle of the heated section. The figure shows the recirculating areas of the air forming the flow pattern similar to the circular tubes case. In the back cavity the heated air travelling from the front cavity reflects off of the back wall and is forced downwards by the cooling effects of the riser tubes.

Figure 8.2-61 shows the velocity vectors in the front and back part of the cavity. The vectors were taken on a plane that intersects the heated section between the middle and third riser tube. The vectors show the recirculation cell that forms in the top elevations of the back cavity. The flow pattern at this location prevents the heated air from reaching the topmost elevation of the back wall. At the elevations slightly lower than shown in the figure, the strong downward flow pattern between the riser tubes becomes more dominant. This forces the air downwards next to the riser tubes, while upward flow dominates on the heater – and back walls.

Against the back wall at the higher elevations the flow direction changes from upward to downward flow, creating a point at which the velocity in the y-direction is very small. This point is indicated in Figure 8.2-60 as the dot and for the purposes of the discussion is referred to as a stagnation point.

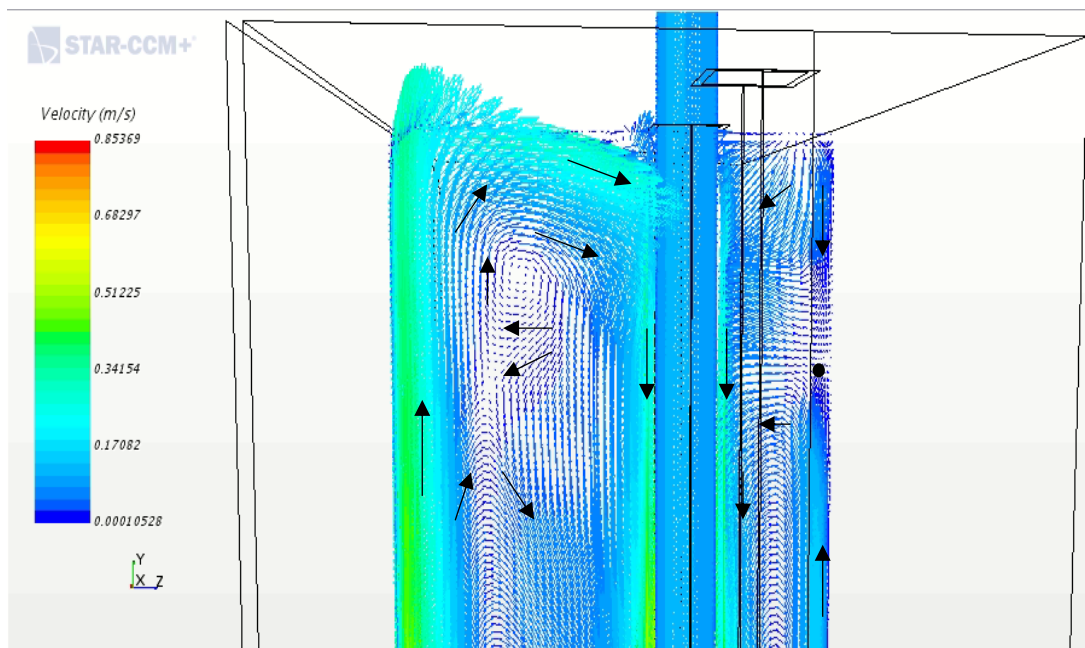


Figure 8.2-60. Velocity vectors in the front – and back cavity.

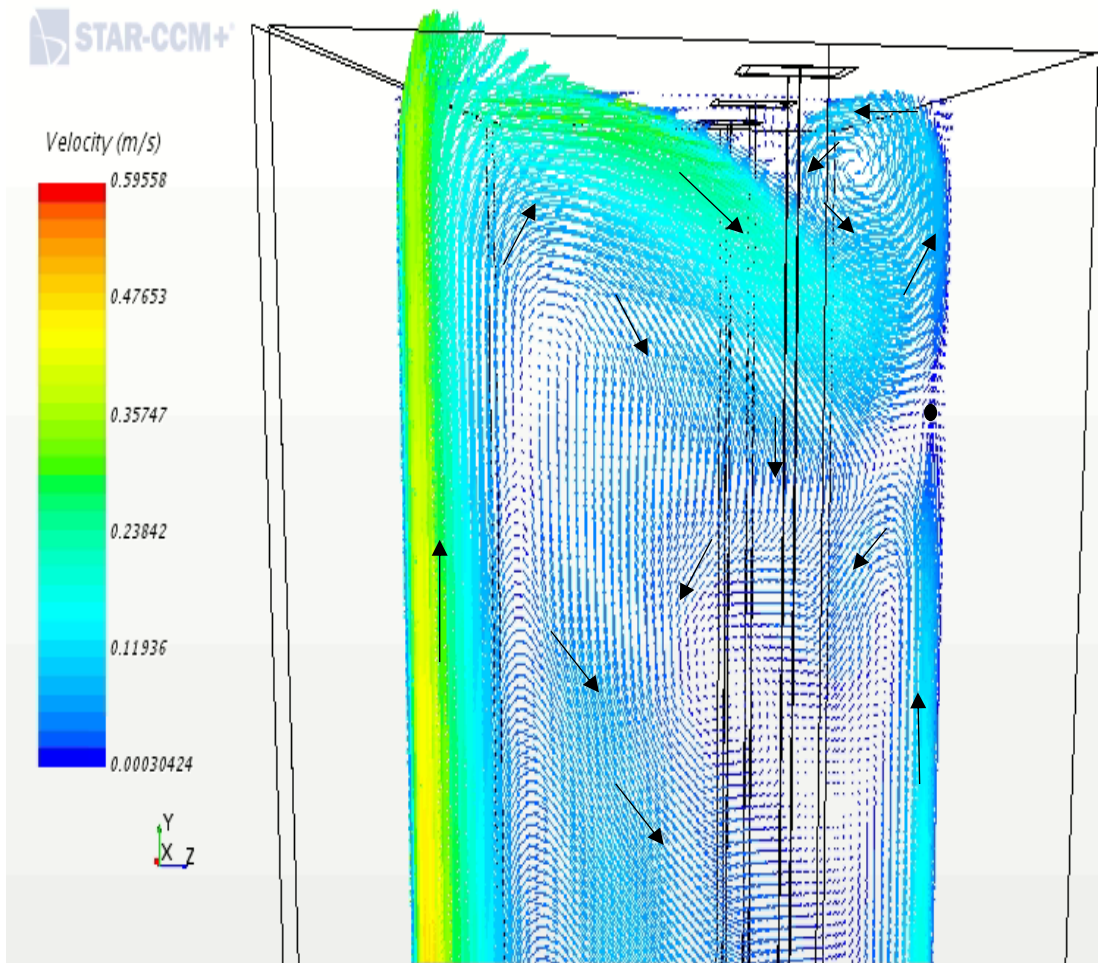


Figure 8.2-61. Velocity vectors in the front - and back parts of the cavity (continued).

Figure 8.2-62 shows the velocity vectors in the middle riser tube at four different elevations as viewed on the x-y plane. The velocities at the lowest elevation showed a profile representing laminar flow. The second elevation profile broadened out more across the cross-sectional area. The third and fourth elevations velocity peaks formed close to the wall, owing to the buoyancy induced flow near the walls. The peaks seem to be near the front wall and thus associated with the hot corners mentioned in the discussion of Figure 8.2-57.

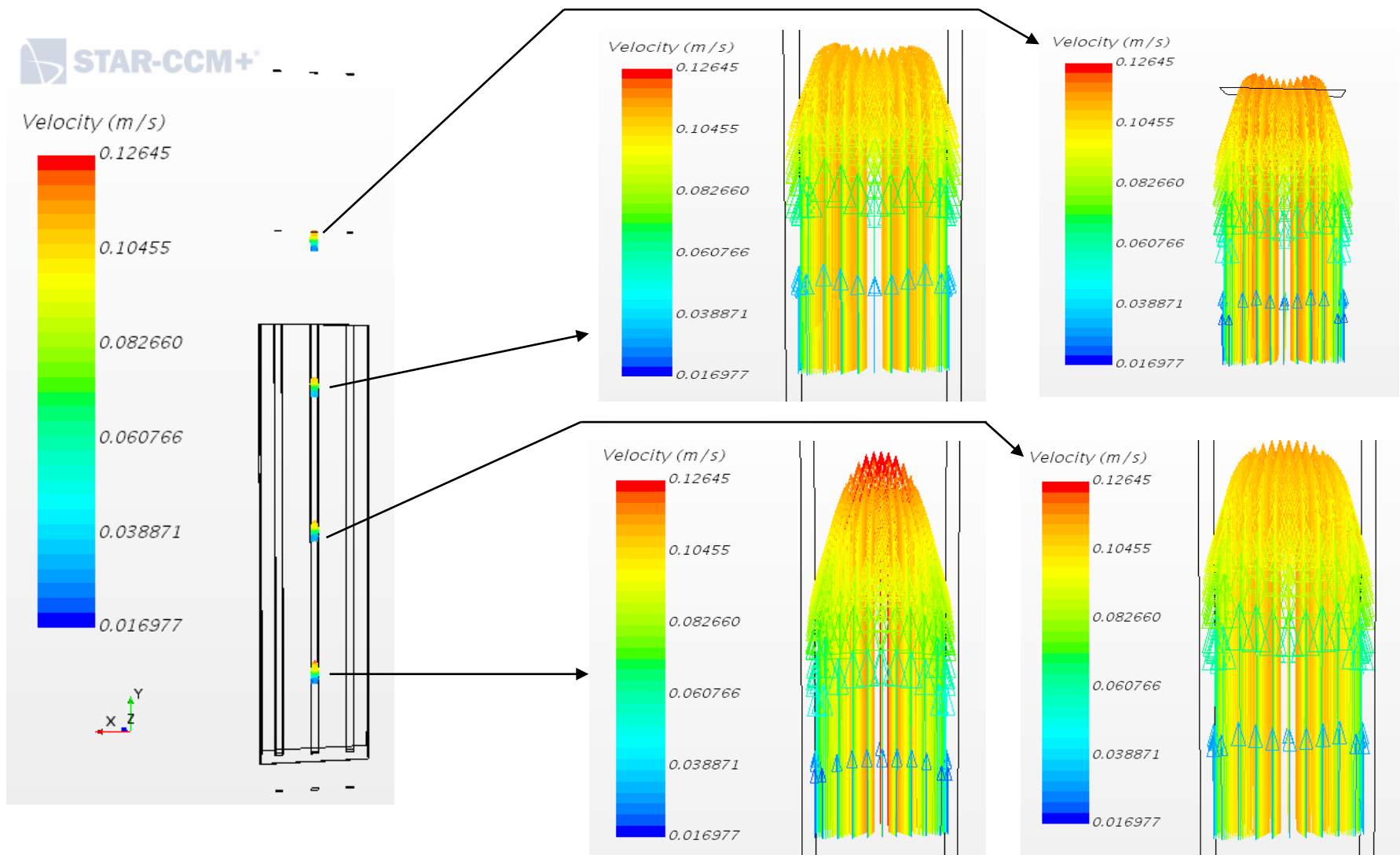


Figure 8.2-62. The velocity vectors in the middle riser tube.

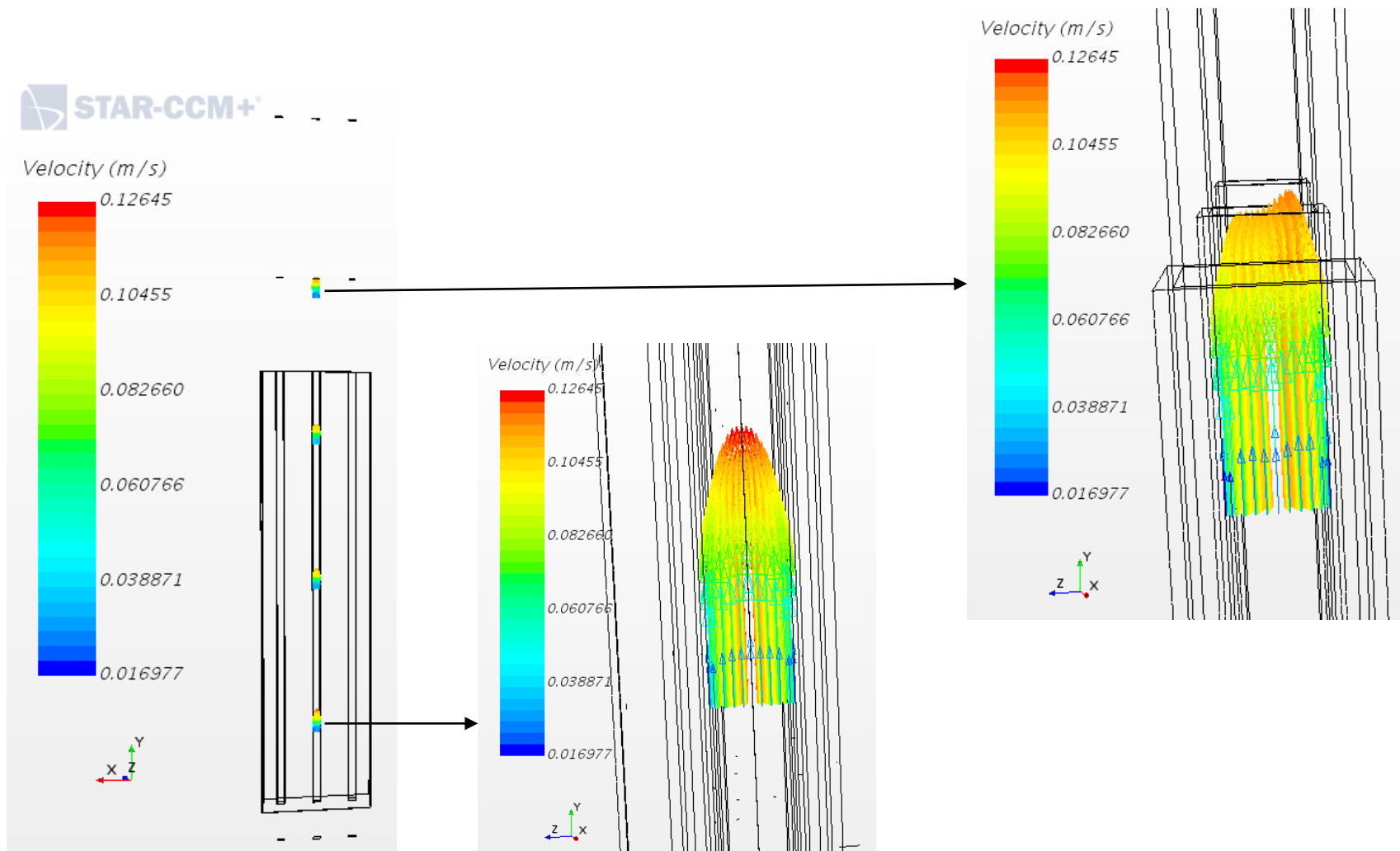


Figure 8.2-63. Velocity vectors in the middle riser tube (continued).

Figure 8.2-63 shows the velocity vectors inside the middle riser tube as viewed from the x-direction (i.e. on the y-z plane). The velocity profile at the lowest elevation was symmetric, as it was influenced primarily by the inlet boundary condition. The velocity profile at the highest elevation was asymmetric as it was strongly influenced by the asymmetric heating.

8.2.5.4 Heat transfer coefficients

Figure 8.2-64 shows the convection heat transfer coefficients calculated for the inner surfaces of the middle riser tube. The coefficients show an overall increase with increasing elevation. The coefficients for this case for the front surface were higher than the corresponding circular tubes case. The other surfaces however had lower coefficients than the corresponding circular tubes case. The coefficients associated with the front side were higher than the corresponding coefficients for the other sides because the temperature difference between the fluid and wall was higher on the front surface. The higher temperature difference resulted in a higher Grashof number, resulting in turn in a higher Rayleigh and Nusselt number, resulting in a higher convection heat transfer coefficient.

At the first datapoint however, the coefficients on the other surfaces are higher than the coefficient on the front surface. This was because of the stronger free convection Nusselt number coefficient on the front side, caused by more heating that occurs on the front side. The mixed convection Nusselt number is therefore lower for the front side at the lowest elevation, since the mixed convection Nusselt number is calculated by subtracting the free convection from the forced convection Nusselt number (see Equation (16) in section 3.3). The lower mixed convection Nusselt number in turn leads to a lower convection heat transfer coefficient for the lowest elevation of the front side. After the other three sides are heated (by the second datapoint) the free convection Nusselt number becomes larger, resulting in the decrease seen on these sides from the first to the second datapoint. Note that the Reynolds number was in excess of 6 000.

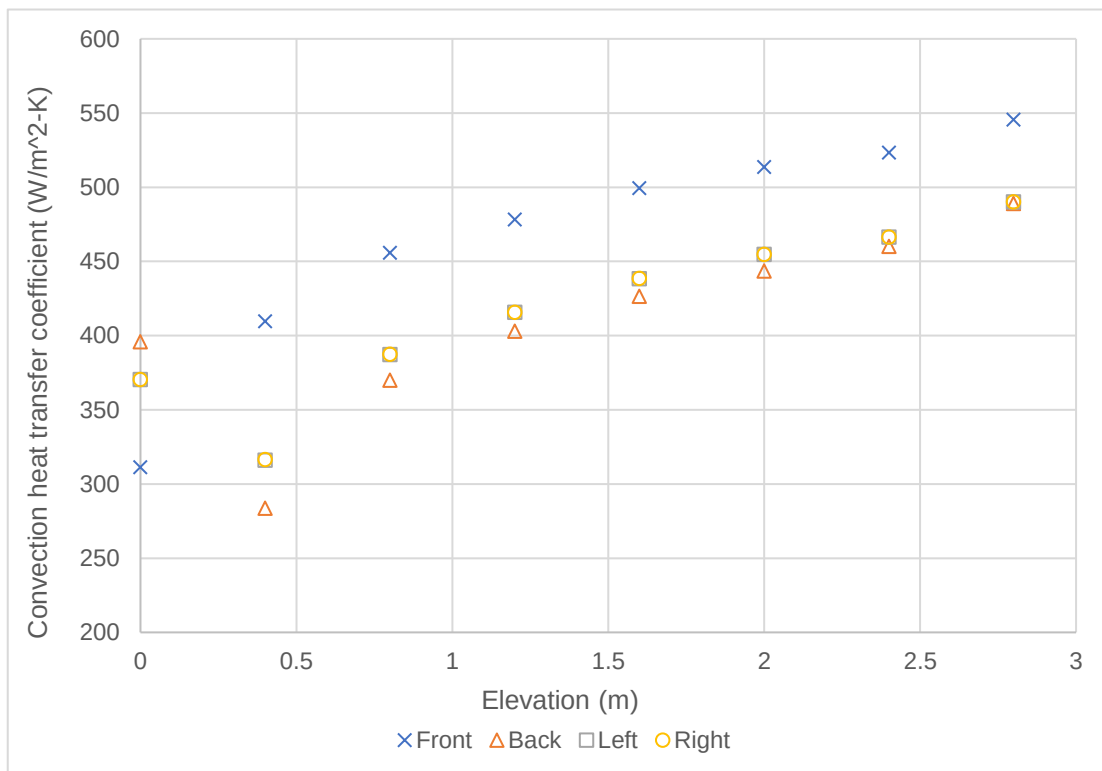


Figure 8.2-64. Convection heat transfer coefficients for the inner surfaces of the middle riser tube.

Figure 8.2-65 shows the Aicher number plotted as a function of elevation for the water side of the middle riser tube. The Aicher number was well within the mixed convection criterion for the entire elevation on all surfaces within the riser tube ($0.05 < \text{Aicher} < 0.2$). There was however an overall increase along the elevation which moved the regime closer to free convection.

Figure 8.2-66 shows the Nusselt number ratio plotted as a function of the Aicher number. The first datapoint for all surfaces showed deteriorated Nusselt number, while the subsequent points were on the curve of recovery of the Nusselt number. The ratio was below a ratio of 1, which indicates convection that is not as effective as pure forced convection. In the corresponding case with round tubes the Nusselt number ratio was also primarily below 1, with only the very last datapoints exceeding 1. From this perspective the circular tubes configuration is the better geometry as it limits the occurrence of the mixed convection regime and its detrimental effect on heat transfer.

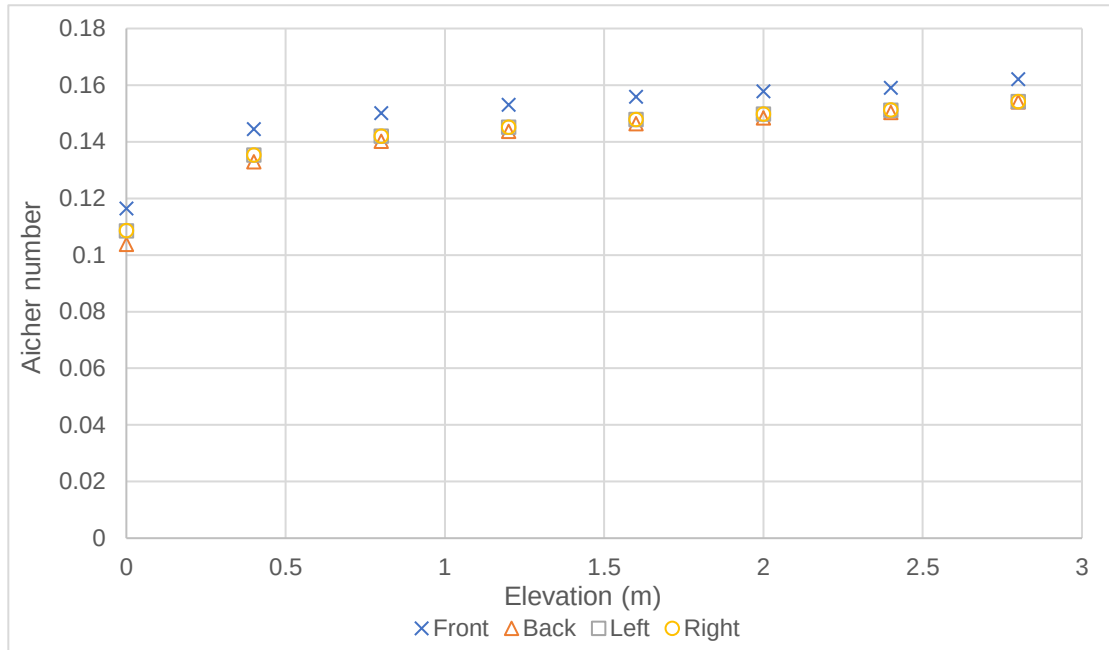


Figure 8.2-65. Aicher number as a function of elevation in the middle riser tube.

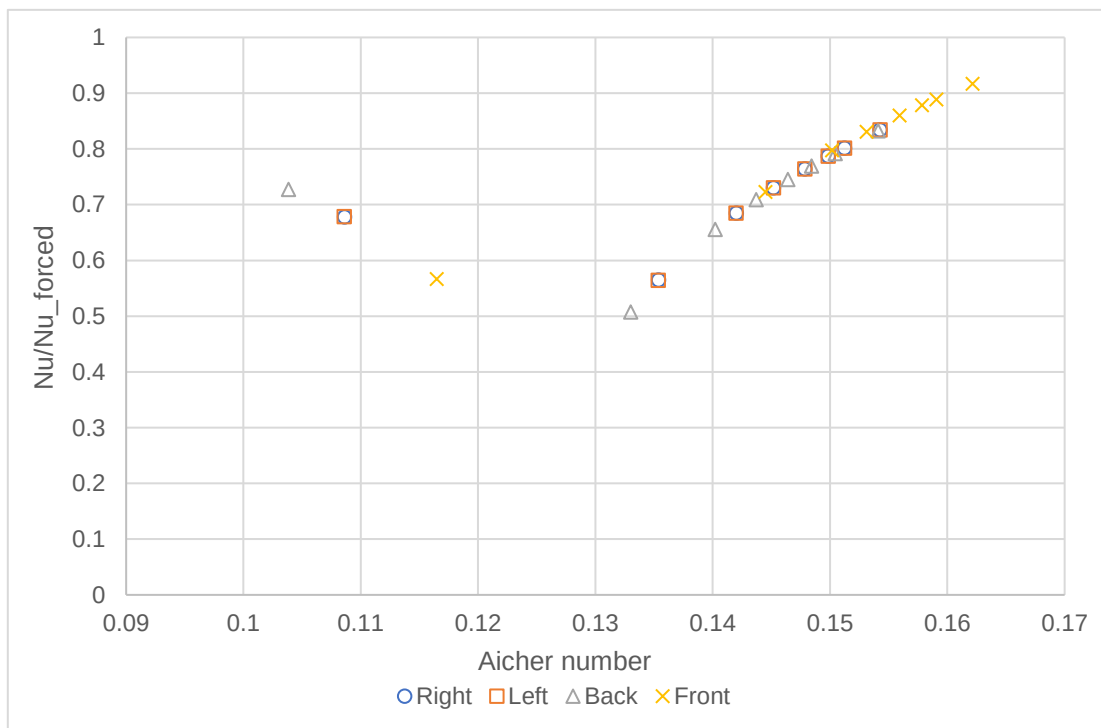


Figure 8.2-66. Nusselt number plotted as a function of Aicher number.

Figure 8.2-67 shows the convection heat transfer coefficients on the outer surfaces of the riser tubes for the front – and back parts of the cavity. The coefficients on both sides show an overall

increase across the elevation and are almost the same as the corresponding finned case (less than 1% difference along most of the elevation). The coefficients on the back-cavity side were slightly higher, the reason being the larger difference between the wall and fluid temperatures at these locations. At the bottom elevations however the coefficients on the back sides are lower, since a cold spot occurs at this location, seen with the back-cavity air temperatures and cavity back wall temperatures. The low temperature gradients at this location resulted in the lower convection heat transfer coefficients against the back walls of the riser tubes.

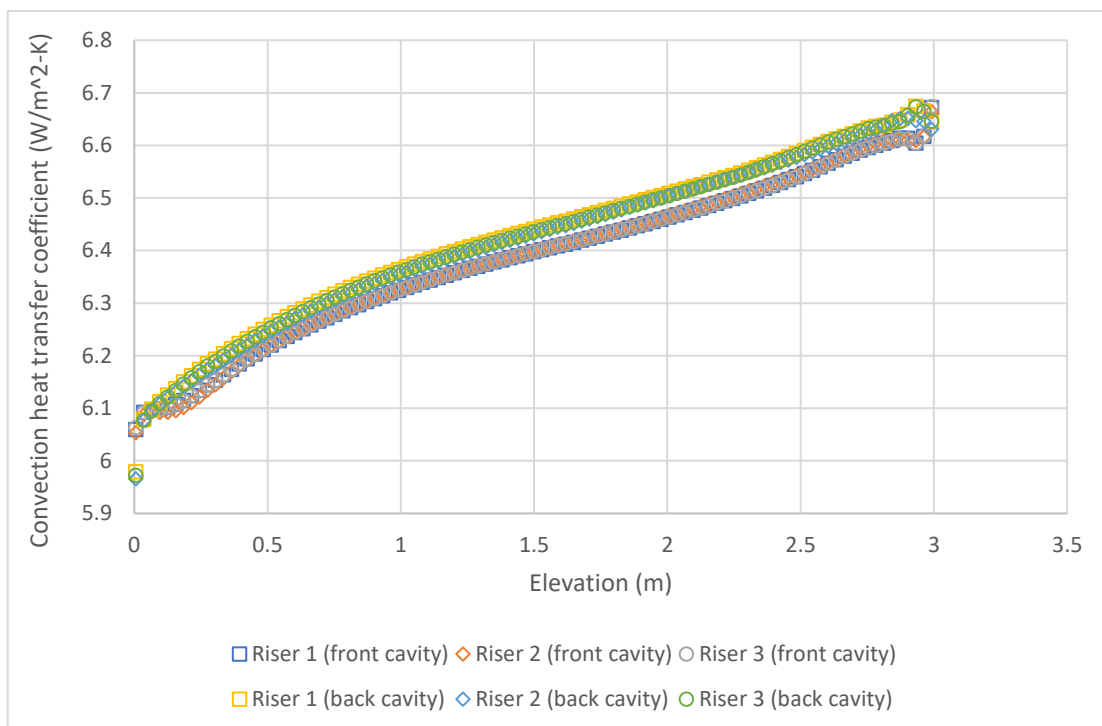


Figure 8.2-67. Convection heat transfer coefficients on the front – and back cavity sides of the riser tubes.

Figure 8.2-68 shows the calculated convection heat transfer coefficients for the heater- and back walls. The coefficients showed an overall decrease with increasing elevation, as the air temperatures at the higher elevations increased to a value closer to the wall temperatures. The values of the surface and fluid temperature were almost the same at the back wall between 2.8 and 2.9 m, which caused a lower coefficient to be calculated at that location.

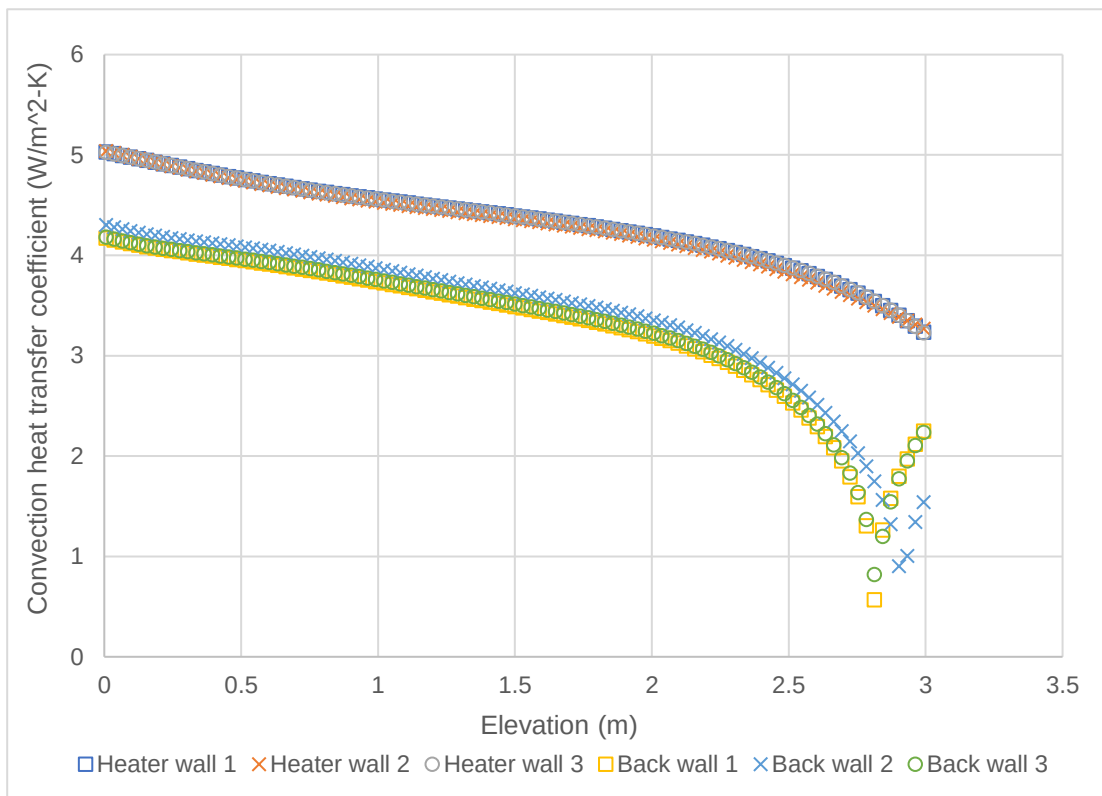


Figure 8.2-68. Convection heat transfer coefficients as a function of elevation.

The change between upward and downward flow at the higher elevations of the heated section caused a 'stagnation point' (indicated in Figure 8.2-60) which was detrimental to the convection heat transfer. This also explains the decrease in the convection heat transfer coefficient between 2.5 and 3 m in Figure 8.2-68.

The parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix D.5

8.2.5.5 Conclusion

The geometry of the UW RCCS facility was modified to have square riser tubes and the fins were omitted from the design. The cross-sectional area was kept the same as the circular tubes design. The same boundary conditions were also used.

The temperature profiles and velocity vectors were investigated, and the convection heat transfer coefficients were calculated based on the extracted results.

The velocity vectors on the inside of the middle riser were shown and provided insight into the prevailing flow regime that existed in the riser tubes.

The distribution of the heat fluxes on the riser tubes' outer surfaces was also discussed. The square geometry increases the area exposed to the heat transfer in the cavity. Another observation caused by the square geometry was the hotspots that were observed in the corners (of the front cavity side) of the tubes, which was especially apparent at the higher elevations.

The Aicher numbers that were calculated indicated the prevalence of the mixed convection in the riser tubes along the elevation of the heater box. The convection heat transfer coefficients are 20% lower (at the highest elevation) than the corresponding circular tubes case. The square tubes are therefore not as good as the circular tubes from a heat transfer perspective, since the mixed convection regime is more prevalent in the square tubes.

The temperatures in the front and back of the cavity were closer related in terms of value, since the air in the cavity could move freely between the front and back side of the cavity. This distributed the heat more evenly than in the corresponding finned case.

8.2.6 Rectangular tubes with fins

The geometry of the rectangular tubes with fins that was used to simulate the case is shown in Figure 8.2-69. Not only was the geometry changed from circular (from the baseline case) to rectangular but the number of tubes was also doubled for this case. This tested the dependency of the geometry but also the dependency on the number of riser tubes. The total cross-sectional area of the riser tubes was kept the same as the other cases. The naming convention remained the same from the previous cases with the Risers 1 through 6 and Fins 1 through 7 from left to right in the figure.

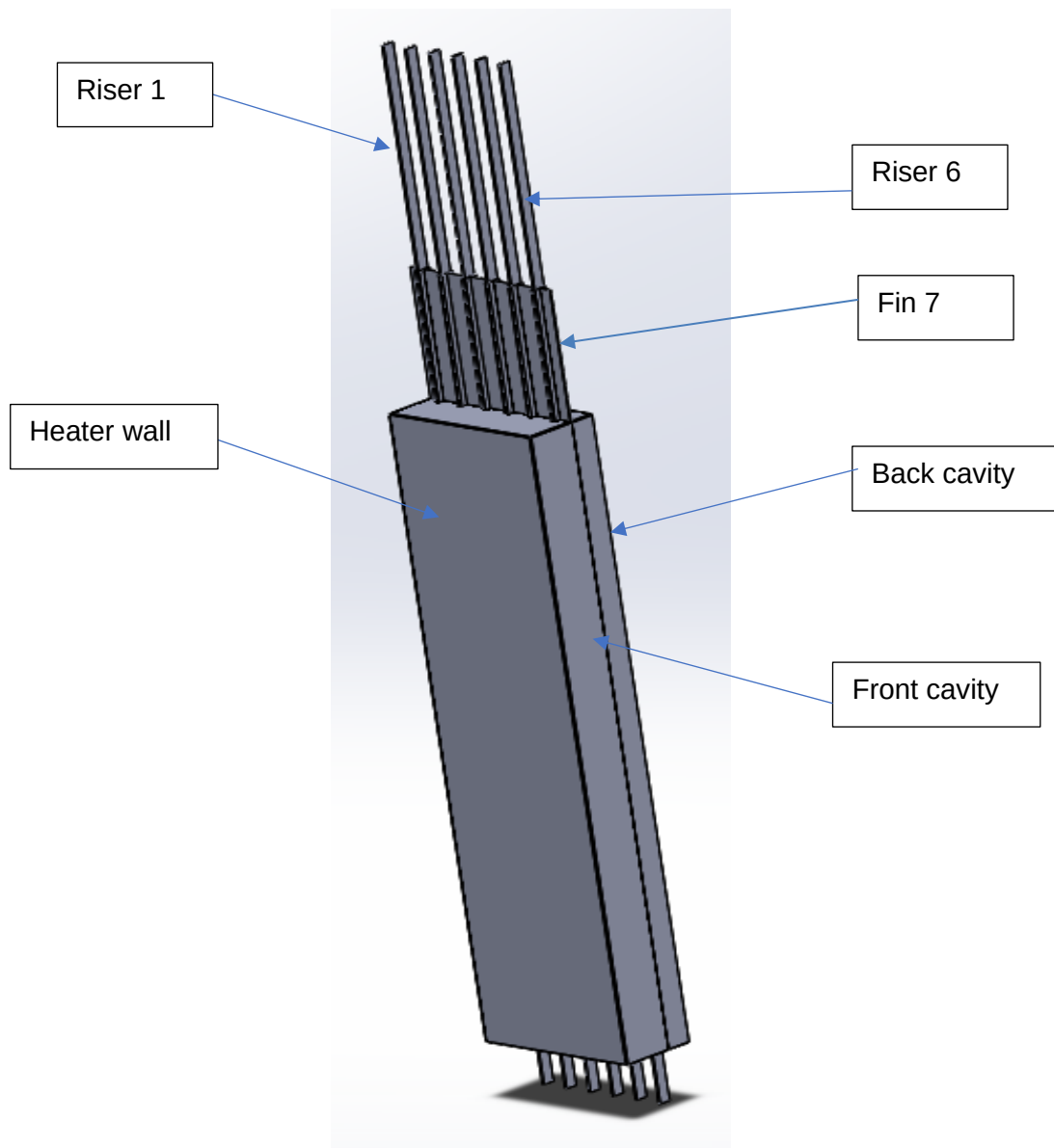


Figure 8.2-69. Geometry used in the CFD model.

8.2.6.1 Heat fluxes

Table 8-5 shows selected results for the case with rectangular tubes with fins. The surface average of the heat flux on the front and back cavity sides are shown in Table 8-5, as well as, the surface average of the radiation heat flux on the front and back cavity sides. The integral of the heat flux was also calculated to obtain the total heat energy received from the front and back cavity sides. Lastly, the ratio of radiation to total heat flux was calculated as a percentage for the front and back cavity sides. The remaining percentage consisted of heat energy that was transferred by the mechanism of convection. The middle risers #3 and #4 received the most heat followed by risers #2 and #5 (2% lower) followed by #1 and #6 (approximately 3% lower than #2 and #5). The surface of Fin 4 had the highest temperature. The temperatures therefore were at the highest in the middle of the heated section, similar to the previous cases. Note that the heat transfer on the front side for example, includes the front surface as well as $\frac{1}{2}$ of the length of the left and right sides.

Table 8-5. Selected heat fluxes for the case with rectangular tubes with fins.

	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Ratio of radiation/total heat flux %	
	Front side	Back side	Front side	Back side	Front side	Back side	Front side	Back side
Riser 1	2966.68	142.01	2112.89	70.53	869.79	41.63	71.22	49.67
Riser 2	3055.35	143.38	2209.62	70.80	895.79	42.04	72.32	49.38
Riser 3	3118.99	144.14	2275.30	71.56	914.44	42.26	72.95	49.65
Riser 4	3119.59	144.07	2274.84	71.53	914.62	42.24	72.92	49.65
Riser 5	3055.58	142.97	2209.68	70.49	895.85	41.92	72.32	49.30
Riser 6	2967.34	142.04	2113.10	70.55	869.98	41.64	71.21	49.67
Fin 1	2721.29	-121.01	2025.93	-59.16	393.78	-17.51	74.45	48.89
Fin 2	3054.12	-144.48	2442.98	-74.86	883.86	-41.81	79.99	51.81
Fin 3	3139.41	-153.16	2548.75	-78.99	908.58	-44.32	81.19	51.58
Fin 4	3154.18	-154.37	2565.70	-79.50	912.88	-44.68	81.34	51.50
Fin 5	3140.09	-152.34	2548.71	-78.55	908.77	-44.09	81.17	51.56
Fin 6	3053.60	-144.66	2442.94	-74.92	883.74	-41.87	80.00	51.79

Fin 7	2720.39	-120.56	2025.14	-58.92	393.65	-17.45	74.44	48.87

8.2.6.2 Temperature distributions

The temperature distributions in the water in the riser tubes, the riser tube walls, fin walls, heater wall, back wall and cavity air are shown. The water temperatures on the line probes in the middle of the tubes are shown in Figure 8.2-70. The temperatures are slow to increase at the bottom elevations, since the heat had to spread from the front of the riser tubes to where the probes were located. The temperature distributions are very similar between the tubes, except for the highest elevations. At these elevations, the two middle tubes had the highest temperature and the two outermost tubes had the lowest temperature. The reason for the differences in outlet temperature can be explained by using Figure 8.2-71, which shows the temperature scalar scene of Risers 1 through 3. The figure shows that the heat spread further into the tube (from the front cavity side) the closer the tube was to the middle of the heated section. The probes in the middle of the tubes therefore tracked a higher temperature for risers 3 and 4 than for risers 2 and 5 than for risers 1 and 6.

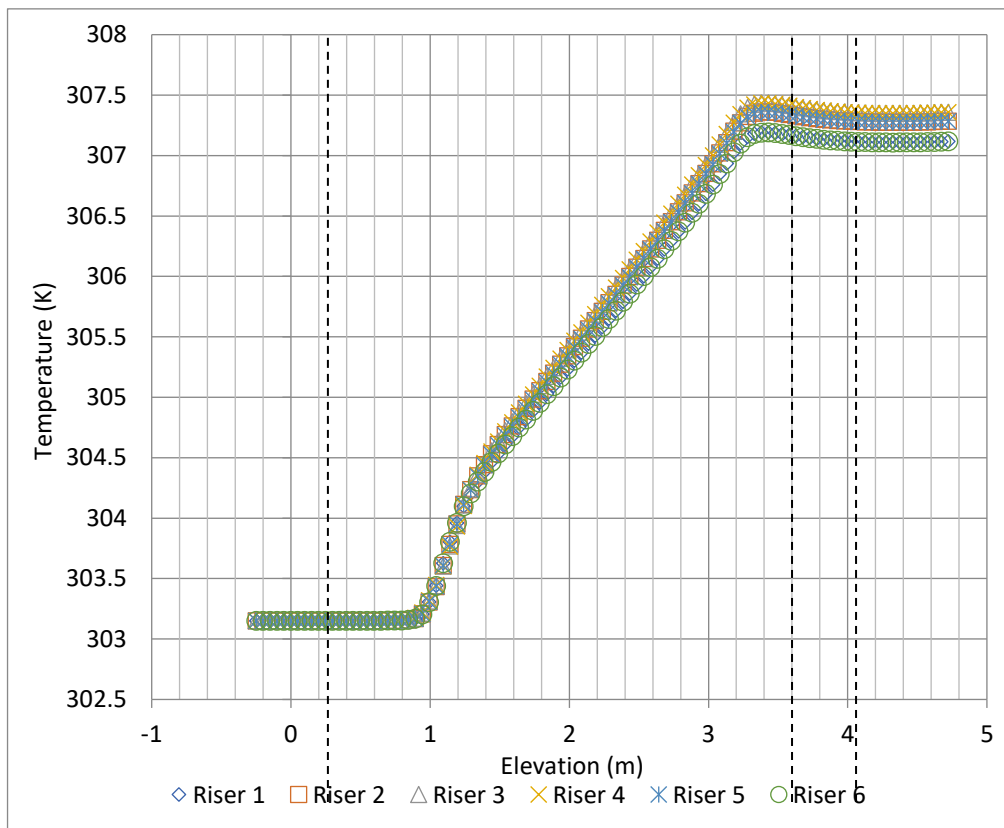


Figure 8.2-70. Water temperature distributions in the middle of the riser tubes.

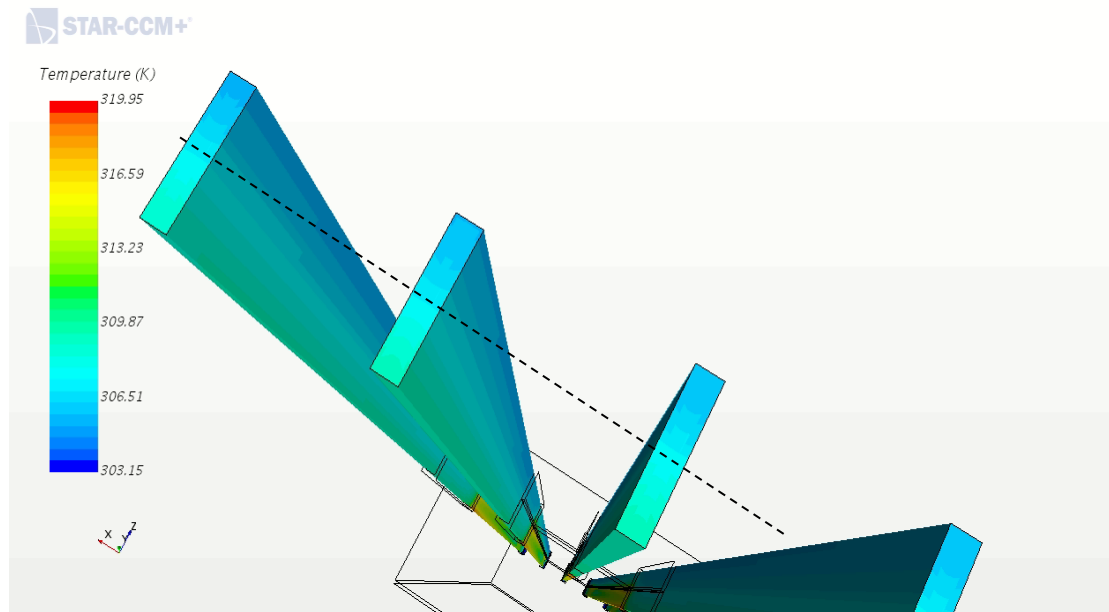


Figure 8.2-71. Temperature scalar scene of the first three risers.

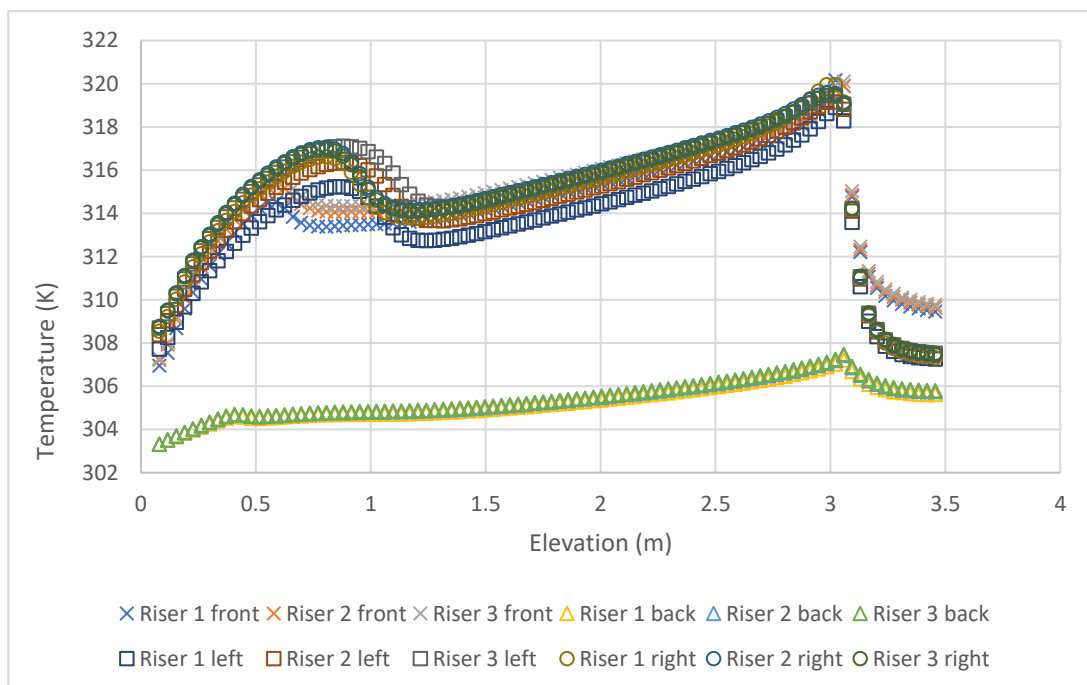


Figure 8.2-72. Riser tube wall temperature for the case with rectangular tubes with fins.

Figure 8.2-72 shows the temperature distribution in the walls of the riser tubes for riser tubes 1 through 3. Since the solution was symmetrical Risers 4 through 6 are omitted from Figure 8.2-72 to enhance clarity. The distribution that was found for Riser 3 would be representative for Riser 4, Riser 2 would be representative for Riser 5 and Riser 1 representative for Riser 6. The temperatures show an overall increase within the heated section and a swift decrease after leaving the heated section (at ≈ 3 m). The side surfaces had the highest temperatures, followed by the front sides and the back wall had the lowest temperatures. The rise in the temperature below 1 m was due to the heat being transported downwards from the higher elevations and then being slowed down by the denser cooler air at the lower elevations (as shown later in Figure 8.2-78) causing a build-up of heat in this location.

Figure 8.2-73 shows the temperature distributions in the fins for the case with rectangular tubes with fins. The temperatures of the side fins (1 and 7) were the lowest and the temperatures were the highest for the fin in the middle (Fin 4). As with the risers the temperature dropped dramatically above the heated section. A temperature rise can be seen on the surfaces below 1 m similar to the surfaces of the riser tubes. The heat transfer was however less pronounced since the flow velocity was higher, meaning a shorter exposure time between the warmer air and the fin surfaces. The local velocities in front on Fin 4 are shown in Figure 8.2-79.

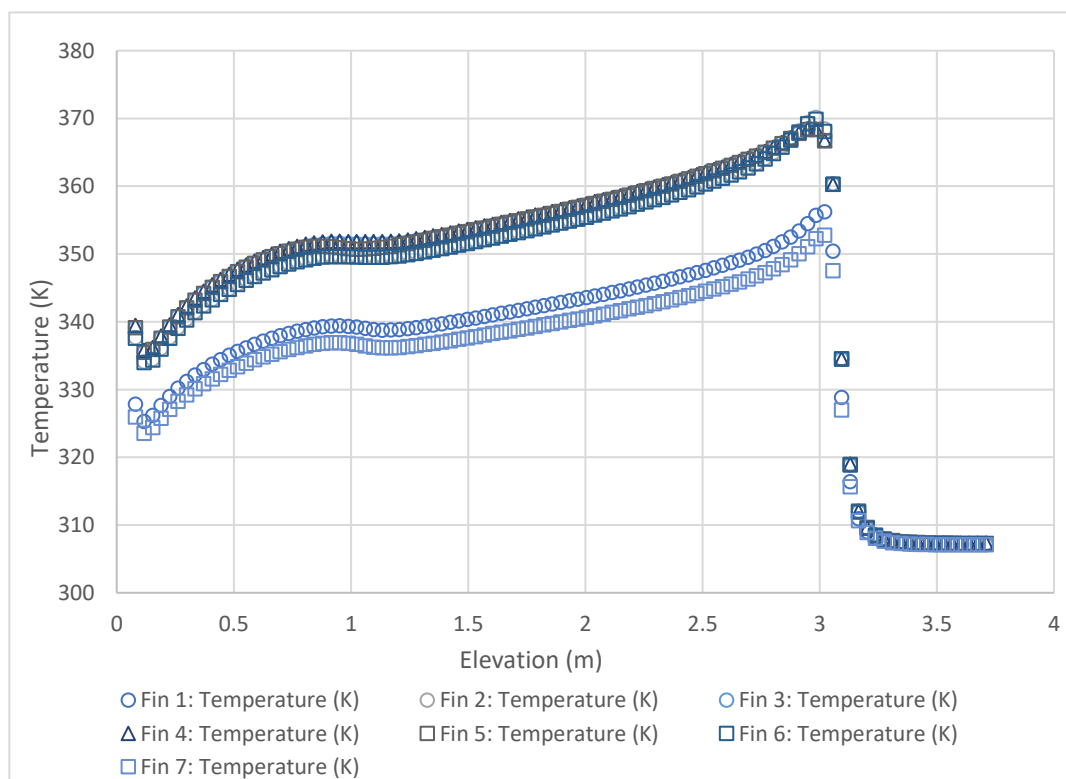


Figure 8.2-73. Fin temperature distributions for the case with rectangular tube with fins.

The increase in the number of riser tubes for the rectangular configurations is instrumental for the effects seen in Figure 8.2-72 and Figure 8.2-73. The cross-sectional area between the riser tubes was smaller in the rectangular configurations, causing the fluid in these zones to be cooled more effectively (by means of the stronger downward buoyancy force) than it is with the tubes spaced further apart. The geometry created almost a venturi effect between the riser tubes which accelerated the fluid downward, removing the heat from the higher elevations and transporting the heat to the lower elevations where the peaks were observed.

Figure 8.2-74 shows the temperature contour on the outer surfaces of the riser tubes and fins at the top part of the heated section. The figure shows that the hotspot on the fin surfaces increased towards the middle of the heated section. The figure also shows the cooling effect of the risers on the fin temperatures, which were ≈ 30 K cooler next to the risers.

Figure 8.2-75 shows the temperature contours on the outer surfaces of the riser tubes at the top of the heated section. The figure shows the hotspots that occur in the front corners of the riser tube walls. The other location where hotspots are visible are the locations where the fins connect to the riser tubes. The side surfaces of the riser tubes show variations of ≈ 20 K between the hottest and coldest points. The higher temperatures against the fins contributed in the stronger natural circulation force in the boundary layer against the fins, which resulted in the higher flow velocities in Figure 8.2-79 as compared with Figure 8.2-78.

The temperature distributions in the heater - and back wall are shown in Figure 8.2-76. The temperatures for each wall were obtained by line probes at three equally spaced positions. The distributions were very similar between the three probed positions for the two walls. There is a difference of approximately 260 – 300 K between the heater wall and the back wall. The shielding effect of the fins can be seen in the temperatures of the back wall increasing by only ≈ 20 K from the lowest to highest elevation, while the heater wall temperatures differ ≈ 70 K from the lowest to highest elevation. The heater wall temperature was 82-91% higher than the back wall.

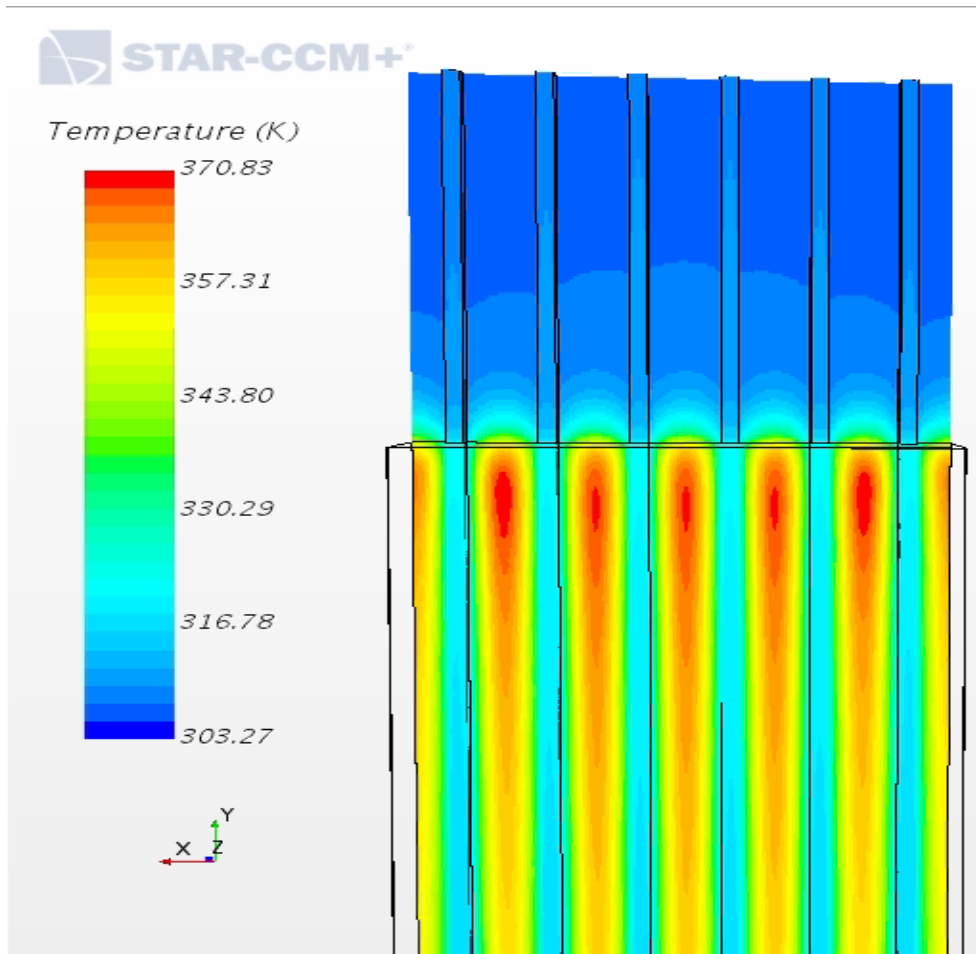


Figure 8.2-74. Temperature contour on the front surface of the risers and fins.

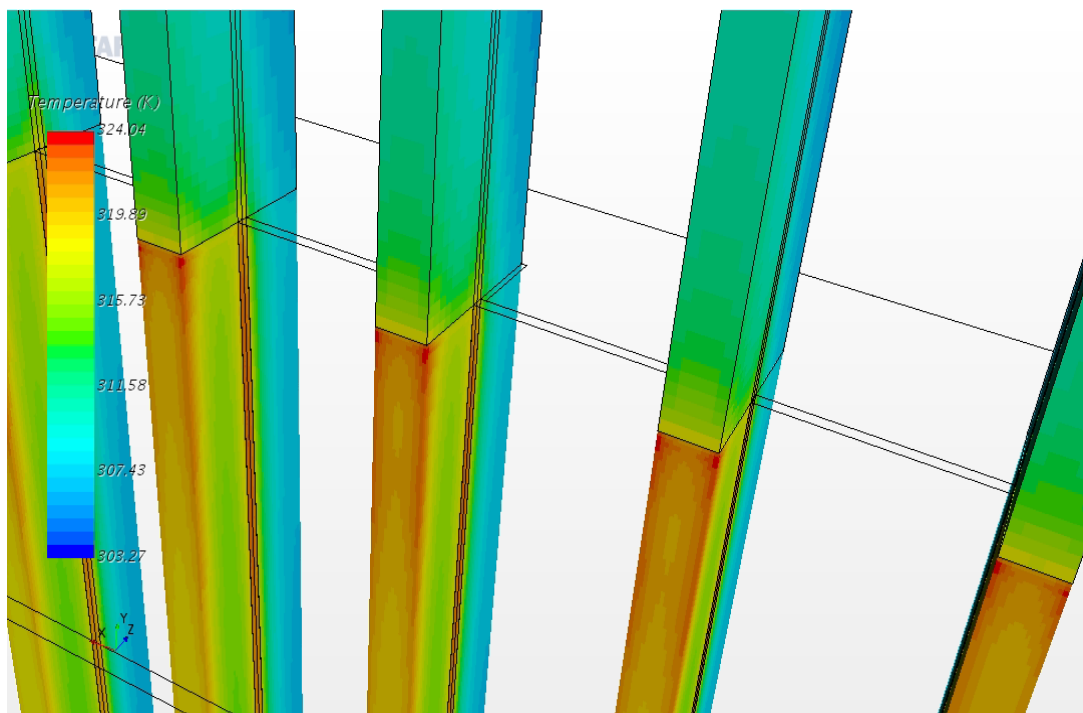


Figure 8.2-75. The temperature contour on the front and sides of the riser tubes.

The heater wall temperatures indicate the existence of a hot spot and a cold spot at the highest and lowest elevations respectively. A hotspot formed in the vicinity of the highest elevation since the rising hot air was slowed down by the presence of the cavity's top wall. The cooled air moving downwards on the riser tubes' and fins' surfaces were directed by the cavity's bottom wall towards the heater wall, which caused the lower elevations of the heater wall to be cooled down.

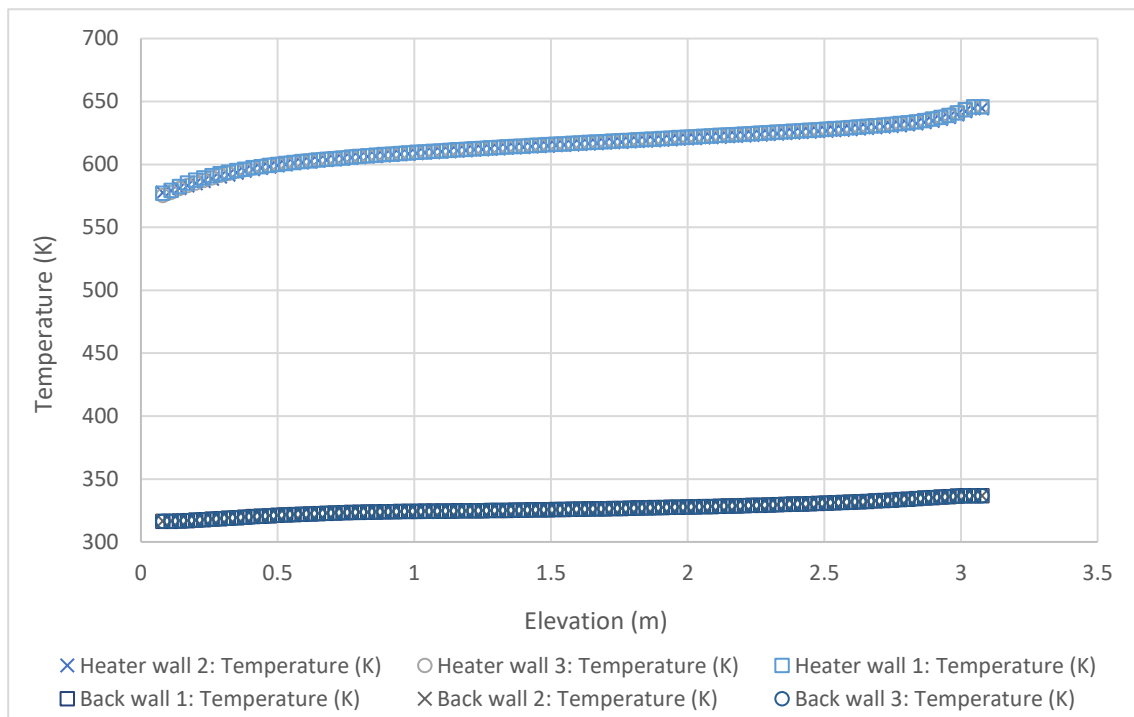


Figure 8.2-76. Heater - and back wall temperature distributions.

Figure 8.2-77 shows the cavity air temperature distributions for the front and back cavity. The temperatures were taken on line probes that were placed in the middle of the respective cavities. The distributions in each respective wall were approximately the same with negligible differences in the temperatures between the three locations where the probes were placed in

each wall. The hotspots in the front cavity are visible in the graph and was due to the fluid being slowed down by the top and bottom walls of the cavity. The front cavity air showed a 41% increase from the lowest to the highest elevation, while the back-cavity air showed a 9% increase over the same distance.

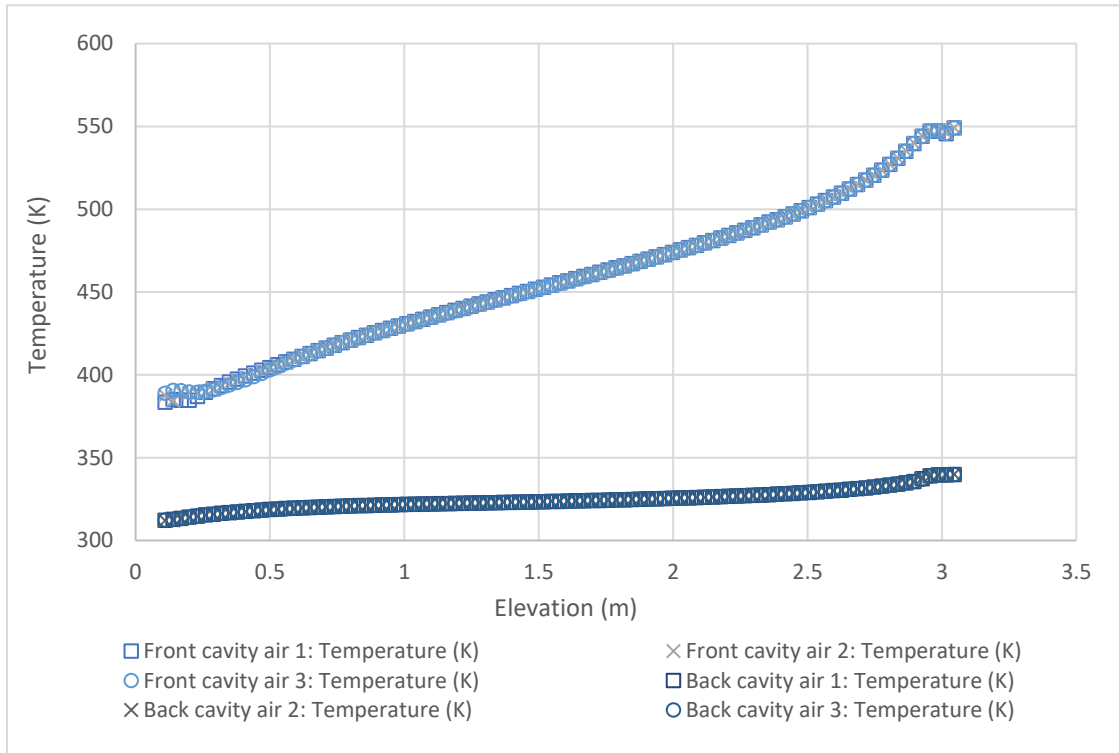


Figure 8.2-77. Cavity air temperature distribution for the case with rectangular tube with fins.

8.2.6.3 Velocity vectors

Figure 8.2-78 shows the velocity in the y-direction (parallel to the direction of gravity) taken on a line probe that was placed in front of the third riser in the front cavity. The velocities represent local values at each respective point. The graph shows the downward acceleration of the air in the top half of the heated section, followed by the rapid deceleration below 1.5 m into the denser cooler air in the bottom of the cavity. This effect caused the heat to be transported to the lower elevations of the heated section causing the peak that was shown in Figure 8.2-72.

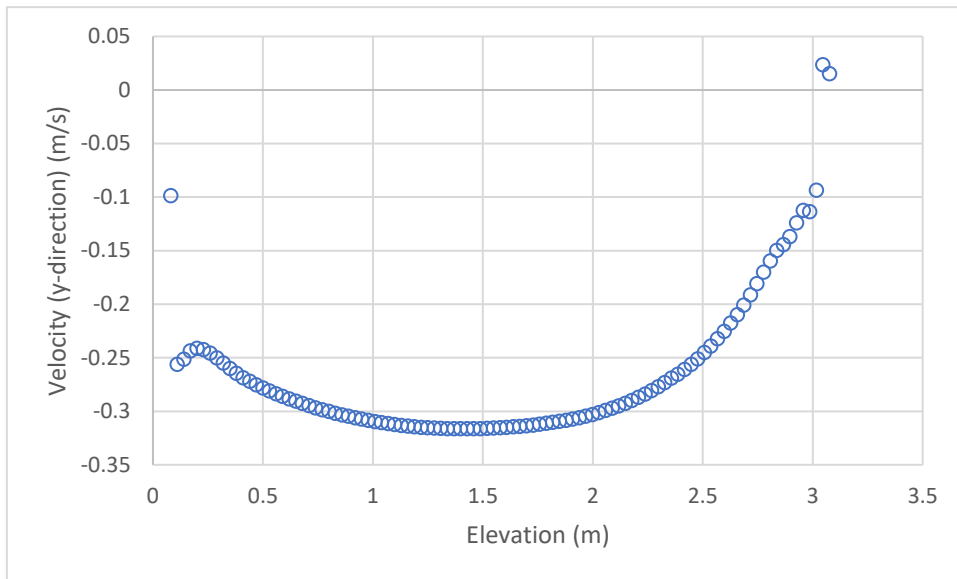


Figure 8.2-78. Velocity in the y-direction on a probe in the front cavity next to the third riser.

Figure 8.2-79 shows the velocity in the y-direction on a line probe taken in the front cavity in front of Fin 4. From the top elevations the air was accelerated downwards, slowing down at the elevation of 1.5 m, causing a similar temperature rise seen in the riser tubes to occur on the fin surfaces. The sampled velocities in front of the fins were higher than the corresponding velocities in front on the risers (87% larger maximum velocity). The temperature peak at the lower elevations (that was shown in Figure 8.2-73) were therefore lower for the fins, since the warmer air was in contact with the fins for a shorter time period.

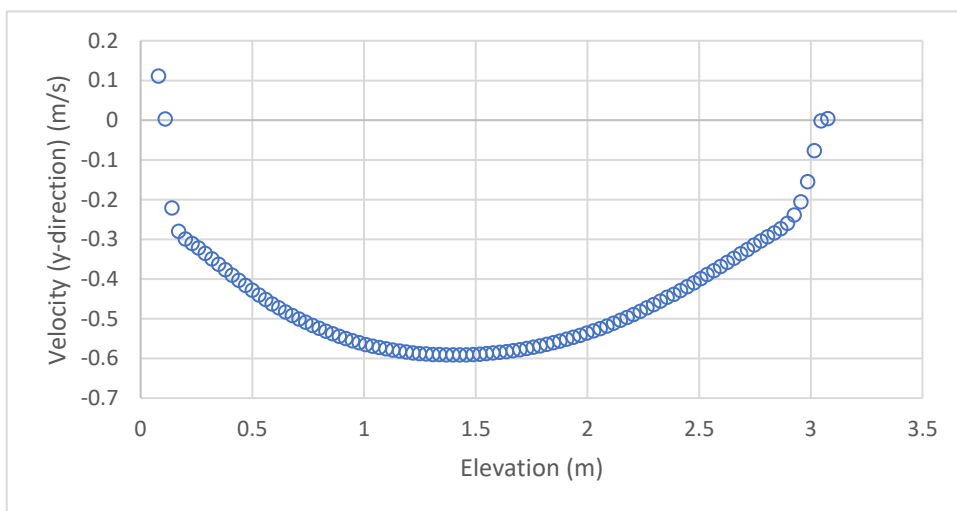


Figure 8.2-79. Velocity in the y-direction on a probe in front of Fin 4.

The flow patterns of the air in the cavity was also examined to gain insight into the phenomena in this region. A plane was used to make a section in the geometry on the YZ-plane on the fin between Riser 3 and 4. The vector scene obtained by this section plane is shown in Figure 8.2-80. The arrows were added to show the dominating flow directions. The flow was upward in the vicinity of the heater wall and downward on the fin's wall in the front cavity. A recirculating cell formed at the very top elevation against the fin in the front cavity. In both the front and back cavity, a recirculation cell formed around the turning point where the fluid flow direction changed from vertical to horizontal and vice versa.

The dot in Figure 8.2-80 shows a location where the flow in the y-direction was very small and is approximated by a stagnation point. The point lies between regions where the flow direction changes between upward and downward flow.

In Figure 8.2-80 the vectors shown in the back cavity are upward only. Another section plane was inserted in the XY-plane in the middle of the back cavity. The resulting vector scene is shown in Figure 8.2-81. Figure 8.2-81 shows that the main flow direction was upward in-line with the fins and downwards in line with the riser tubes.

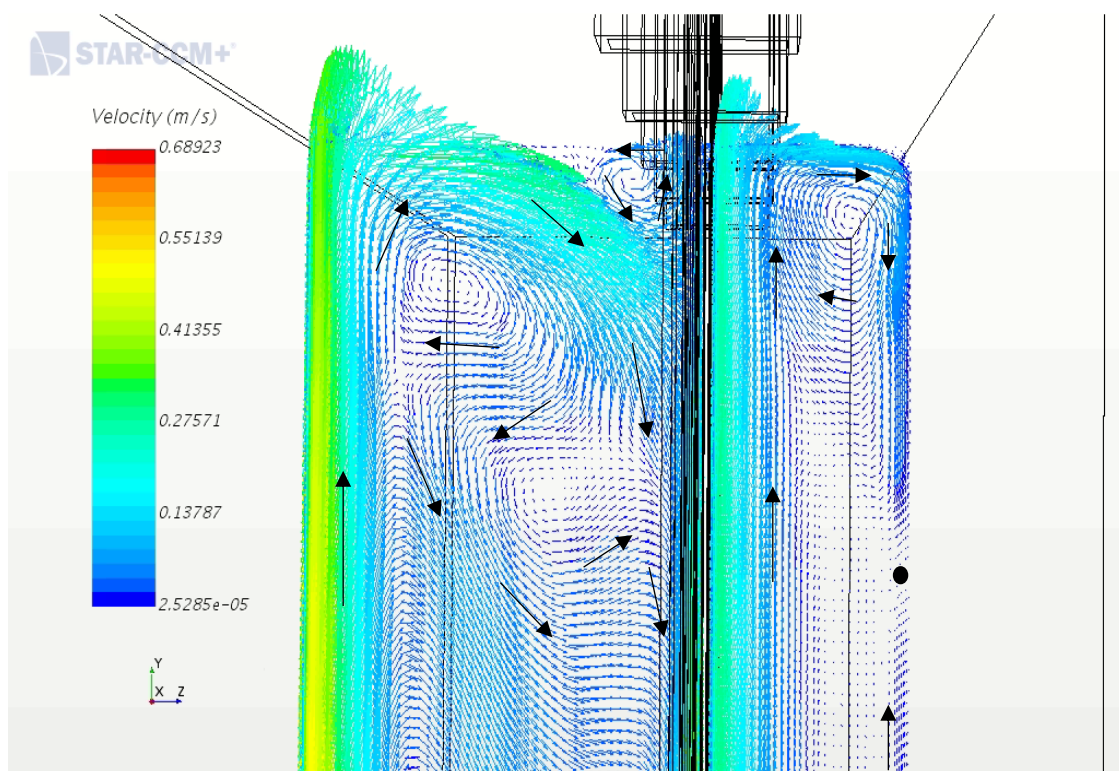


Figure 8.2-80. Vectors in the front and back cavity.

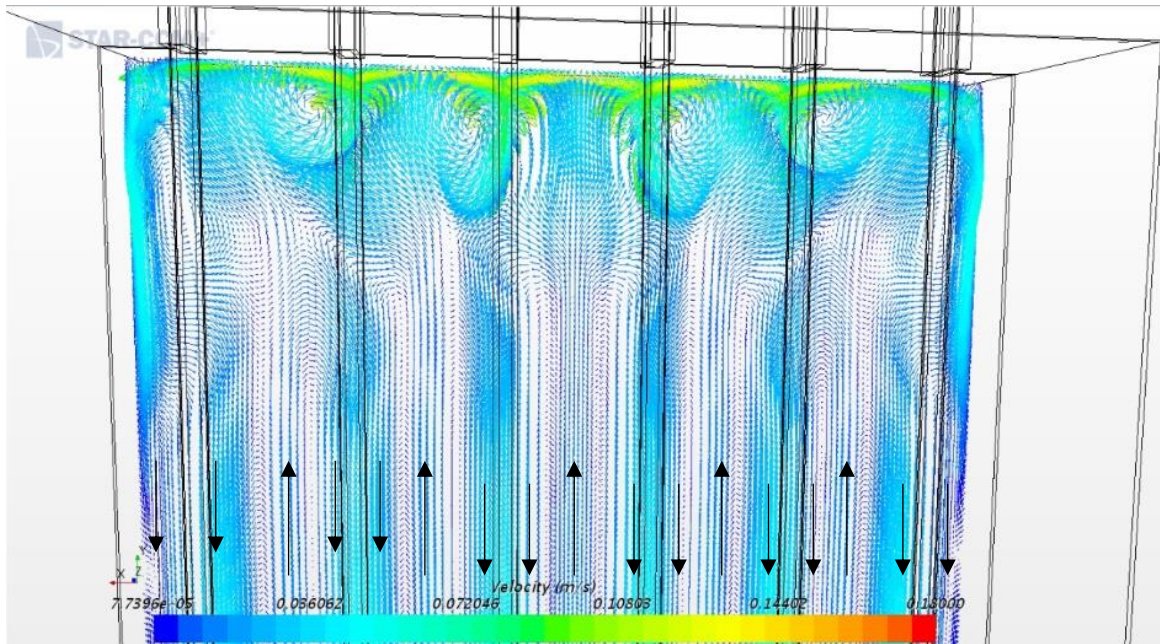


Figure 8.2-81. Vectors in the back cavity.

Figure 8.2-82 shows the velocity vectors on a section plane in the middle of the front cavity (XY-plane). The vectors show a dominating downward flow direction throughout the cavity except for in the direct vicinity of the side walls.

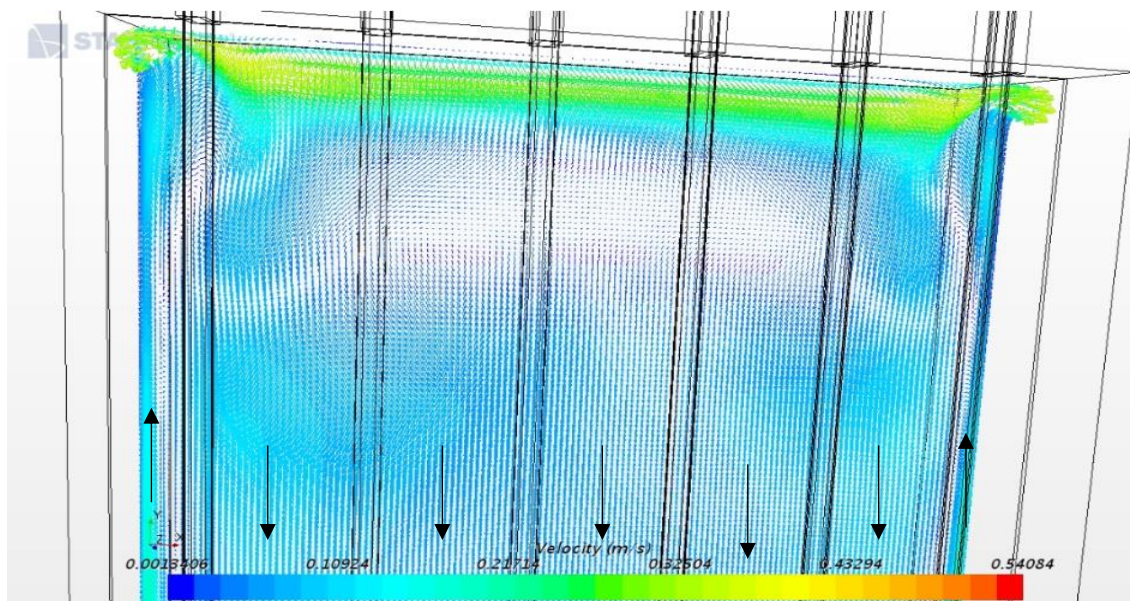


Figure 8.2-82. Vectors in the front cavity.

Figure 8.2-83 shows the velocity vectors in Riser 3 at four different elevations as viewed on the x-y plane. The profiles all resembled the velocity profile commonly associated with laminar flow. The effect of the fins on the velocity profiles seem negligible, in contrast with the previous geometric configurations.

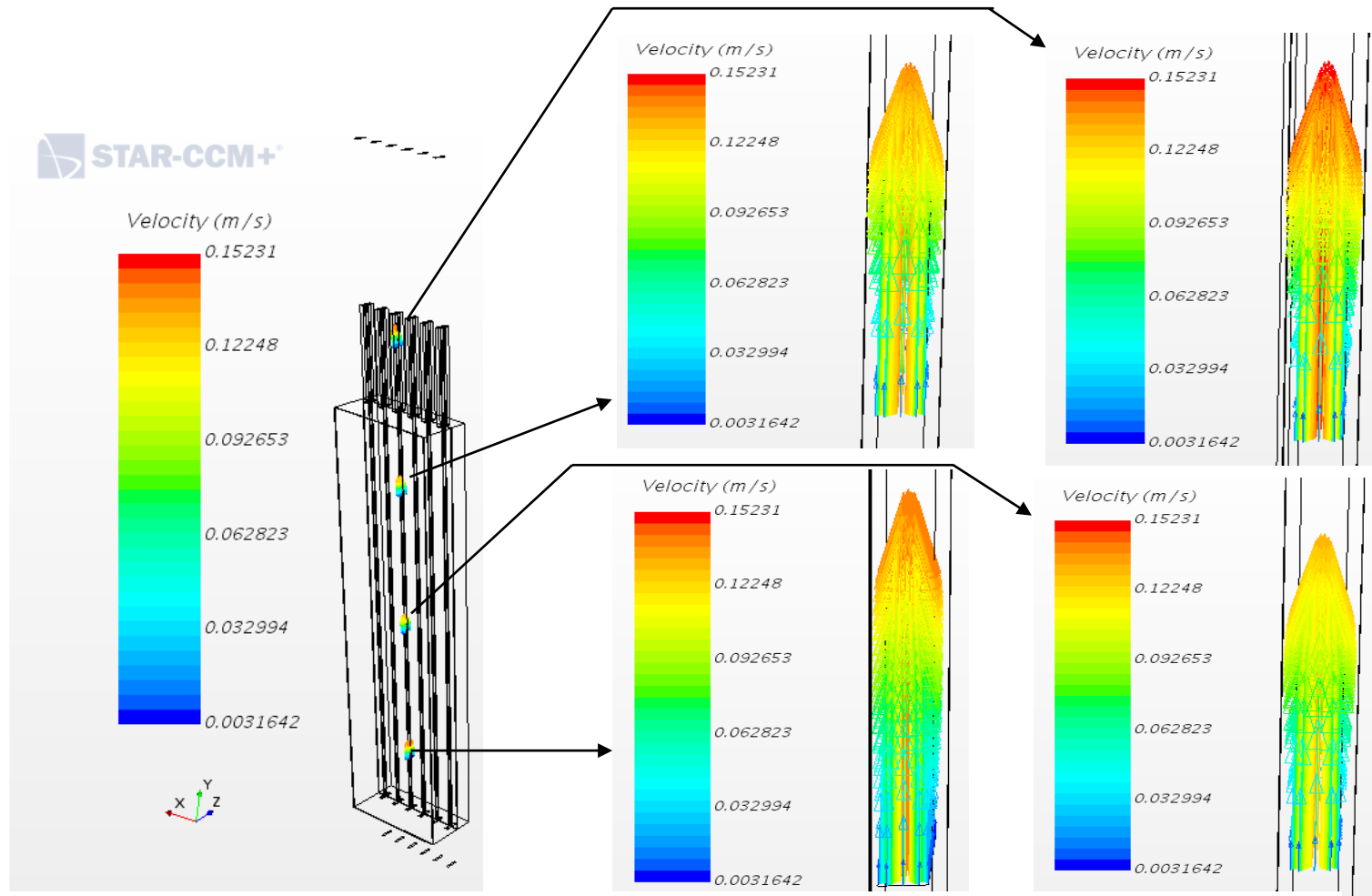


Figure 8.2-83. Velocity vectors in the one of the middle riser tubes.

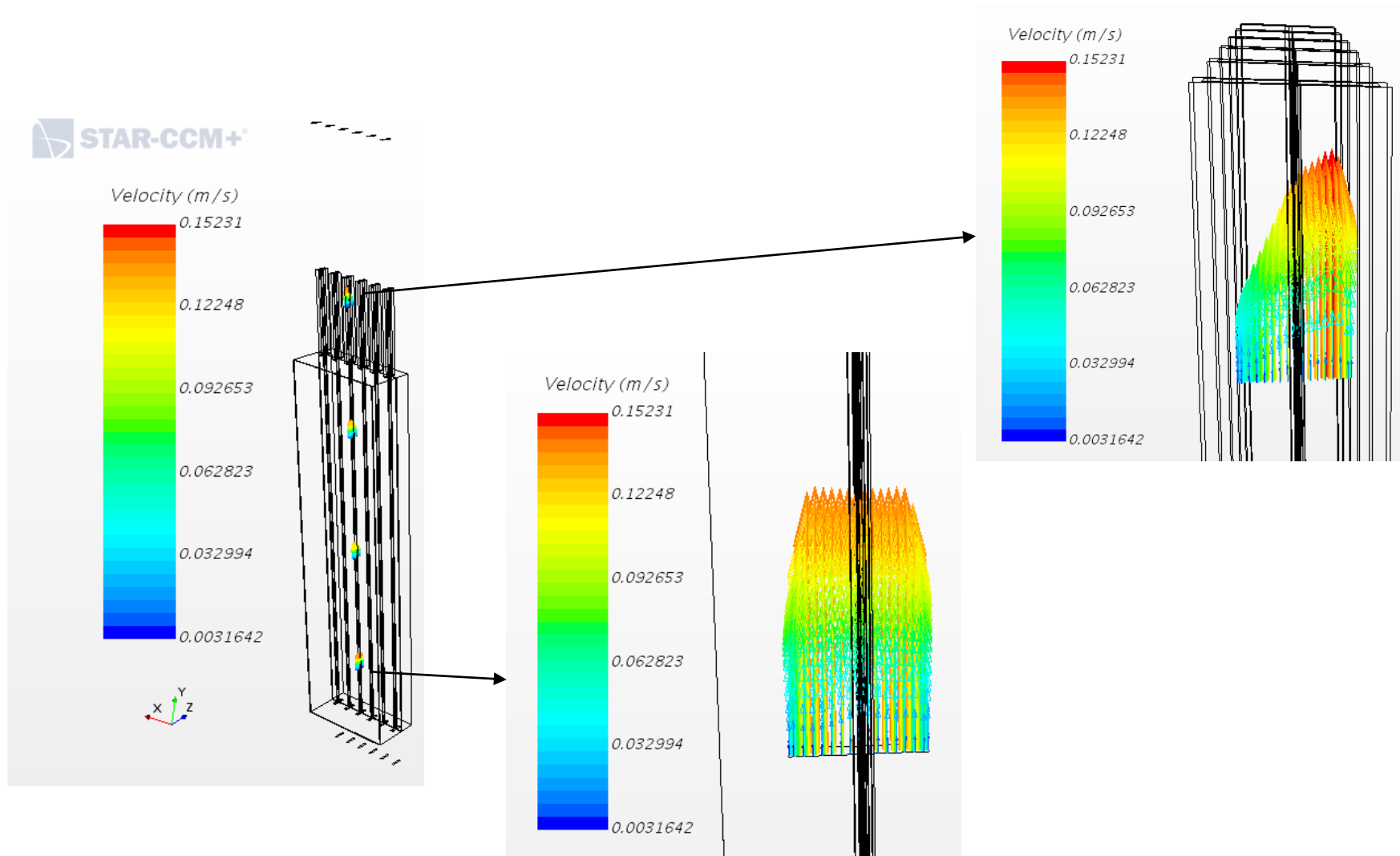


Figure 8.2-84. The velocity vectors inside one of the middle risers (continued).

Figure 8.2-84 shows the velocity vectors inside Riser 3 as viewed from the x-direction (i.e. on the y-z plane). The lowest elevation showed a symmetric velocity profile. The velocity profiles of the subsequent elevations was increasingly asymmetric. The highest elevation's velocity profile is also shown in Figure 8.2-84 and shows the highest velocities next to the front wall. Again it is apparent that the effect of the fins was negligible, in contrast to the previous configurations.

8.2.6.4 Heat transfer coefficients

The convection heat transfer coefficients were calculated by using the temperature distributions to calculate the non-dimensional parameters discussed in the theory chapter (see section 3.3). The coefficients for the inside surfaces (water side) of Riser 3 are shown in Figure 8.2-85. Riser 3 is considered as being representative of the other tubes since the coefficients of the other riser tubes were similar. The coefficients show an overall increase for the front and sides, with an overall decrease for the back side over the elevation. The peak at the lower elevations ≈ 0.5 m was due to the temperature peak in the temperatures within the riser wall, which caused a stronger buoyancy force and subsequently a higher coefficient at that elevation.

Note that the Reynolds number was between 2900 and 3400 for the flow in the riser tubes.

The temperature of the riser tubes' back wall was comparable with the water temperatures in the middle of the tubes, causing a weaker buoyancy force on the back walls compared with the other walls. The riser tubes' bulk fluid temperature increased at a faster rate than the back walls of the riser tubes, causing the decrease in the convection heat transfer coefficient on the back wall as seen in Figure 8.2-85. At the elevation of ≈ 2 m the fluid and wall temperatures on the back wall were almost the same, resulting in the minimum value that was observed. The front and side walls in contrast have more rapid increase in temperature with elevation that resulted in a stronger buoyancy force with an increase in elevation, resulting in the overall increase in the corresponding coefficient.

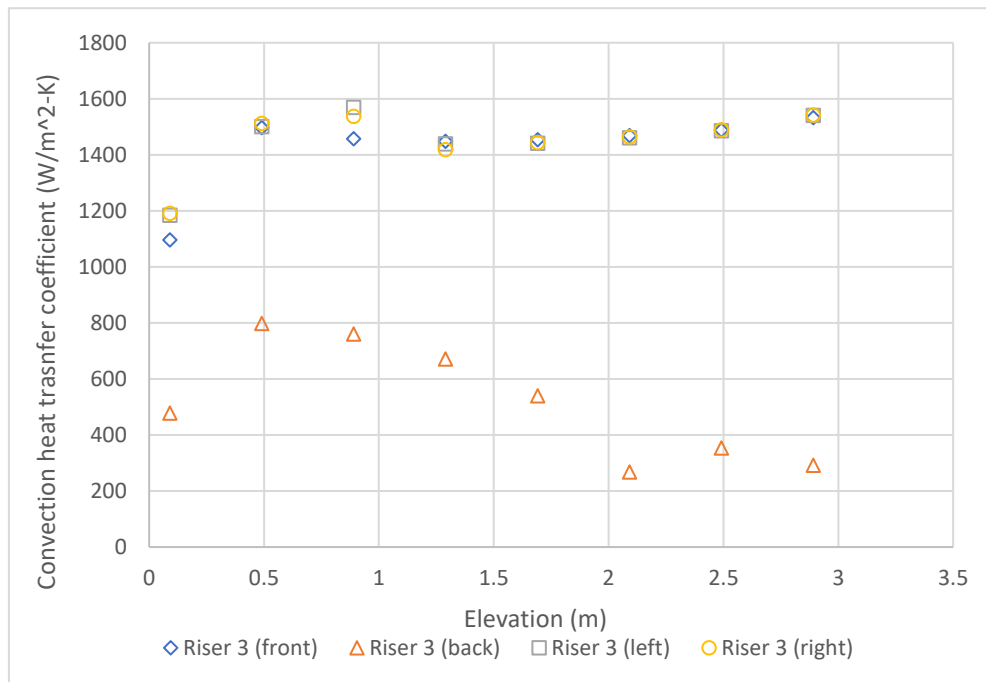


Figure 8.2-85. Convection heat transfer coefficients for the case with rectangular tubes with fins.

Figure 8.2-86 shows the Nusselt number ratio as a function of the Aicher number. The ratio is larger than one for the front and sides of the tube while being lower than 1 at two datapoints for the back wall. The ratios indicate a strong presence of the free convection regime. The ratios on the front and sides of the riser tube were between 4 and 4.5 with Aicher numbers that are similar, which led to the close proximity of the datapoints in Figure 8.2-86.

In Figure 8.2-86 the data points of the front and sides form a C shape. The lowest elevation (unheated) datapoints for these walls fall between an Aicher number between 0.25 and 0.3. The second elevation points have the maximum Aicher number and Nusselt number ratio. Subsequent points have progressively lower Nusselt number ratios, with an initially decreasing, followed by an increasing Aicher number, forming the C shape. The driving force behind the occurrence was the difference between the corresponding fluid and wall temperatures at each elevation. This difference initially decreased to approximately the middle elevation of the heated section, where after it increased again (which is reflected in the decrease followed by the increase in the Aicher number). The decreases in the Nusselt number ratio was driven by increases in the forced Nusselt number component.

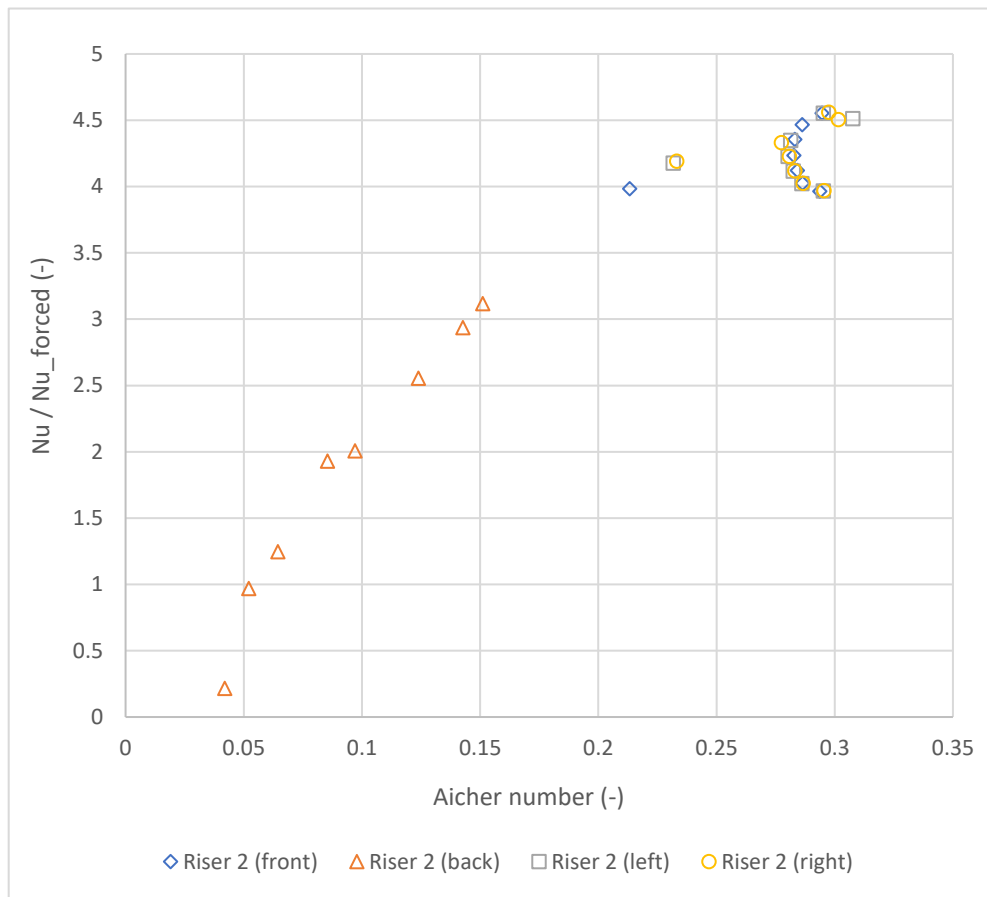


Figure 8.2-86. Nusselt number ratio vs. Aicher number for the case with rectangular tubes with fins.

Figure 8.2-87 shows the Aicher number as a function of elevation. The data points on the front and side surfaces have Aicher numbers larger than 0.2, indicating that the flow was within the free convection regime. On the back surface the regime was mixed convection until the highest elevation where the flow transitioned to pure forced convection. Figure 8.2-87 of the Aicher number and convection heat transfer coefficient (Figure 8.2-85) followed the same trends. The reasons have been discussed for the convection heat transfer coefficients and will therefore not be repeated for the Aicher number.

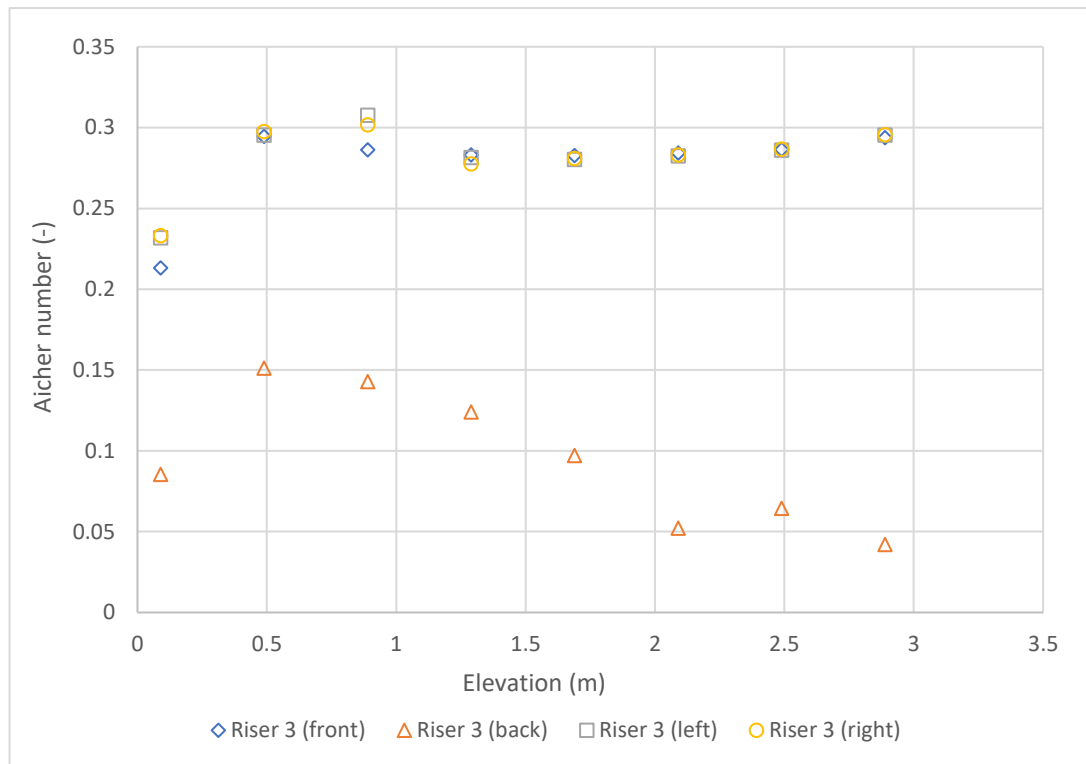


Figure 8.2-87. Aicher number as a function of elevation for the water in the riser tubes.

Figure 8.2-88 shows the convection heat transfer coefficients for the front and back cavity sides of the fins. The coefficients for the front sides of the fins show a noticeable increase over the elevation of the heated section. The coefficients on the back-cavity side however remain quasi constant since the fluid temperatures in the back cavity did not increase as much over the elevation as in the front cavity. Similar to the circular tubes and square tubes finned configurations, the behaviour of the coefficients for the front and back cavity are the opposite for the fin on the side vs. fins in the middle. In the front cavity the lower temperature that was sampled on the side fin (Fin 1) resulted in a larger difference between the fluid and wall temperatures, resulting in a higher Grashof number, which resulted in a higher convection heat transfer coefficient. In the back cavity, the lower temperature that was sampled from Fin 1 resulted in a smaller difference between the fluid and wall temperatures, resulting in a lower convection heat transfer coefficient than Fins 2, 3 and 4.

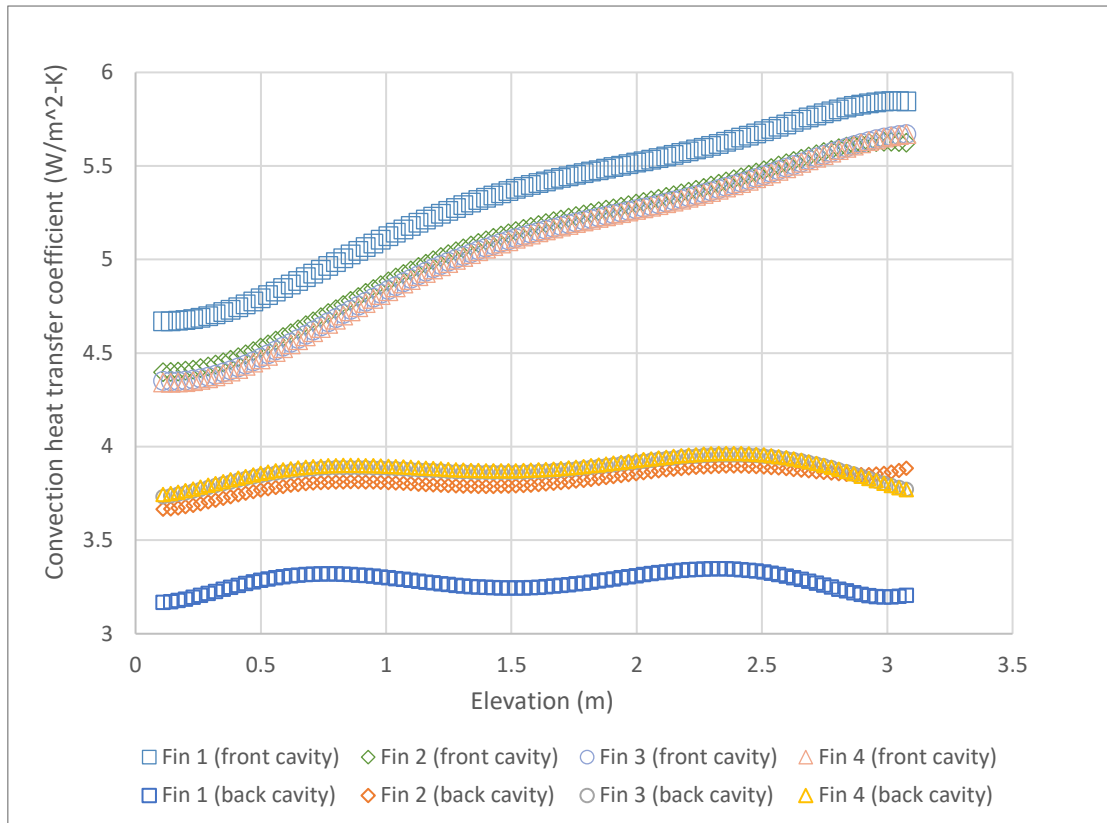


Figure 8.2-88. Convection heat transfer coefficients on the outer surfaces of the fins.

Figure 8.2-89 shows the convection heat transfer coefficient calculated for the front - and back cavity sides of the riser tubes. The coefficients show an overall increase over the elevation on both sides. On the front cavity side at approximately 0.5 m elevation the coefficients are slightly lower due to the peaks in the riser wall temperature at this elevation, which resulted in a smaller difference between the fluid and wall temperatures to result in a weaker buoyancy force. The heat at these low elevations were transferred to the back wall of the risers, causing a larger difference between the fluid and wall temperature, in turn causing the noticeable rise in coefficient on the back-cavity side. At the higher elevations (2.7 m – 3.1m) the coefficients on the back-cavity side show a further increase. The reason was a combination of the fluid temperature rising at this elevation due to the heated air converging at the top of the cavity and the conduction within the riser tube walls which was most pronounced at the high elevations. The fluid temperature however increased at the highest rate which caused an ever-increasing difference between the fluid and the wall temperature, subsequently resulting in the increase in the coefficient shown in Figure 8.2-89.

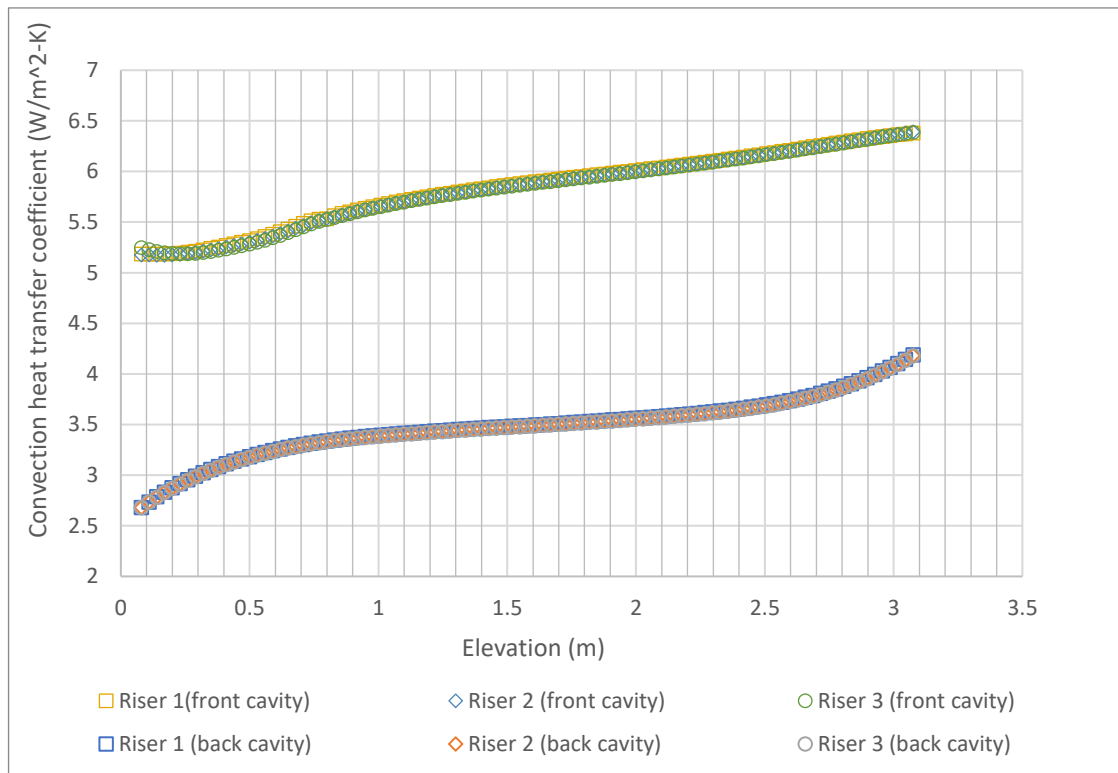


Figure 8.2-89. Convection heat transfer coefficient on the riser tubes' outer surfaces for the case with rectangular tubes with fins.

Figure 8.2-90 shows the convection heat transfer coefficients as a function of the elevation on the heater and back wall. The coefficients on the back wall show an overall decrease over the elevation, caused by the increase in the cavity air temperature. At the elevation of ≈ 2.8 m the back-cavity coefficients show a sharp decrease, which was due to the fluid and wall temperatures being almost the same at this location. Above 2.8 m the difference between the fluid and wall temperature increases which leads to the increase and recovery of the coefficient. Additionally, the flow direction changes at the topmost elevations against the back wall because of a recirculating flow that exists at this location. This formed a 'stagnation point' that was detrimental to convection heat transfer and explains the decrease in convection heat transfer coefficient at this location.

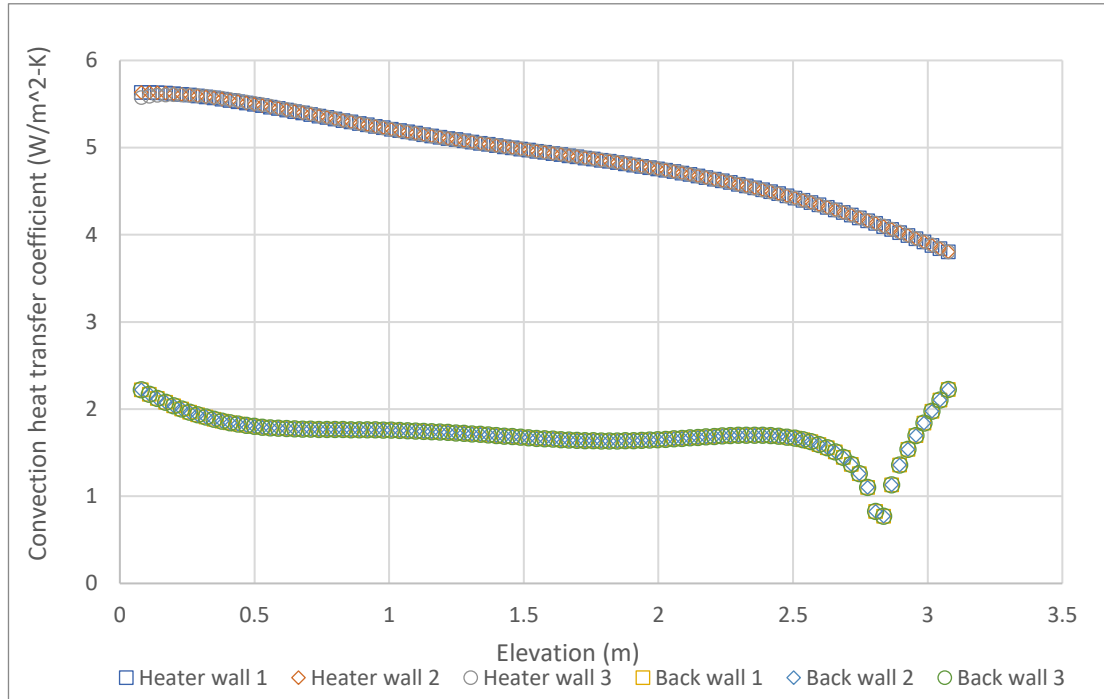


Figure 8.2-90. Convection heat transfer coefficient for the heater - and back wall as a function of elevation for the case with rectangular tubes with fins.

The parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix C.13.

8.2.6.5 Conclusion

The geometry of the baseline UW RCCS facility was changed to contain rectangular tubes instead of circular tubes. The number of tubes were also doubled from the baseline case. The total cross-sectional area was kept the same, as were the boundary conditions.

The temperature distributions and velocity vectors were investigated in detail. The heat flux distributions on the various surfaces in the cavity was also investigated. The velocity vectors revealed that the flow was accelerated downwards next to the riser tubes and fins in the front cavity. This caused a hotspot at the lower elevations in the air in the heater box.

Velocity vectors were shown for the flow inside the middle riser, which provided an insight into the prevailing flow regime inside the riser tubes.

The recirculation patterns that occurred at the top elevations of the front and back cavity was also discussed.

The Aicher numbers indicated that the regime in the riser tubes was fully developed free convection on the front and side surfaces, while being in the mixed convection and forced convection regime on the back surface.

The ratio of radiation/total heat transfer was 4% higher for the front cavity and 2% higher for the back cavity compared with the case with circular tubes with fins. This was because the higher surface area of the rectangular tubes, exposing more surface area to the radiating surfaces in the cavities.

8.2.7 Rectangular tubes without fins

The geometry used to simulate the case with rectangular tubes without fins is shown in Figure 8.2-91. The components that are referred to in the discussion of the results are indicated. The cavity consisted of only one part but was considered as the front and back parts identical to the corresponding finned case.

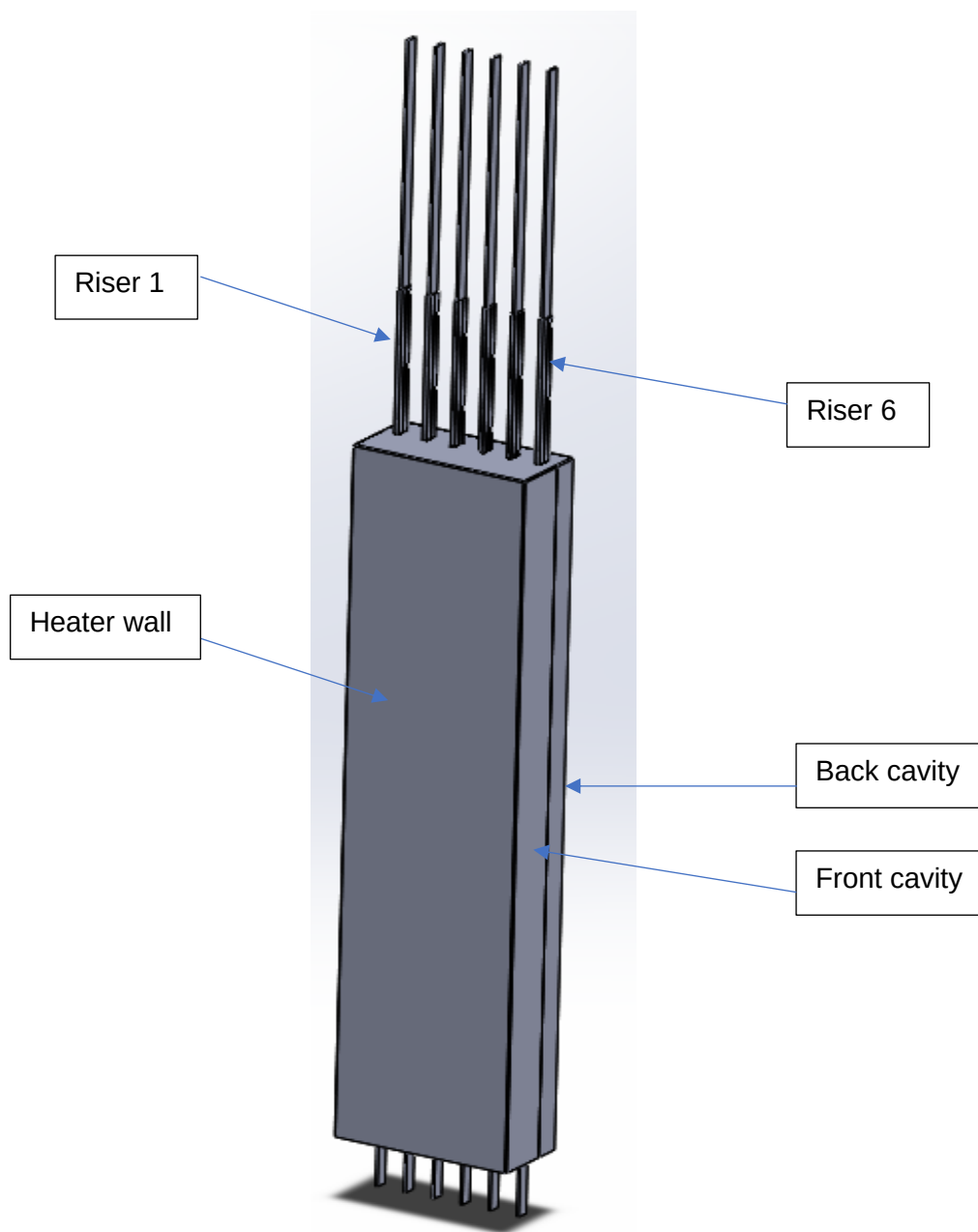


Figure 8.2-91. Geometry used for the CFD simulation.

8.2.7.1 Heat fluxes

Selected results pertaining to the surface average of the heat fluxes are shown in Table 8-6. Table 8-6 distinguishes between the boundary heat flux on the front and back part of the riser tube surfaces (from the cavity's side). The radiative heat flux was computed for the front and back part of the risers' outer surfaces. The integral on the heat flux was also calculated to obtain the heat energy added to the riser tubes' outer surfaces from the front and back part of the cavity. Lastly, the ratio of radiation to total heat transfer was calculated as a percentage. Note that for example the front side represents the front surface of the riser tube as well as ½ of the length of the sides.

Table 8-6. Selected results from the case with rectangular tube without fins.

	Heat flux (W/m ²)		Radiative heat flux (W/m ²)		Heat energy (W)		Ratio of radiation/total heat flux %	
	Front side	Back side	Front side	Back side	Front side	Back side	Front side	Back side
Riser 1	3004.81	2547.66	2191.54	1680.29	935.00	792.75	72.93	65.95
Riser 2	3125.45	2590.28	2260.09	1757.95	972.54	806.02	72.31	67.87
Riser 3	3162.04	2672.54	2333.24	1839.51	983.93	831.61	73.79	68.83
Riser 4	3165.23	2671.85	2333.27	1839.53	984.92	831.40	73.72	68.85
Riser 5	3125.22	2591.00	2260.35	1758.71	972.47	806.24	72.33	67.88
Riser 6	3005.04	2548.13	2191.40	1681.01	935.08	792.90	72.92	65.97

The heat flux distributions in Table 8-6 show that the middle risers (#3 and #4) received between 1.1 and 1.3% higher flux than Risers 2 and 5, which in turn received 2.3% higher flux than Risers 1 and 6 on the front side. The heat energy added from the back cavity was 18% less than the energy added from the front cavity. The majority of the heat transfer occurred by radiation, with heat by convection making up between 26 and 35%. There is less than 10 % difference between the ratio that radiation contributes between the front and back cavity.

8.2.7.2 Temperature distributions

The water temperatures as measured on line probes in the middle of the riser tubes are shown in Figure 8.2-92. The first dashed line in the figure indicates the elevation at which the risers enter the heated section and the second indicates the risers' exit point from the heated section. The temperatures in the middle weren't immediately affected after entering the heated section and kept increasing after exiting the heated section. The heat was first absorbed by the boundary layer, which then rose upwards due to buoyancy forces, which meant the temperature in the middle didn't immediately equalize. While the water is within the heater box, the fluid next to the wall will be hotter than the fluid on the centreline, because of the temperature gradient required to transfer heat from the wall to the fluid. After exiting the heater box, the walls become adiabatic and the temperature across the cross section will endeavour to equalize. This is why the water temperature shows a 1 K increase after leaving the heated section as the temperature in the middle of the riser was heated by the warmer boundary layer, equalising the temperature over the cross-sectional area of the risers.

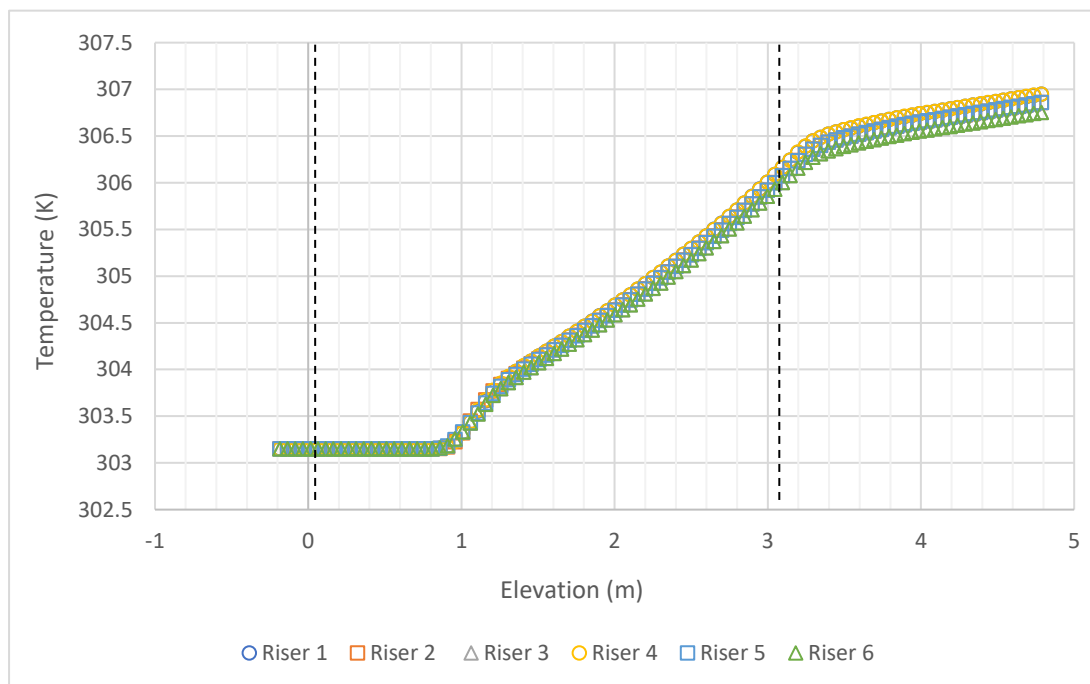


Figure 8.2-92. Temperature distributions in the middle of the riser tubes.

The temperatures within the riser tube walls are shown in Figure 8.2-93. Temperatures are shown for four sides of the riser tubes. Riser tubes 4 through 6 are omitted, since the phenomena were symmetric meaning Riser tubes 1 through 3 were representative of the results in the other riser tubes. The temperatures on the front sides were the highest, with the temperatures on the sides being the lowest. Temperature peaks were observed for the

elevations below 1 m on all of the surfaces. The mechanism was similar to the corresponding finned case, except in this case the flow can pass between the tubes, also affecting the temperatures. The heat was deposited below 1 m elevation by the downward flow of the warmer air on the risers' surfaces. At an elevation of 1.5 m the downward flow encountered the denser cooler fluid which rapidly decelerated the stream that was transporting the heat. Subsequently the heat build-up occurred in this region. The temperatures on the sides can be seen in Figure 8.2-93 to be lower than the temperatures for the back surfaces of the riser tubes. Without the shielding effect of the fins the cavity back wall had a higher temperature allowing more direct radiation heat transfer to occur from the cavity back wall to the back walls of the riser tubes. The riser tubes' side walls saw improved cooling by convection from the downward flow that formed between the riser tubes in the absence of the fins. The noticeable drop in the temperatures above 3 m elevation was due to the risers' exit from the heated section. At the outlet the temperature distribution around the walls of the riser tubes was more uniform, with a maximum of 2 K difference in temperature.

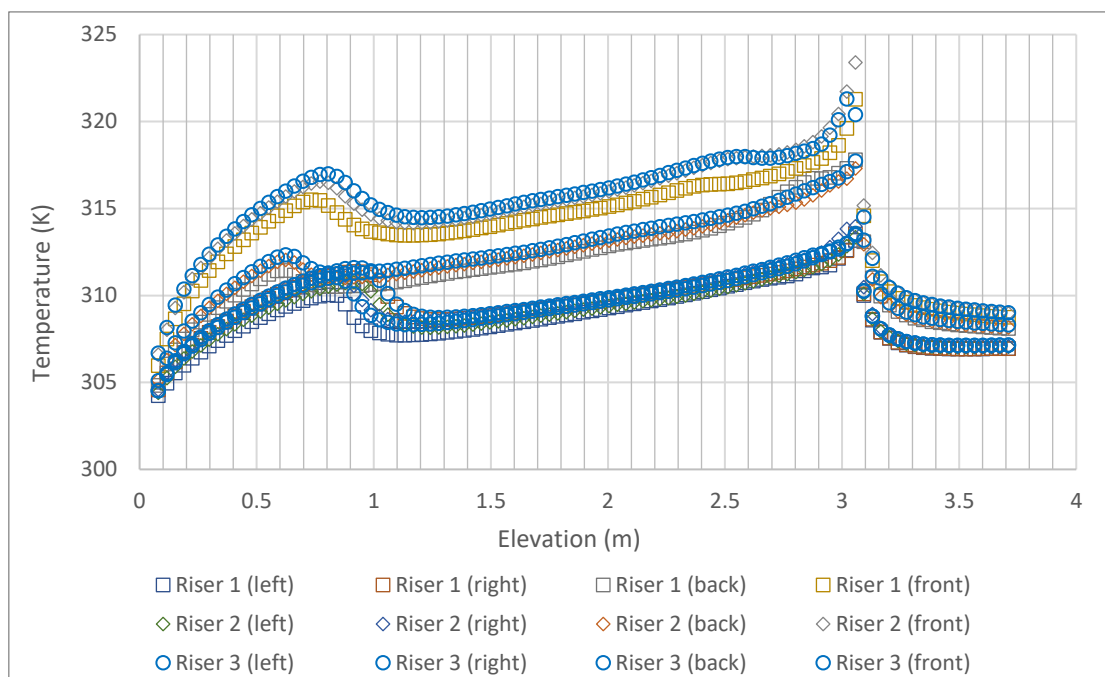


Figure 8.2-93. Riser tubes' wall temperature distributions for the case with rectangular tubes without fins.

Figure 8.2-94 shows the temperature contour on a section plane taken in between Riser 2 and 3, perpendicular to the heater wall. The contour reflects the upward movement of air next to the heater wall and the back wall and the downward movement of air next to the riser tubes. The average temperature of the air in the front and back cavity increases from the bottom to the top.

The shape of the contours near the heater and back walls imply that the air velocity was higher next to the heater wall compared to the back wall. In the region of the riser tubes the downward velocity seems to be the highest on the front cavity side dragging the hotter air further down (indicated by the dashed line in Figure 8.2-94).

The cross-sectional area between the riser tubes was reduced by the use of 6 riser tubes, as opposed to the 3 riser tubes in the square and circular tubes cases. This meant more effective cooling of the fluid between these locations and a stronger downward buoyancy force, enhanced by a venturi effect, which accelerated the flow downward. The downward acceleration of the heated air caused the heat transport away from the higher elevations towards the locations where the hotspots were observed.

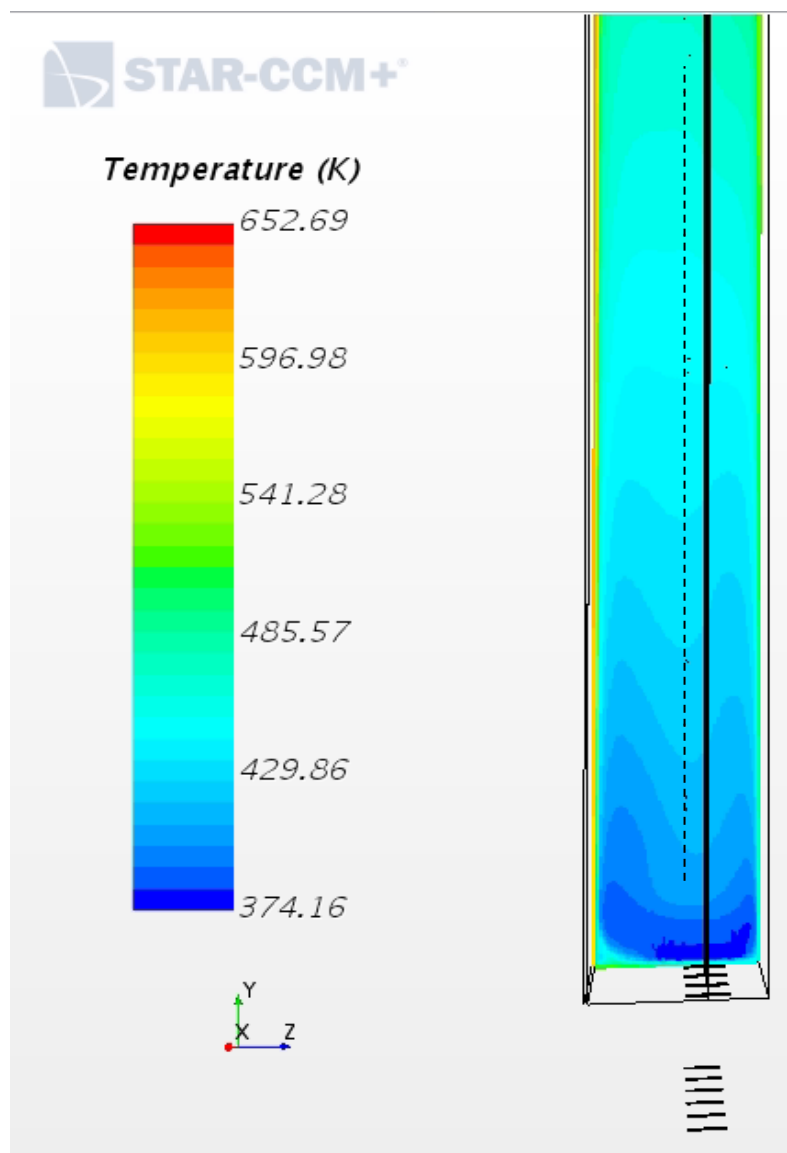


Figure 8.2-94. Temperature contours in the lower part of the heated section.

Figure 8.2-95 shows the temperature distributions on the heater – and back wall. The back wall temperatures were between 20% and 29% lower than the temperatures on the heater wall. The heater wall temperatures show an increase at the topmost elevations, since the rising hot air in the boundary layer got slowed down by the top wall of the cavity. This formed the hotspot that caused the temperature rise in the heater wall at that location. On the back wall, the temperatures plateaued because the flow patterns didn't allow for transport of the warmer air to the topmost part of the back wall (see the velocity vectors in Figure 8.2-98).

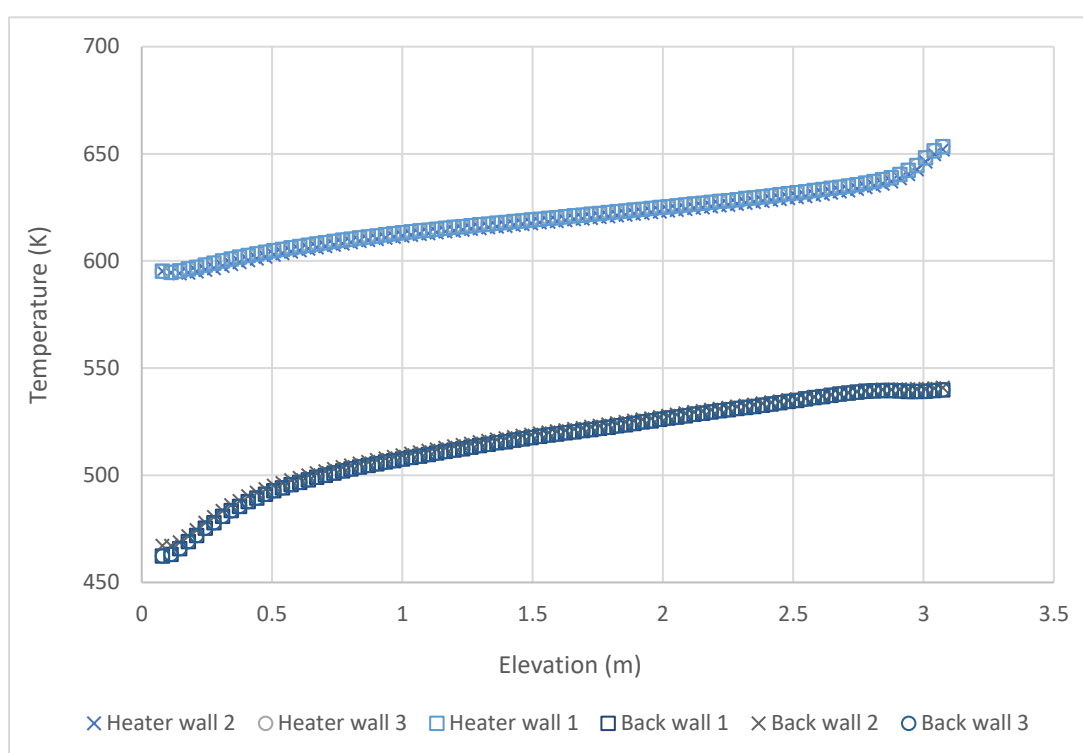


Figure 8.2-95. Temperature distributions on the heater – and back wall.

Figure 8.2-96 shows the cavity air temperature distributions in the front and back part of the cavity. The temperature distributions were very similar, since the omission of the fins allowed the air of the front and back cavities to mix. The temperatures at the topmost elevations of the front cavity show a slight decrease, with a simultaneous increase in the back-cavity side. This was due to the heated air being transported from the front cavity, between the tubes and into the back cavity.

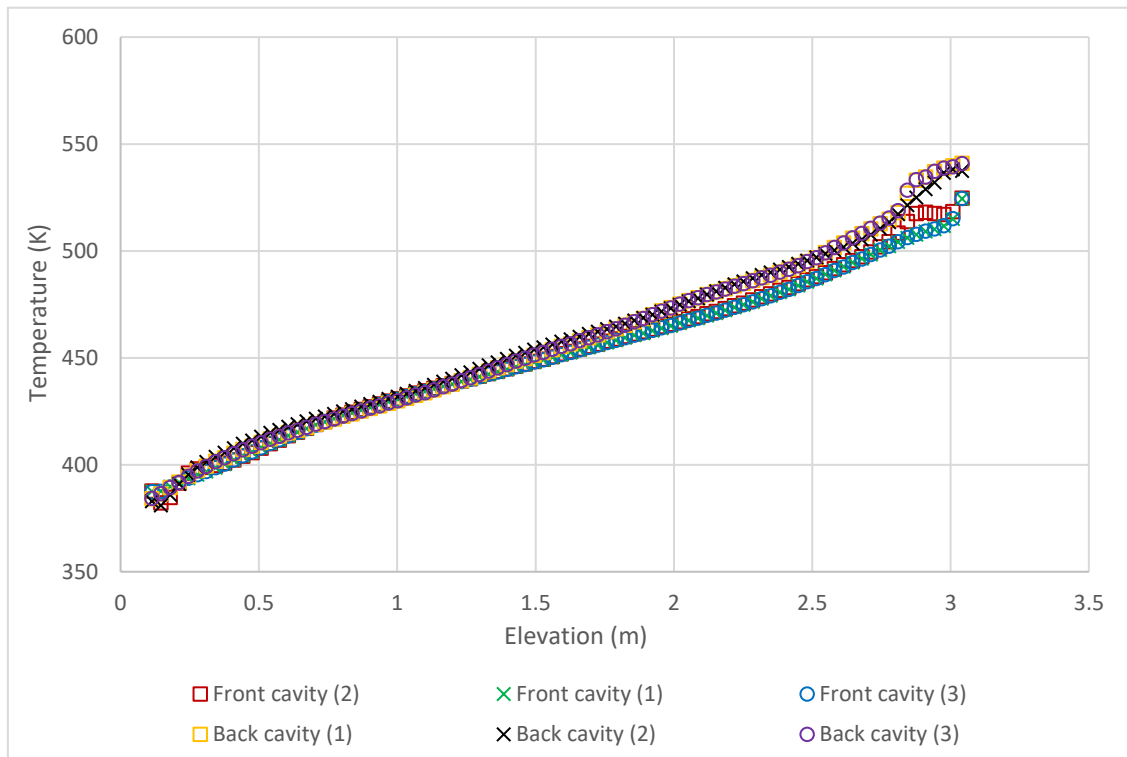


Figure 8.2-96. Cavity air temperature distributions in the front - and back cavity.

8.2.7.3 Velocity vectors

The velocity vectors were taken on a line probe between the second and third risers and is shown as a function of elevation in Figure 8.2-97. The velocity component, specifically in the y-direction is considered (parallel with the direction of gravity). The graph shows the downward acceleration of the fluid from the higher elevation until the elevation of ≈ 1 m is reached. Below 1 m the fluid was decelerated causing the heat to build up in this region resulting in the peaks seen in the riser wall temperatures (that were shown in Figure 8.2-93). Since the velocities were sampled from point locations the jumps in velocity are observed. The locations of the jumps are located on cells where the velocity components in the other directions were minimal and the y-direction vectors were at a maximum.

The velocity vectors were also generated on section planes to provide qualitative results on the flow patterns within the heated section. Figure 8.2-98 shows the velocity vectors at the higher elevations in the front and back part of the cavity. The vectors are located on a section plane that intersects the geometry between Riser 3 and 4, perpendicular to the heater wall. The

arrows were added to show the dominating flow direction. Three recirculation cells are visible and are indicated; two in the front part and one in the back part of the cavity.

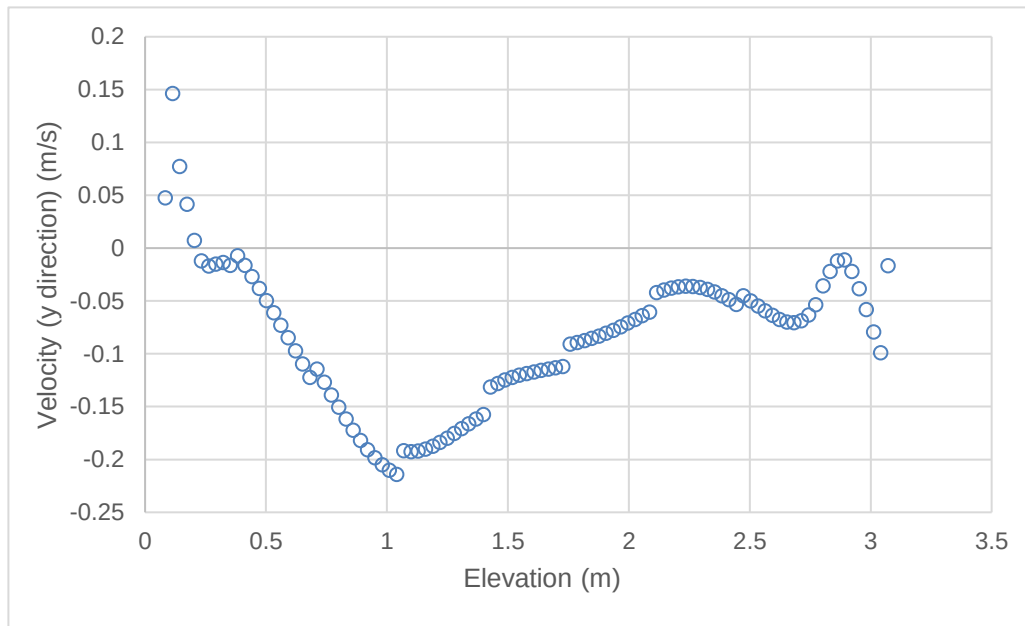


Figure 8.2-97. Local y-velocity between second and third risers.

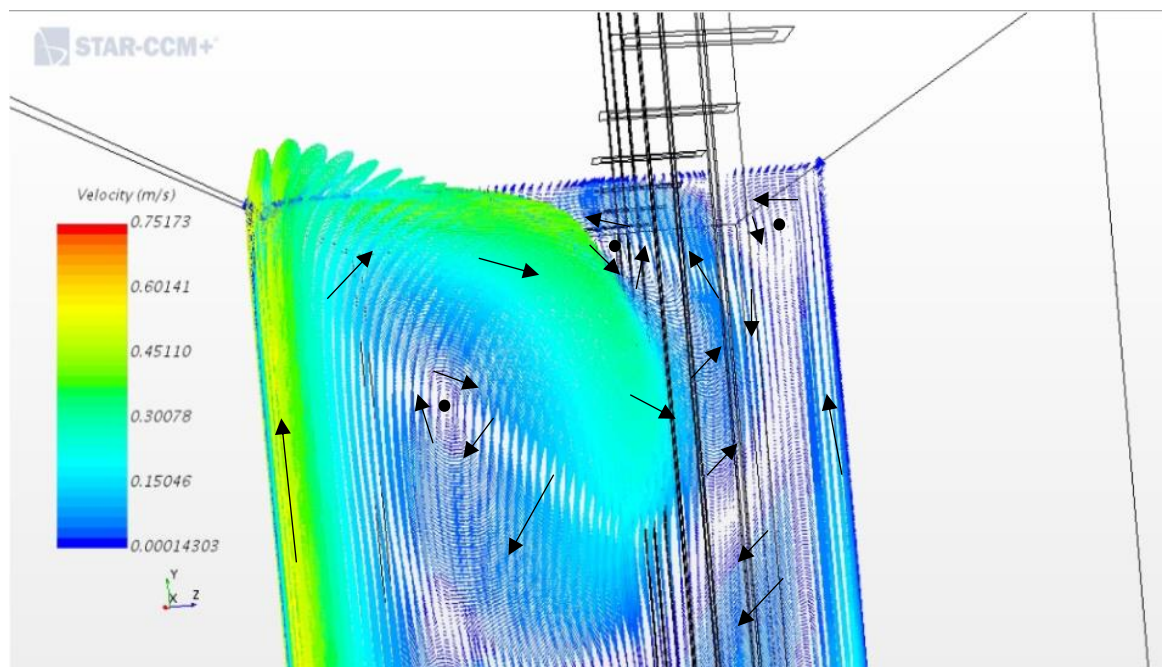


Figure 8.2-98. Velocity vectors of the air in the front and back part of the cavity.

Figure 8.2-99 shows the velocity vectors in the back part of the cavity. The vectors are on a plane section that intersected the back cavity approximately half way between the riser tubes and back wall and was parallel to the back wall (perpendicular to the section plane in Figure 8.2-98). The vectors show downward flow for most of the elevations in the back cavity. The flow field was especially complex in the top part of the cavity, the maximum velocities were at the top elevation and were in the direction normal to the figure.

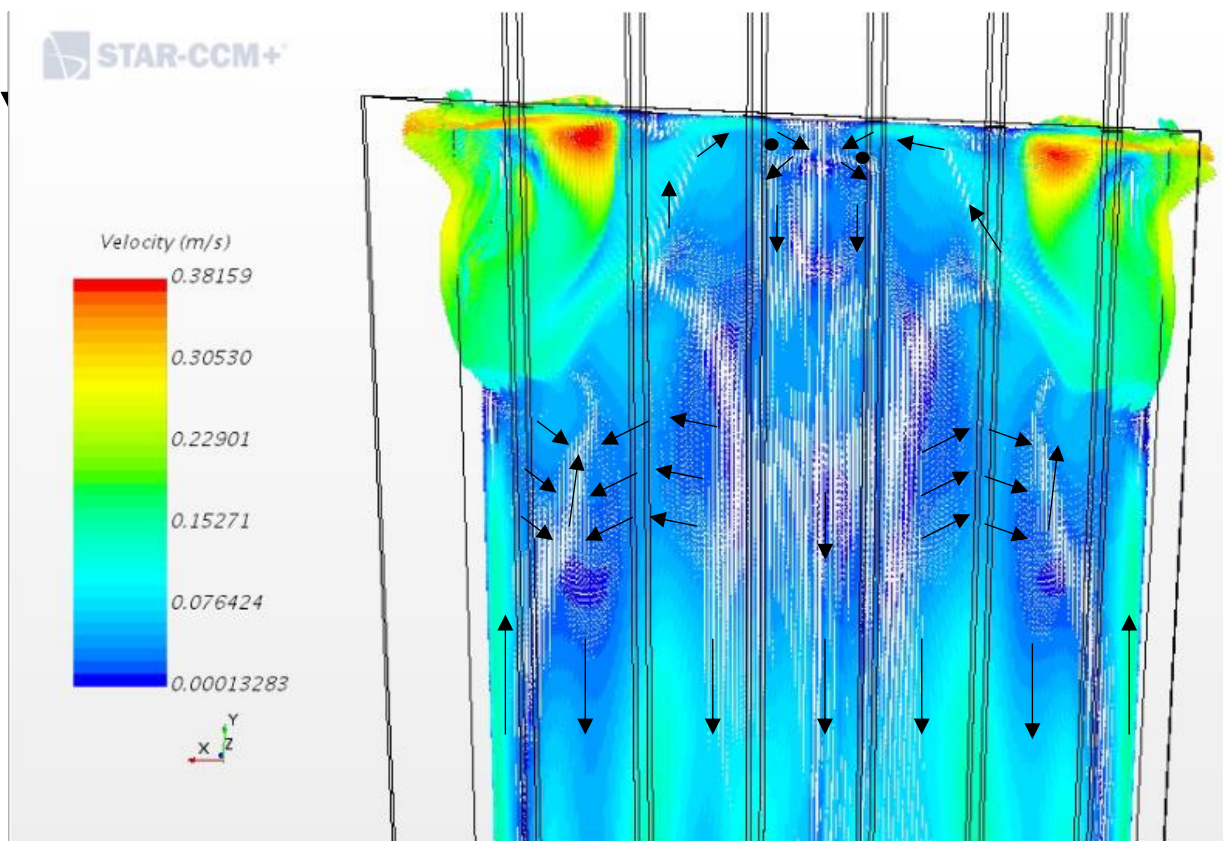


Figure 8.2-99. Vectors in the middle of the back cavity.

Figure 8.2-100 shows the velocity profiles at four different elevations inside Riser 3 as viewed on the x-y plane. The profiles all appear to resemble the profile commonly associated with laminar flow. The velocities appear to decelerate from the lowest elevation to the second elevation. This was possibly due to the buoyancy effects close to the walls forcing the profile to spread out across the area, thereby decreasing the flow velocity. The profile recovered some of the flow velocity from the heating effects by the top elevation.

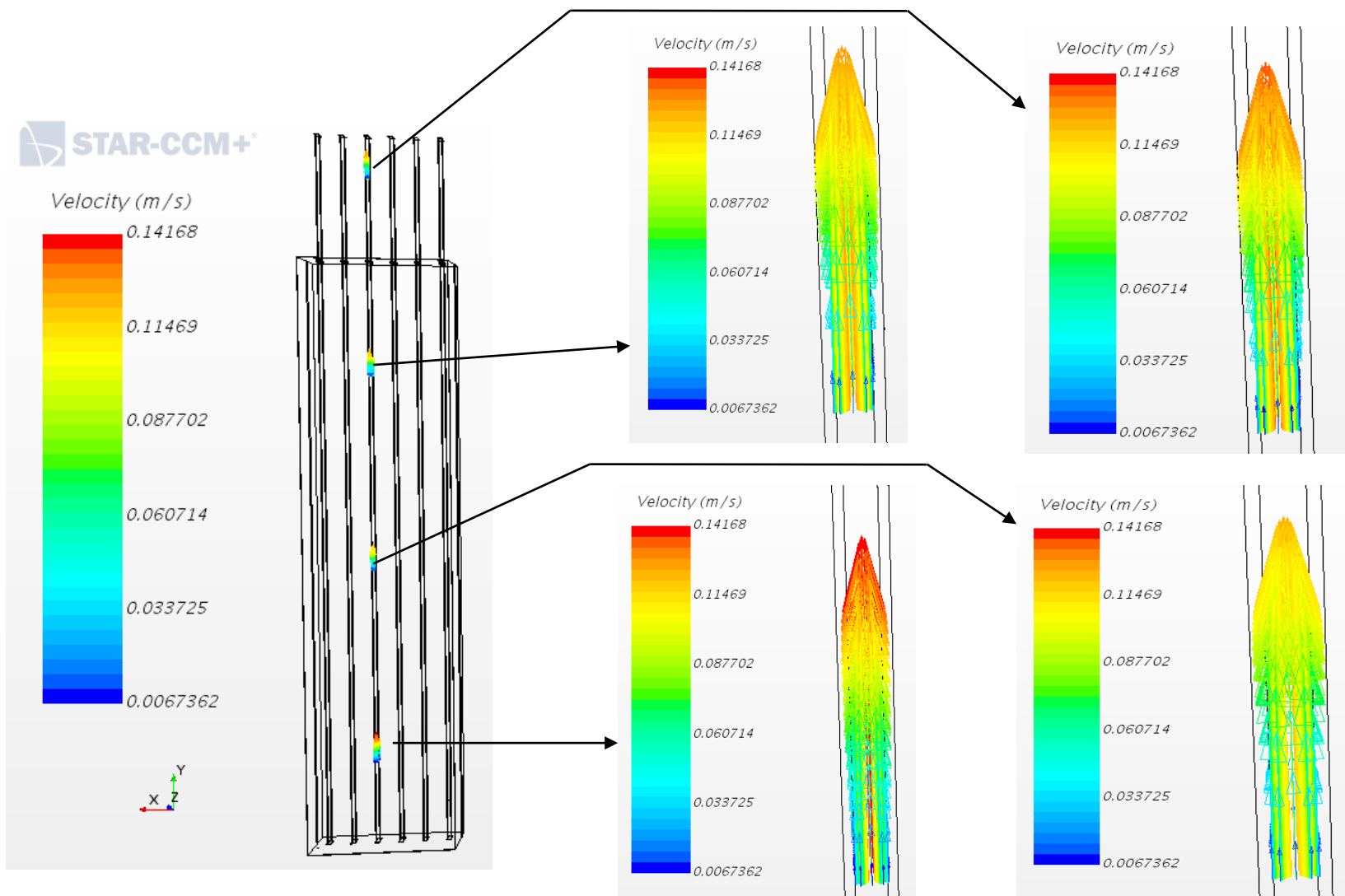


Figure 8.2-100. Velocity vectors inside Riser 3 at four different elevations.

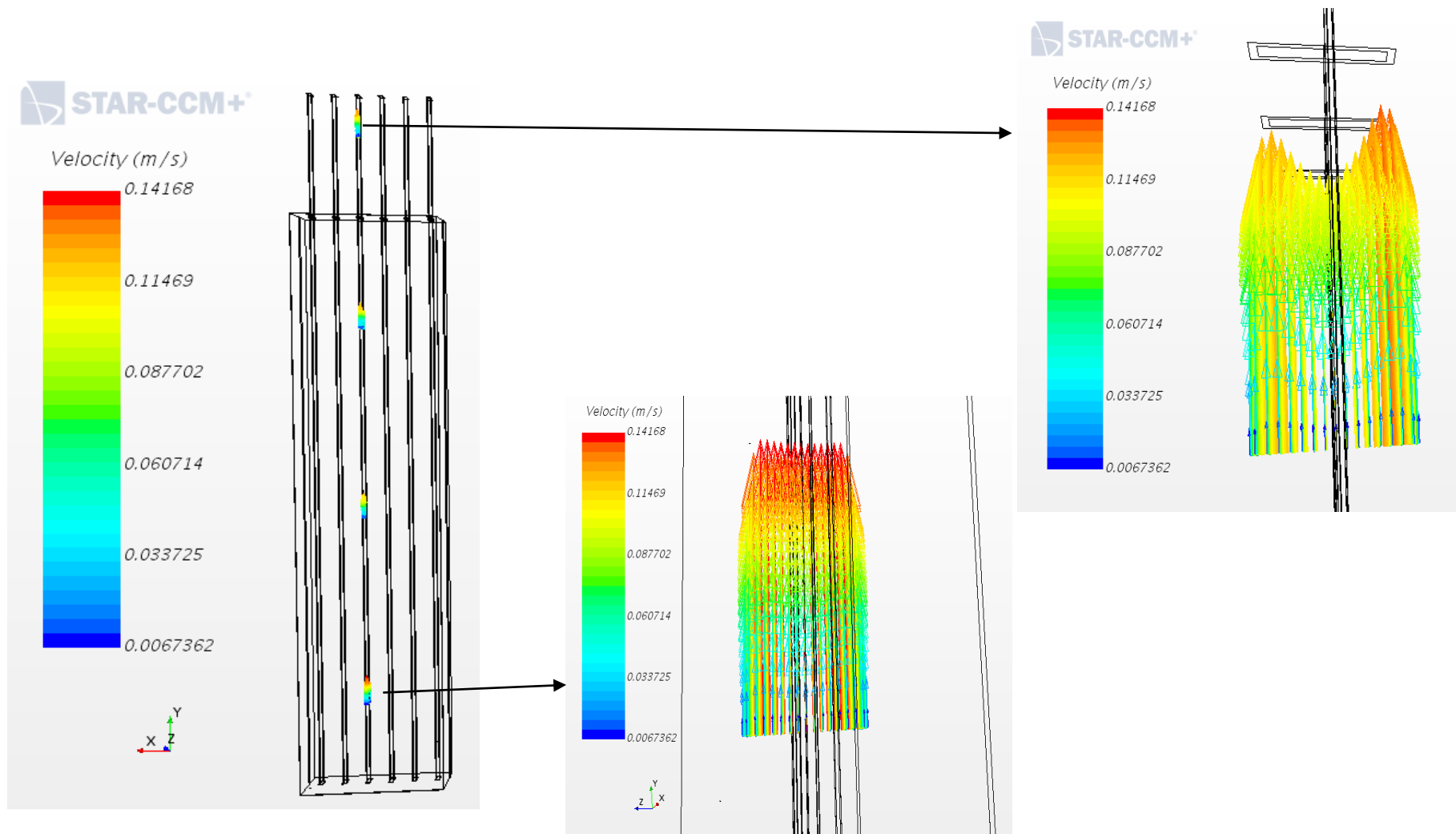


Figure 8.2-101. Velocity vectors in one of the middle risers (continued).

Figure 8.2-101 shows the velocity vectors inside Riser 3 as seen from the x-direction (i.e. as viewed on the y-z plane). The vectors at the lowest elevation showed a symmetric profile, while the profile at the highest elevation showed an asymmetric profile. The profile at the highest elevation was impacted most by the buoyancy effects on the front and back surfaces as evidenced by the locations of the velocity peaks.

8.2.7.4 Heat transfer coefficients

The convection heat transfer coefficients on the inner surfaces of the third riser tube are shown in Figure 8.2-102. The coefficients show an upward trend with an increase in elevation. The front side had the highest coefficients and the sides had the lowest coefficients. The lower temperature distributions on the sides (Figure 8.2-93) resulted in lower Grashof number which resulted in a lower Rayleigh number, Nusselt number and subsequently lower convection heat transfer coefficients. The peaks at the lower elevations (under ≈ 1 m) correspond with the temperature peaks found in the wall temperatures (that were shown in Figure 8.2-93).

Note that the Reynolds number was between 2900 and 3400.

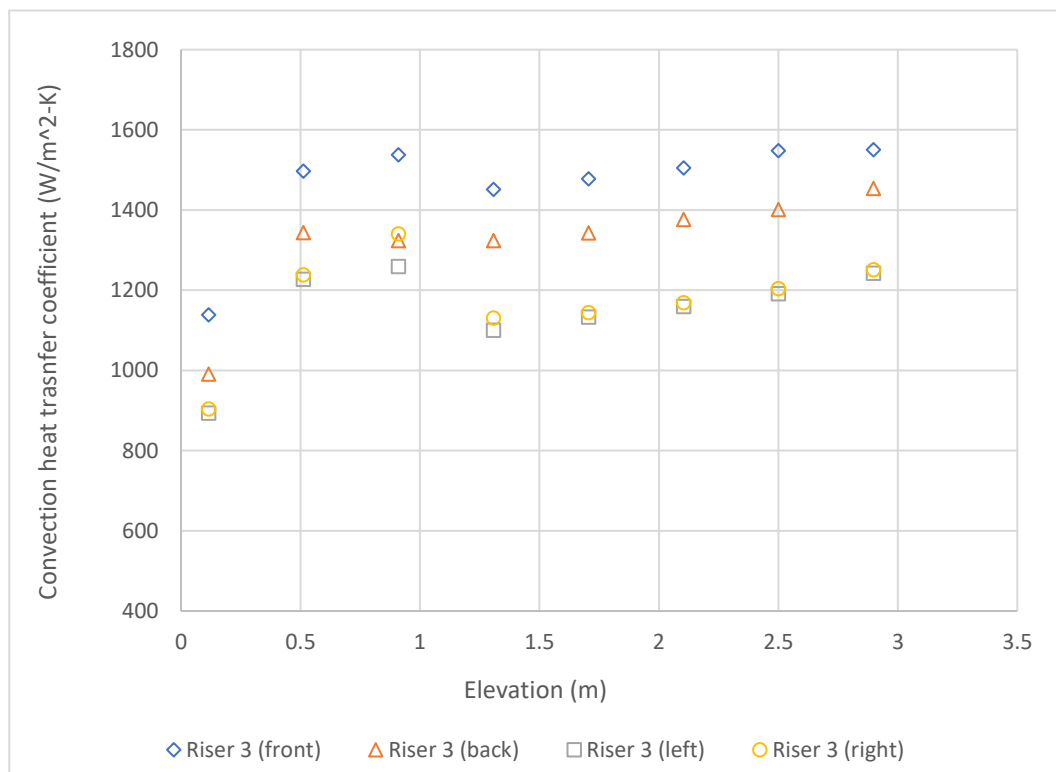


Figure 8.2-102. Convection heat transfer coefficient on the inner surfaces of Riser 3.

Figure 8.2-103 shows the Aicher number as a function of elevation on the inner surfaces of the third riser tube. The Aicher number was higher than 0.2 (except for three points corresponding to the sides and back for the bottom most elevation) for most of the elevation indicating fully developed free convection.

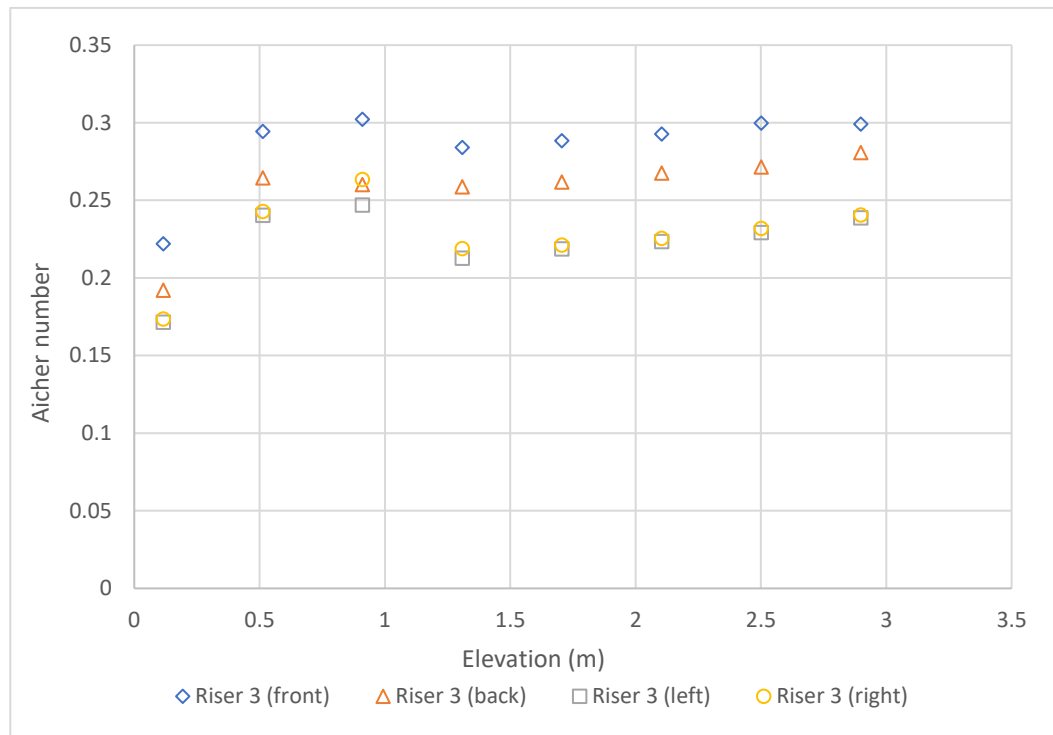


Figure 8.2-103. Aicher number as a function of elevation.

Figure 8.2-104 shows the Nusselt number ratio as a function of Aicher number. The data points form C shapes as with the finned rectangular tubes case. The data points corresponding to the lowest elevations have a noticeably lower Aicher number than the data points that follow and are excluded from the C shapes. The data points corresponding to the second lowest elevations had high Aicher numbers and Nusselt number ratios (top part of the C shape) and the general trend shows an initial decrease in the Aicher number, followed by increases in Aicher number for subsequent elevations. The overall trend of the Nusselt number ratios showed a decrease over the elevation, which is attributed to the increases in the forced Nusselt number component. Simultaneously, the free convection component Nusselt number showed initial decreases followed by increases onwards from the middle elevation of the heated section. The free convection component of the Nusselt number was in turn driven by the fluid/wall temperature difference which showed the same behaviour.

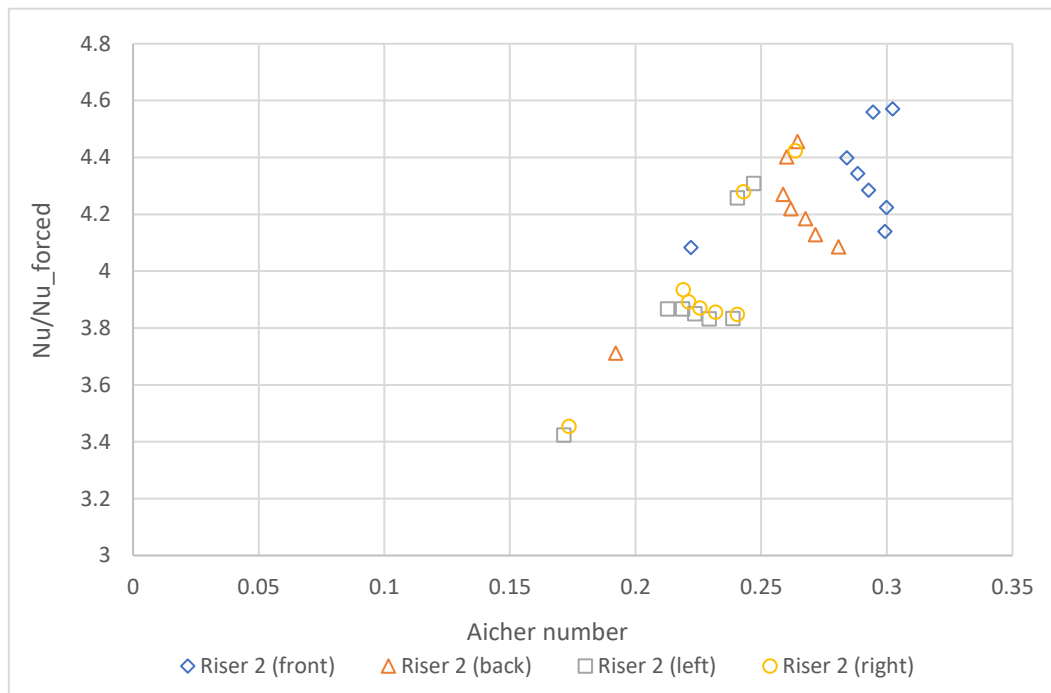


Figure 8.2-104. Nusselt number ratio as a function of Aicher number.

Figure 8.2-105 shows the convection heat transfer coefficients for the heater – and back walls. The coefficients show a downward trend, due to the rising air temperature with increasing elevation, which resulted in a smaller difference between the fluid and wall temperatures. This in turn resulted in lower Grashof number, Rayleigh number, Nusselt number and subsequently convection heat transfer coefficient. At an elevation of 3 m in the figure the back wall and cavity air temperature were almost the same, resulting in a lower value to be calculated for the convection heat transfer coefficient. Additionally, a recirculating cell existed at this location, also contributing to the coefficients to behave in this way.

Figure 8.2-106 shows the convection heat transfer coefficients calculated on the front and back sides of the riser tubes. The coefficients all show an increase in value with an increase of elevation. The omission of the fins resulted in a more equalized temperature in the front and back cavity as compared with the corresponding finned case, which resulted in the values of the coefficients for the heater and back walls being more similar than in the finned case (Figure 8.2-89).

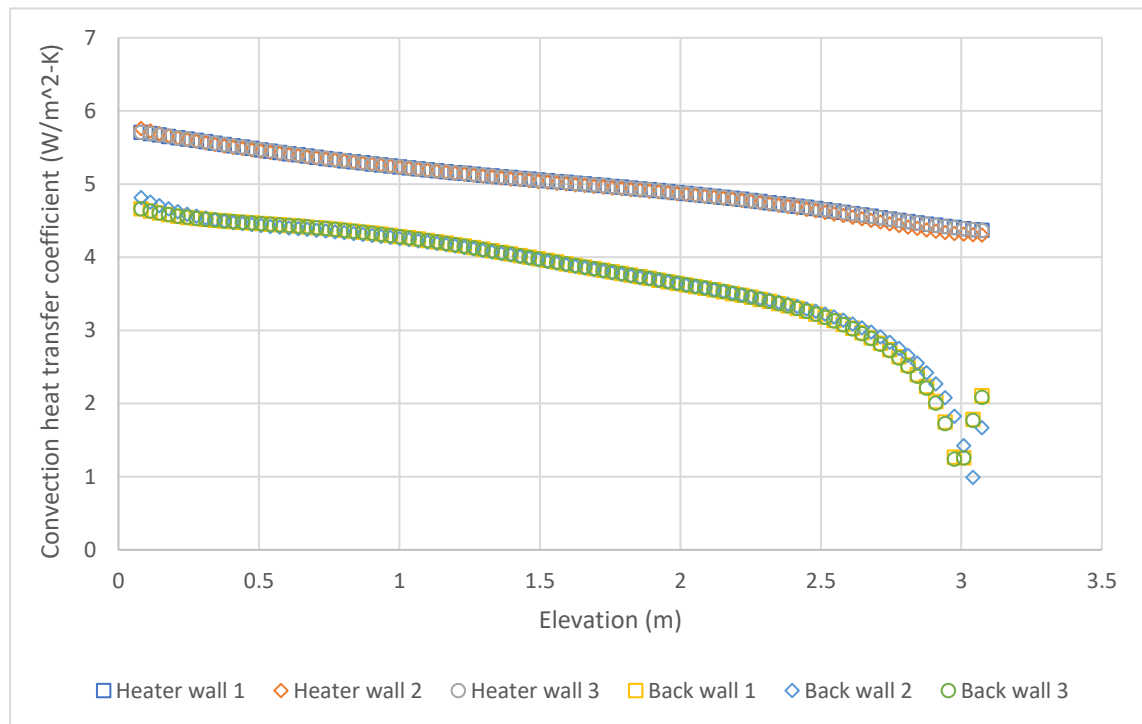


Figure 8.2-105. Heater – and back wall convection heat transfer coefficients.

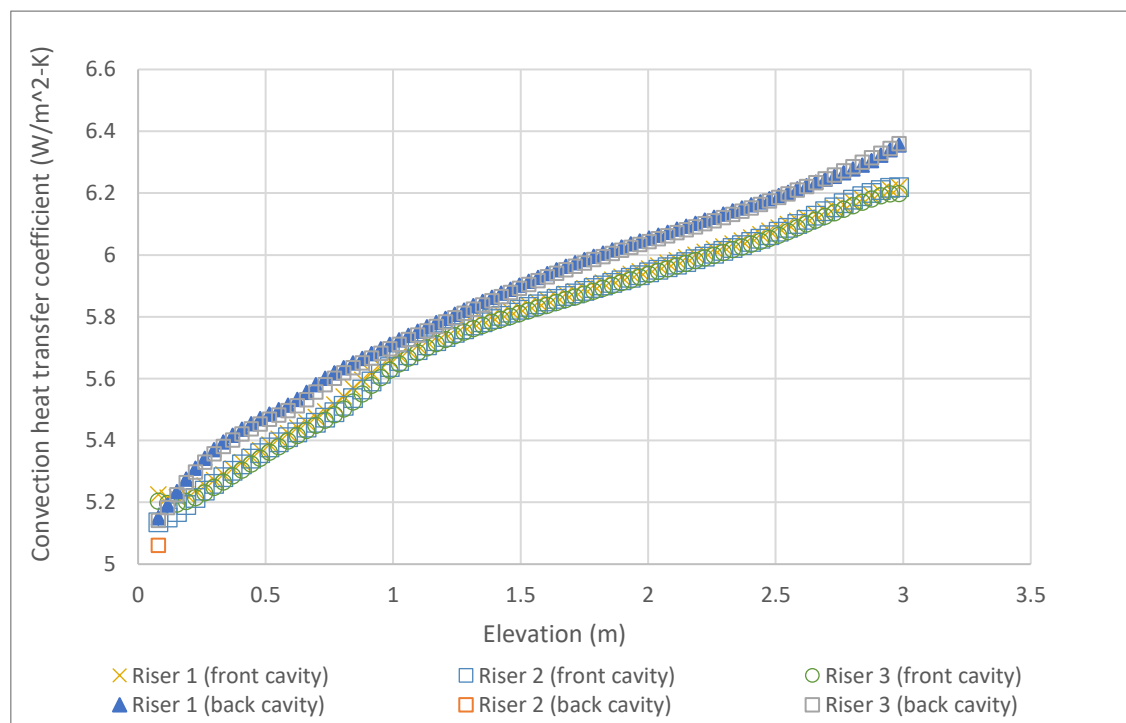


Figure 8.2-106. Convection heat transfer coefficient on the front and back sides of the riser tubes.

The

parameters and properties used to calculate the convection heat transfer coefficients, including the Grashof number, Reynolds number, Rayleigh number, Nusselt number and Prandtl number are shown in Appendix D.7.

8.2.7.5 Conclusion

The geometry of the rectangular riser tubes without fins was simulated with the same boundary conditions as the baseline UW RCCS facility. The number of tubes were doubled from the baseline design, but the total cross-sectional area was the same.

The heat fluxes on the important surfaces in the cavity was investigated, as well as the temperature distributions and velocity vectors. Velocity vectors were shown in the cavity as well as in the riser tubes. The vectors in the cavity showed the prevailing fluid flow patterns. A hotspot was observed at the lower elevations due to the hot fluid being accelerated downwards, which transported the heat towards the bottom of the heater box. The velocity vectors in the riser tubes showed the velocity profiles which could give insight on the prevailing flow regime.

The Aicher numbers indicated that the flow inside the riser tubes was primarily in the free convection regime.

The convection heat transfer coefficients were calculated for some of the surfaces in the cavity and for the risers' inside surfaces.

8.2.8 Summary on the 3D CFD results

8.2.8.1 Heater and back wall temperatures

The finned configurations shielded the back wall from direct radiation heat transfer, resulting in much lower temperatures than the cases without fins. In the circular fins case, the back-wall temperature was approximately 250 - 270 K lower than the heater wall temperature. In the corresponding case without fins the back wall temperature was 60 – 100 K lower than the heater wall (for reference see Figure 8.2-10 vs. Figure 8.2-28).

In the case with square tubes with fins the back wall temperatures were 250 - 280 K lower than the heater wall temperatures, while the corresponding case without fins resulted in temperatures that were 80 -100 K lower (for reference see Figure 8.2-43 vs. Figure 8.2-59).

In the case with rectangular tubes with fins the difference between the heater and back wall was between 245 - 310 K. In the corresponding case without fins the heater wall temperature was 110 K -140 K higher than the back wall (for reference see Figure 8.2-76 vs. Figure 8.2-95).

8.2.8.2 Riser tube wall temperatures

The cases without fins showed a more uniform temperature distribution around the walls of the riser tubes and in the cavity air temperatures. In the circular tubes without fins the temperature difference between the maximum and minimum temperature in the walls didn't exceed 6 K, while differences of 21-24 K were observed for the corresponding finned case (Figure 8.2-5 vs. Figure 8.2-25).

In the square tubes with fins case, the temperatures of the side walls were the highest and a maximum difference of 8 % in the wall temperatures around the circumference of the tubes. In the corresponding case without fins the front wall had the highest temperature and less than 1% difference between the wall temperatures around the circumference (for reference see Figure 8.2-40 vs. Figure 8.2-56).

In the case with rectangular tubes with fins the maximum difference between the riser tube walls was 14 K while in the corresponding case without fins the temperatures were within 8 K around the circumference of the riser tubes' walls (for reference see Figure 8.2-72 vs. Figure 8.2-93). The temperatures in the front and side walls showed a noticeable increase in the lower

elevations that formed peaks that were not observed for the other geometries. The reason was the decreased cross-sectional area between the rectangular riser tubes, which improved the cooling (and therefore downward buoyancy force) which was enhanced by a venturi effect that accelerated the heat downward towards the elevations where the peaks were observed.

8.2.8.3 Cavity air temperature

The cavity air temperature in the circular tubes finned case showed a difference of approximately 90 K to 210 K. In the corresponding case without fins, the temperatures were virtually the same until the top elevations, where the maximum temperature difference of approximately 30 K (for reference see Figure 8.2-9 vs. Figure 8.2-27).

In the case with square tubes without fins the temperature distributions were uniform, while in the corresponding finned case the difference between the front and back cavity air temperature ranged between 100 – 190 K (for reference see Figure 8.2-44 vs. Figure 8.2-58).

In the case with rectangular tubes with fins the air temperature in the front and back sides of the cavity differed by 80 – 200 K. The temperature distribution in the corresponding case without fins the maximum difference between the air temperature in the front and back cavity was 30 K (for reference see Figure 8.2-77 vs. Figure 8.2-96).

8.2.8.4 Radiation heat transfer

The radiation heat transfer in the finned circular tubes case constituted between 76 to 89 % of the heat transfer in the front cavity and between 52 – 61% in the back cavity. In the corresponding case without fins, the radiation heat transfer in the front cavity constituted 60% of the heat transfer and 54% in the back cavity (for reference see Table 8-1 vs. Table 8-2).

In the case with square tubes with fins the radiation heat transfer comprised 74 – 87 % of the heat transfer in the front cavity and 52 – 59 % in the back cavity. In the case with square tubes without fins the radiation comprised 76 – 77 % in the front cavity and 72 – 73% in the back cavity (for reference see Table 8-3 vs. Table 8-4).

In the case with rectangular tubes with fins the radiation heat transfer comprised 71 – 81 % in the front cavity and 50 – 52 % in the back cavity. In the corresponding case without fins the

radiation heat transfer in the front cavity comprised 73 % and 66 - 69 % in the back cavity (for reference see Table 8-5 vs. Table 8-6).

8.2.8.5 Convection heat transfer coefficients in the cavity

The convection heat transfer coefficients on the outer surfaces of the riser tubes for the case with circular tubes without fins were more uniform than the corresponding finned case in which the coefficients were 19-30% larger on the front cavity side than on the back cavity side (for reference see Figure 8.2-20 vs. Figure 8.2-36).

The convection heat transfer coefficients on the heater and back walls show a decrease with increasing elevation in both cases with and without fins (circular tubes). The coefficients on the back wall show the same minimum value in the top part of the cavity (for reference see Figure 8.2-21 vs. Figure 8.2-37).

For the square tubes with fins case the coefficients on the front side were 17 – 34 % higher than in the back cavity. In the corresponding case without fins (for reference see Figure 8.2-52 vs. Figure 8.2-67) the coefficients on the back cavity side were slightly higher (less than 1 %) than the coefficients in the front cavity.

The convection heat transfer coefficients for the heater- and back wall (square tubes cases) show an overall decrease across the elevation. The coefficients for the heater wall were higher than the back wall by values of between 1.5 – 4 W/m²-K for the finned case and 1 – 3 W/m²-K difference for the case without fins (for reference see Figure 8.2-51 vs. Figure 8.2-68).

In the case with rectangular tubes with fins the coefficients on the front side of the riser tubes were 2 – 3 W/m²-K larger than the corresponding coefficients on the back-cavity side. In the case with rectangular tubes without fins the coefficients were more uniform in the distribution across the elevation with differences of less than 1 W/m²-K (see Figure 8.2-89 vs. Figure 8.2-106).

The coefficients for the heater and back walls for the rectangular tubes with fins differ by values of 1.5 – 3.3 W/m²-K. In the corresponding case without fins, the difference was between 1 W/m²-K and 3.5 W/m²-K. In both cases the coefficients of the heater wall were higher and the coefficients associated with both walls showed an overall decrease across the elevation (for reference see Figure 8.2-90 vs. Figure 8.2-105).

8.2.8.6 Convection heat transfer coefficients in the riser tubes

The convection heat transfer coefficients for the inside surfaces of the circular tubes with fins case showed an increase across the elevation for the front and sides while showing a decrease for the back surface across the elevation. The corresponding case without fins in contrast showed an increase for all surfaces. The coefficients in the latter case had almost the same value (for reference see Figure 8.2-17 vs. Figure 8.2-33).

For the circular tubes the convection regime indicated by the Aicher number was predominantly mixed convection. The side walls however in the finned case transitioned to the free convection regime (for reference see Figure 8.2-16 vs. Figure 8.2-34).

The convection heat transfer coefficients for the square tubes with fins showed an overall increase across the elevation for the front and sides, while an overall decrease was observed for the back side. In the corresponding case without fins the coefficients all showed an increase across the elevation (for reference see Figure 8.2-49 vs. Figure 8.2-64). The regime was predominantly mixed convection with the left - and right-side transitioning to the free convection regime. In the corresponding case without fins the regime remained as mixed convection for the full elevation (for reference see Figure 8.2-50 vs. Figure 8.2-65).

For the case with rectangular tubes with fins the coefficients showed only slight changes after the lowest elevation data point on the front and sides. The back side however showed an overall decrease across the elevation. In the corresponding case without fins the coefficients show peaks below 1 m elevation corresponding to the temperature peaks in these walls at this elevation. Above the peaks the coefficients show a steady increase over the rest of the heated section (for reference see Figure 8.2-85 vs. Figure 8.2-102).

In the case with the rectangular tubes (with or without fins) the values of the convection heat transfer coefficients were higher than the corresponding square or circular tubes cases. The smaller cross-sectional area of the rectangular tubes was heated more effectively and largely avoided the mixed convection regime that was prevalent in the other geometries.

For the rectangular tubes cases the Aicher numbers were generally higher, meaning better transition from mixed convection to free convection for the front and sides of the riser tubes. The back side of the finned case however remained within the mixed convection regime since the back surface in the finned cases experienced less heating than the corresponding case without fins.

The Reynolds numbers in the riser tubes were in excess of 6000 for the circular and square geometry and was between 2900 and 3400 for the rectangular tubes. The forced convection component is therefore in the transitional zone between laminar and turbulent flow in the rectangular tubes.

8.2.9 Concluding remarks

The UW RCCS facility was simulated by using different geometries for the riser tubes and was simulated for configurations with and without fins. The boundary conditions were kept constant between the different cases.

The results extracted were the heat fluxes on the important surfaces, temperature distributions at various locations and velocity vectors on section planes in the heater box. The heat fluxes were tabulated and show the division of the heat transfer on the various surfaces of the fins and tubes. The total heat fluxes and radiative heat fluxes were shown as transferred from the front cavity and from the back cavity onto the surfaces of the fins and tubes. The temperature distributions were taken within the heater wall, back wall, within the fin walls and within the riser tube walls. Additionally, temperatures were taken on line probes in the front and back cavity and in the middle of the riser tubes.

Convection heat transfer coefficients were calculated based on the extracted results by using the theory described in section 3.3. The coefficients were calculated for the water inside the riser tubes, as well as for the air inside the cavity.

The Aicher numbers were calculated for the inside surfaces of the riser tubes and were meant to indicate the convection regime. The mixed convection regime was found in many cases, but transition to free convection was a common occurrence. The back walls were especially susceptible to the mixed convection regime, because of the lower degree of heating compared with the other sides.

The configurations without the fins resulted in the air temperature in the front and back part of the cavity to be almost the same. The temperatures on the back surfaces of the riser tubes were also higher in these configurations, leading to higher convection heat transfer coefficients on these surfaces compared with the finned configurations.

Thermal radiation was the dominating mechanism of heat transfer (50%-90% of the total heat transfer) but convection heat transfer also had non-negligible effect on the heat transfer.

8.3 Results for the coupled cases

The coupled simulations included the entire loop of the UW RCCS facility. The heater box required 3D CFD methods to solve the complex multi-directional phenomena, while the pipe network before and after the heater box could be simulated by using the 1D methodology. The coupling methodology combined the 1D and 3D methodologies in such a way to use the strengths of each code. The coupling also has the advantage of having a feedback loop to improve the accuracy of the assumed boundary conditions.

The coupled simulation results were extracted in a similar way to the 3D results. Line probes were placed at locations within the riser tube walls and at the middle of the riser tubes. The rest of the cavity was however not examined in the detail as in the 3D simulations, as the loop was the most important part to evaluate the feasibility of the coupling between the codes. It was assumed if the temperature distributions within the walls were the same as the 3D codes, that the rest of the cavity's results would also show the same degree of agreement. Two variations are discussed. A third was also attempted, namely Variation A, where Fluent was used to model the manifolds and a single set of input and output data was exchanged between the codes. The variation proved very difficult to obtain a converged solution and an appropriate solution could not be found.

8.3.1 Manifolds modelled by using Flownex (Variation B)

Variation B used 1D methodology to model the flow distribution in the manifolds that split and recombine the flow before and after the heater box. The methodology was described in section 7.3 and the coupling was done in the following manner: the inlet mass flow rates, inlet temperatures and outlet pressures of the three riser tubes were transferred from Flownex to Fluent. In Fluent, the flux on the heater wall was specified and the transferred boundary conditions were used to solve for an outlet temperature. The outlet temperature was transferred from Fluent to Flownex where the loop was recalculated to correct the boundary conditions and the process was repeated. The Flownex model of this variation was shown in Figure 7.3-1.

The mesh quality of this configuration is reported in Appendix B.14 and the solution monitoring is reported in Appendix C.16. The simulation was regarded as being converged after approximately 14000 iterations.

The results from the case are summarized. Selected results are shown in Table 8-7 and include the average heat flux, heat energy added to each riser, outlet temperature, mass flow rate and

calculated flow admittance in each riser. The riser tube in the middle of the cavity, Riser 2, received the most heat energy and therefore had the highest outlet temperature. The flow resistance (inverse of the flow admittance) was the lowest in Riser 1 (closest to the inlet of the manifold), which caused a higher mass flow rate to be obtained than in the other risers. The flow resistance increased towards Riser 3's inlet, which caused the lowest mass flow rate in Riser 3. The mass flow rate of Riser 2 was 1.3% larger than Riser 1 which in turn had a 2% larger mass flow rate than Riser 3.

The Reynolds number for the flow entering the heated section are 6177 for Riser 2, 6095 for Riser 1 and 5985 for Riser 3. In other words, fully turbulent flow. Through the heating effect the Reynolds numbers increase further to 6645, 6741 and 6540 for Riser 1, 2 and 3 respectively.

The water temperature in the middle of the risers are plotted as a function of elevation in Figure 8.3-1. The positions indicated by A, B and C in the figure correspond to the bottom elevation of the heated section, outlet of the heated section and top of the fin assembly respectively. The temperature rises almost linearly with elevation within the heated section. The water temperature was measured on a probe in the middle of the riser tube, which meant that the temperature rise at the lower elevation didn't occur immediately at point A. Similarly, small amounts of heating occurred above points B and C until the temperatures had equalised across the cross-section of the riser tubes.

Table 8-7. Selected results for Variation B

	Average heat flux (W/m ²)	Heat energy (W)	Mass averaged outlet temperature (K)	mass flow rate (kg/s)	Flow admittance (m ² -s)
Riser 1	6064.79	3514.61	307.494	0.1938	7.32E-08
Riser 2	6216.39	3602.47	307.546	0.1963	7.53E-08
Riser 3	6073.39	3519.6	307.607	0.190	7.06E-08
Total	-	10 636.68	-	0.58	

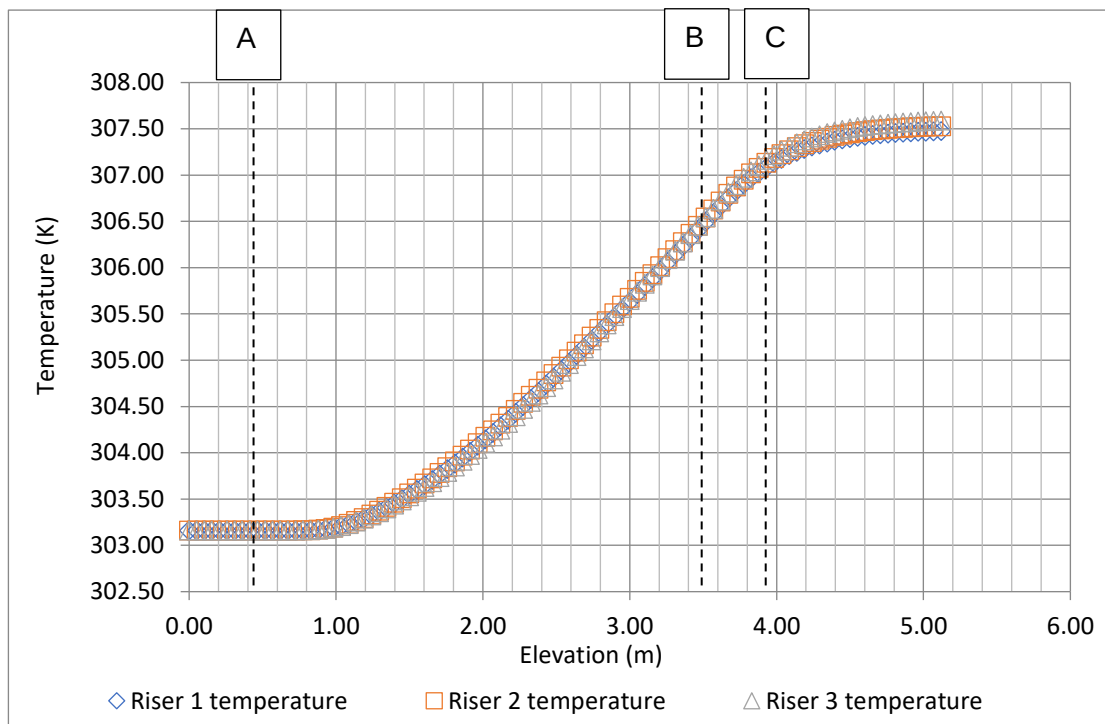


Figure 8.3-1. Water temperature distribution in the middle of the riser tubes.

Figure 8.3-2, Figure 8.3-3 and Figure 8.3-4 show the wall temperatures in the three riser tubes. The origin was moved to start at the bottom most elevation of the heated section. In other words, the graphs start at an elevation corresponding to point A and ends at the elevation corresponding to point C in Figure 8.3-1.

The left- and right-side walls of Riser 2 have almost the same temperature distribution (as seen in Figure 8.3-3). The left- and right-side walls of the other risers however have a higher and lower temperature, depending on the wall's position relative to the middle of the heater box. Therefore, the right-side of Riser 1 and the left-side of Riser 3 (see Figure 8.3-2 and Figure 8.3-4) have a higher temperature, since the heat was concentrated in the middle of the heater box.

The front-side walls have the second lowest temperature and show a sharp increase in temperature at the topmost elevation in the heater box (≈ 3 m) due to a hotspot that occurs at this location.

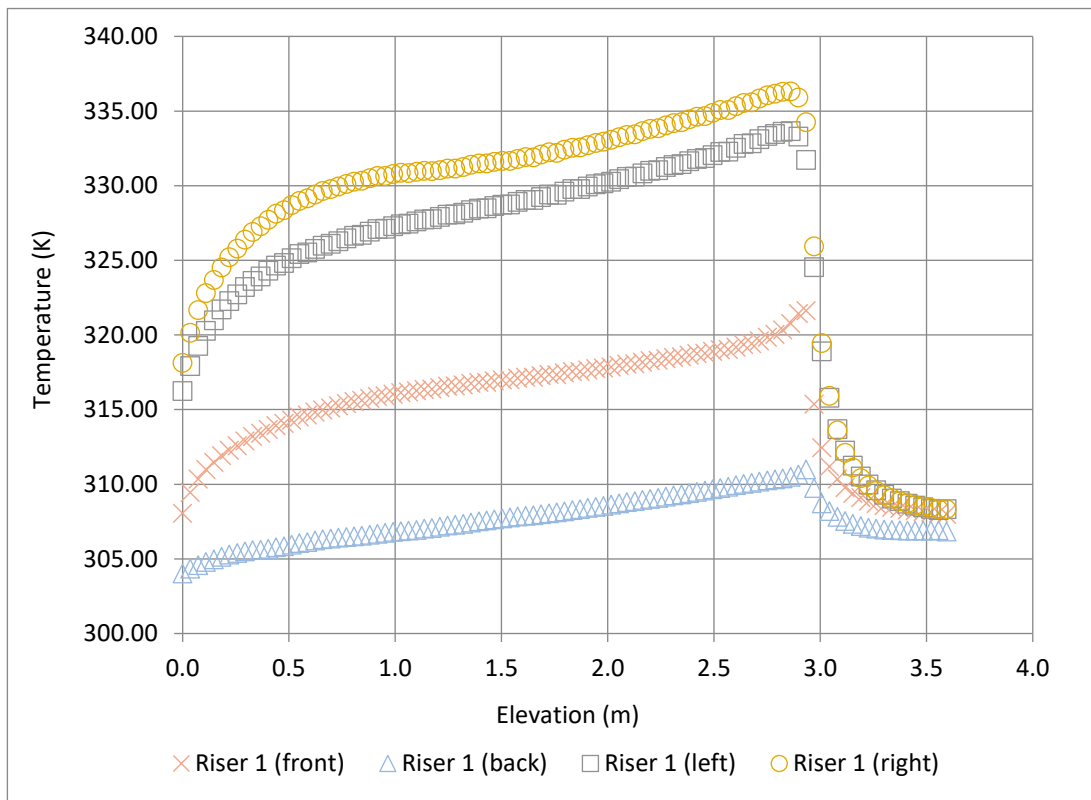


Figure 8.3-2. Riser wall temperature distributions for Riser 1.

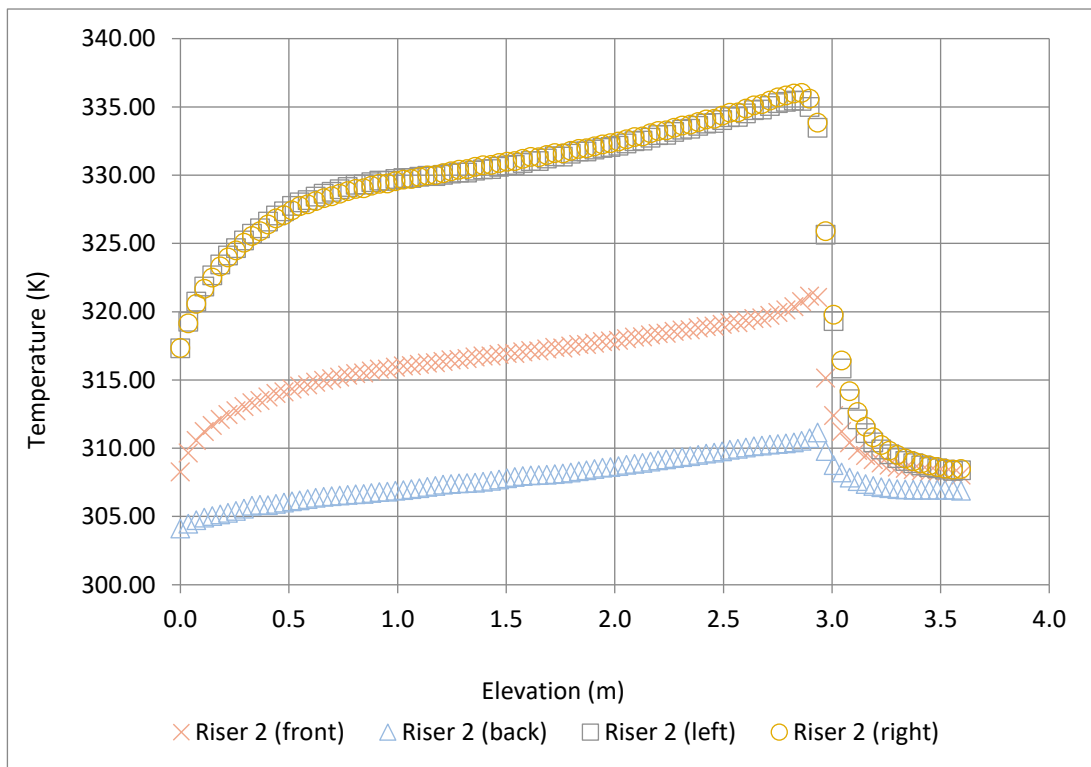


Figure 8.3-3. Riser wall temperature distributions for Riser 2.

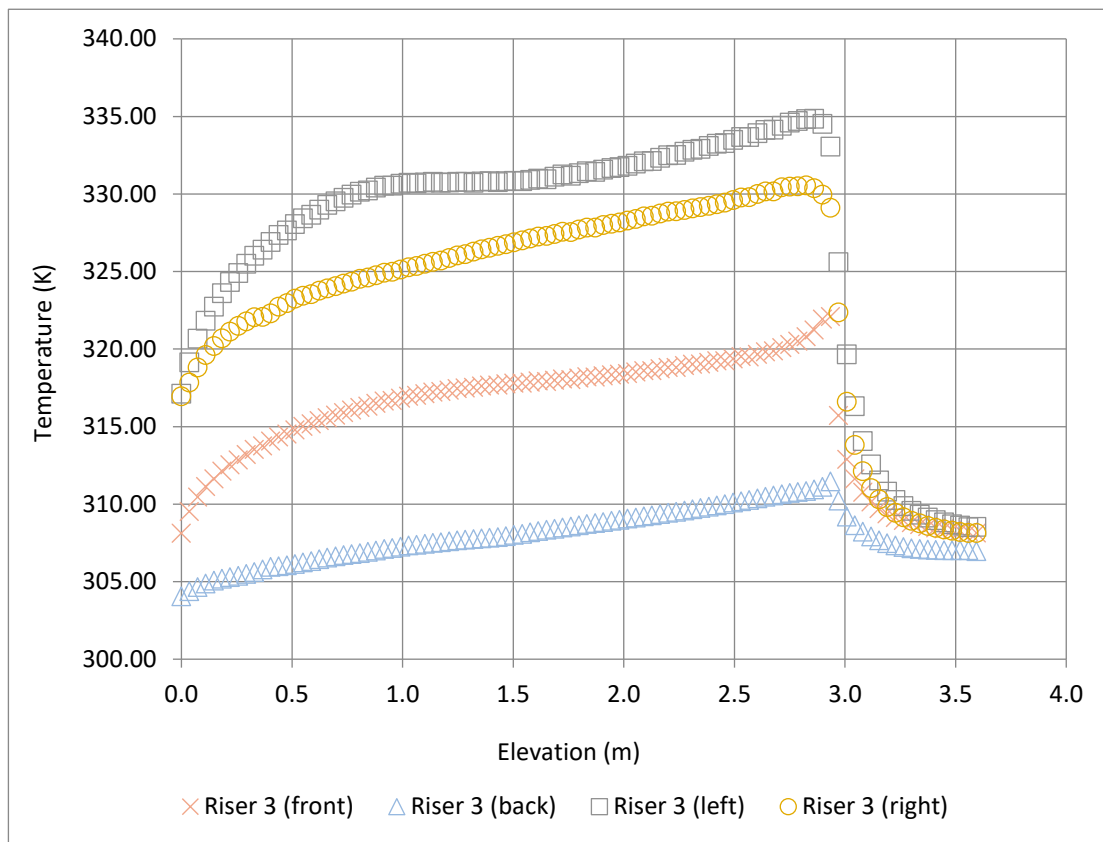


Figure 8.3-4. Riser wall temperature distributions for Riser 3.

The convection heat transfer coefficients were calculated for the four inside surfaces in the riser tubes. The coefficients for the middle riser tube (Riser 2) are shown in Figure 8.3-5. The coefficients were calculated within the heated section of the riser tubes, i.e. the distance between point A and B in Figure 8.3-1. The coefficients on the front and side walls increase as a function of elevation, whereas the corresponding coefficients decrease as a function of elevation on the back wall. Figure 8.3-6 shows the Aicher number as a function of elevation, which is necessary to describe the behaviour of the graph in Figure 8.3-5.

The Aicher number was calculated to use as a guideline to determine the regime of flow (see Figure 8.3-6). Aicher numbers between 0.05 and 0.2 indicate mixed convection, smaller than 0.05 indicates forced convection and larger than 0.2 indicates free convection heat transfer. The discontinuity in the coefficients of the left side wall (in Figure 8.3-5) occurred where a transition occurred between mixed and free convection. The front and back walls fell well within the mixed convection regime for the entire heated section.

The Grashof number for the back surface was significantly smaller than the corresponding number for the front surface, owing to the lower temperature difference between the fluid and wall temperatures on the back wall. This led to a lower free convection Nusselt number for the back surface. The mixed convection Nusselt number was calculated as a function of the free and forced Nusselt numbers by using Equation (16), repeated here for convenience:

$$Nu_{mixed} = \left(|Nu_{forced}^3 - Nu_{free}^3| \right)^{\frac{1}{3}} \quad (16)$$

In both cases the free and forced components of the Nusselt number increases with elevation. In the case of the back wall, the free Nusselt number approaches the forced Nusselt number, causing a decrease in the mixed Nusselt number. The convection coefficient was calculated as a function of the conductivity, mixed Nusselt number and hydraulic diameter. Although the conductivity increased, the Nusselt number decrease dominated the equation, resulting in a decrease of the convection coefficient for the back wall.

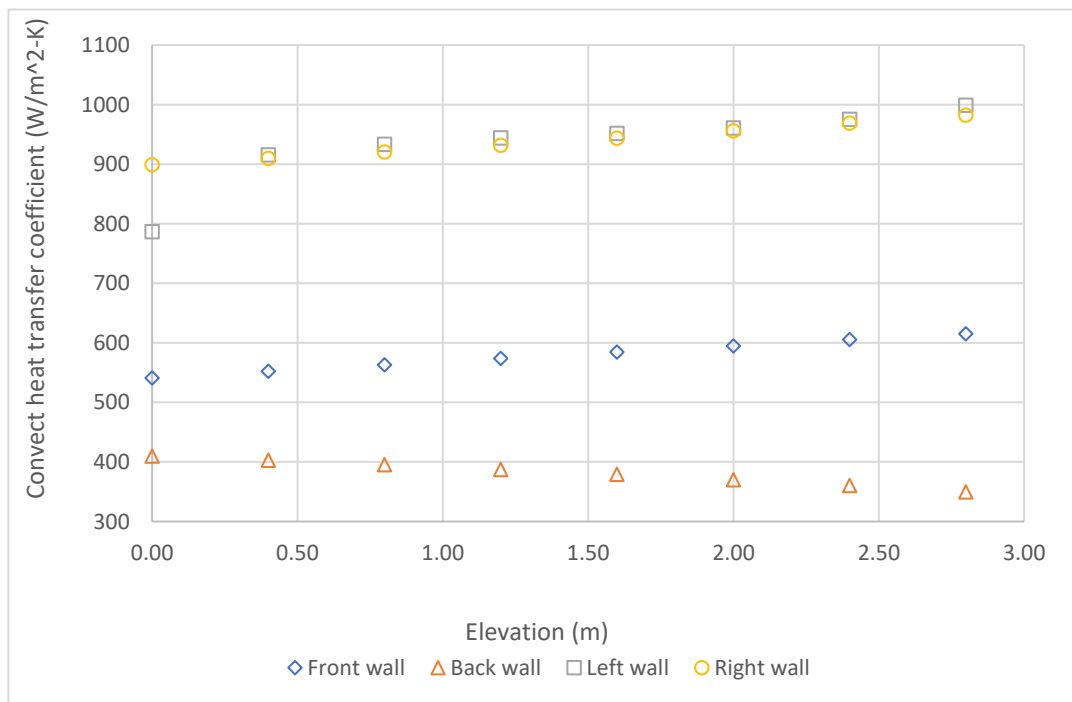


Figure 8.3-5. Convection heat transfer coefficient on the respective surfaces as a function of elevation for Riser 2.

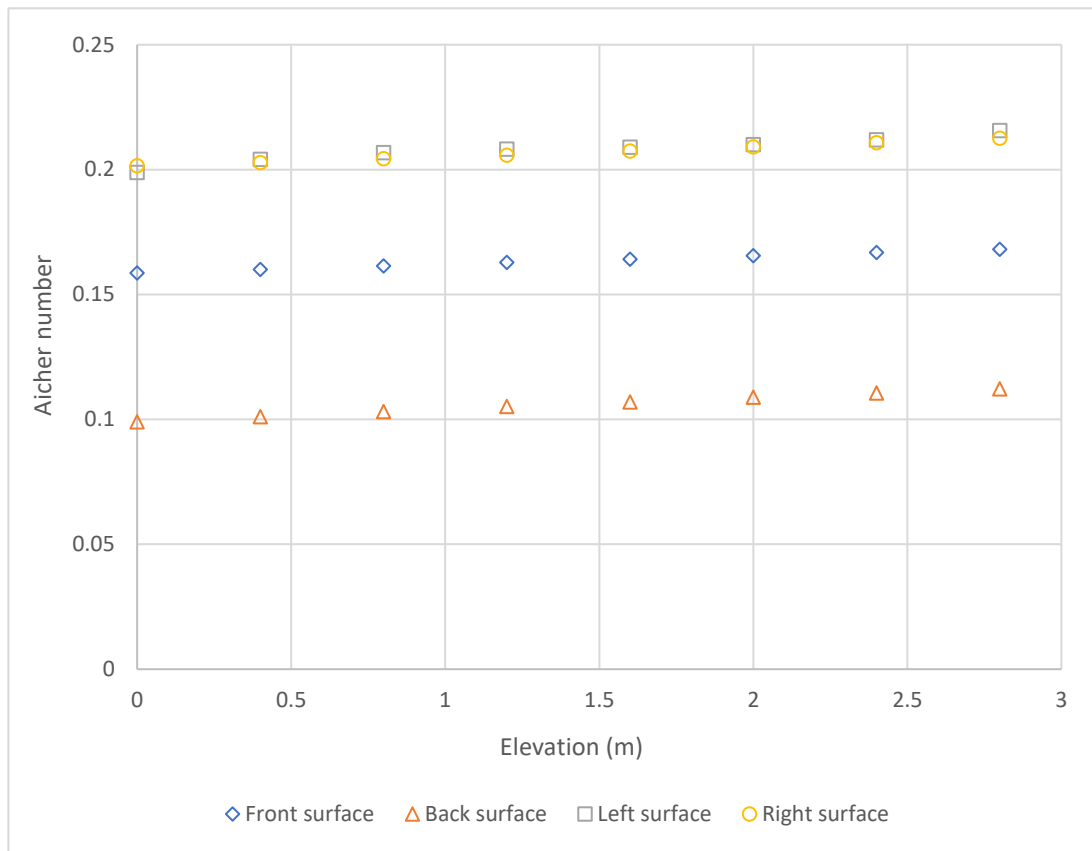


Figure 8.3-6. Aicher number as a function of elevation for the inner surfaces of the middle riser tube.

In conclusion, the 1D/3D coupled case Variation B consisted of the heated section simulated by using Fluent and the connected pipe network and heat sink by using Flownex. The mass flow rate was fixed by Flownex and the flow distributions in the manifolds were calculated. the subsequent mass flow rates, inlet temperatures and outlet pressures were transferred to Fluent, which transferred an outlet temperature.

The temperature distributions in the riser tube walls and temperature distributions in the riser tube water were extracted. Convection heat transfer coefficients were calculated based on the extracted temperatures. The mass flow distribution, heat flux per riser and admittance was also discussed.

8.3.2 Natural circulation coupled case (Variation C)

The results for the natural circulation case (which was described in section 7.4) are discussed in this section. This variation (of which the Flownex model is shown in Figure 7.4-1) differs from the Variation B model in that the mass flow rate was not fixed on the Flownex side, allowing the mass flow rate to numerically develop by the iterative exchange of data between the coupled codes. In addition, instead of the outlet temperature that was transferred from the 3D code to the 1D code as in the case of Variation B, the heat added per riser was transferred.

The mesh properties of the model are discussed in detail in Appendix B.15. The solution monitoring is shown in Appendix C.17 in more detail. The simulation was considered to be converged after approximately 11000 iterations.

Table 8-8 shows selected results pertaining to the riser tubes. The total heat added to the risers was 10 633.8 W, which corresponds well with the heat added at the heater wall (10 646 W). The relative error between energy in and energy out was therefore approximately 0.001. The total mass flow rate calculated by the solver was 0.5804 kg/s, which corresponds well with the mass flow rate obtained during the experiments by Lisowski (2013), which was 0.58 kg/s (relative error < 0.001). Note the exit temperature measured during experiments for this case was 307.42 K (Lisowski, 2013). The exit temperatures are also listed in Table 8-8. Lower limits were imposed on the heat that would be transferred from Fluent to Flownex. This meant that during the initial stages the mass flow rate was lower and gradually increased to the values shown in Table 8-8.

The Reynolds numbers of the flow in the heated section was 6400 for Riser 2, 6345 for Riser 1 and 6346 for Riser 3. In other words, fully turbulent flow. The Reynolds numbers were in excess of 6000 when the flow entered the heated section.

The bulk fluid temperature was taken on a line probe in the middle of the riser tubes. The resulting temperature distribution as a function of the position is shown in Figure 8.3-7. The temperature increase in the water at the probe location only started at 1 m, despite the heated

length starting at 0.32 m. The heating at the probe location also continued past the heated length, which ended at the approximately 3.3 m elevation.

Table 8-8. Selected results for Variation C from the 3D model.

	Average heat flux (W/m ²)	Heat energy (W)	Mass averaged outlet temperature (K)	Mass flow rate (kg/s)
Riser 1	6066.92	3515.85	307.52	0.19297
Riser 2	6214.5	3601.37	307.60	0.19446
Riser 3	6068.19	3516.58	307.52	0.19299
Total		10633.8		0.5804

The temperature distribution (in Figure 8.3-7) was measured at the centreline of the riser tubes, which showed a delayed reaction to the development of the cross-sectional temperature distribution. Thus, the delay in heating near the inlet and the apparent continuation of heating above the heated section where the walls are adiabatic.

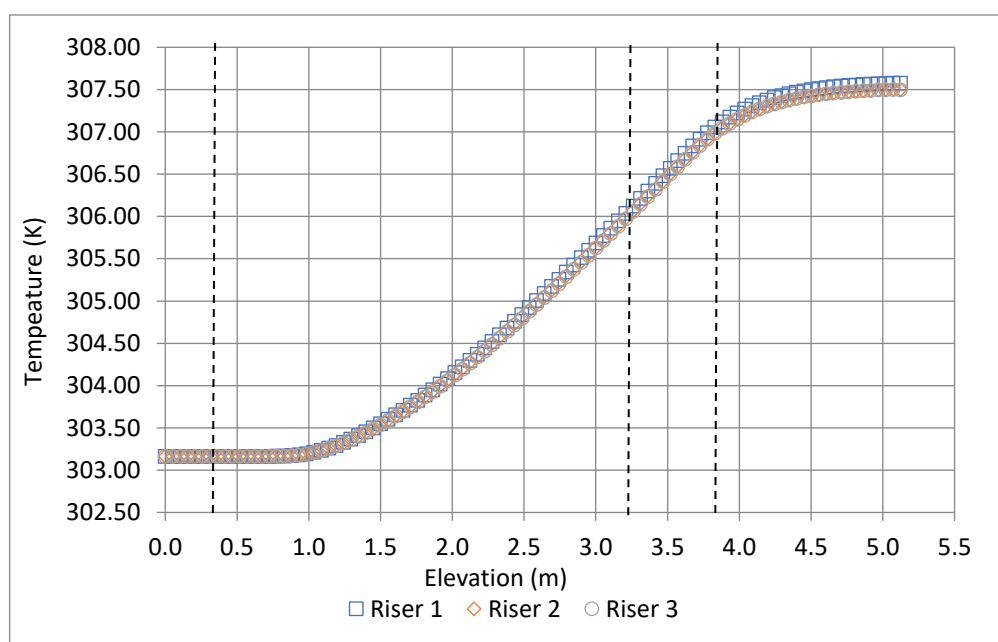


Figure 8.3-7. Water temperature distribution in the riser tubes for Variation C.

The wall temperatures were taken by probes located on the left, right, front and back surfaces of the riser tubes. The resulting temperature distributions for Riser 1, 2 and 3 are shown in Figure 8.3-8, Figure 8.3-9 and Figure 8.3-10 respectively. The left and right-side walls had similar temperature distributions (maximum difference of 0.9%) and the back wall had the lowest temperature (3% lower than the front wall). The side walls had the maximum temperature as they were heated by the fins.

Note that the origins in the figures were moved so that the 0 m elevation points coincide with the 0.33 m elevation point in Figure 8.3-7. The noticeable drop in temperature above 3 m was due to the riser tubes exiting the heated section. In the initial elevations, the front and side walls show a rapid increase in temperature with elevation, as in Variation B (see Figure 8.3-2, Figure 8.3-3 and Figure 8.3-4). This was because of the downward flow of the heated air next to the fins and the riser tubes' front walls. The bottom of the cavity was however adiabatic and along with the upward circulation of the heater wall transported the heat away from the bottom parts of the walls causing the cold spots at the lowest elevations shown in Figure 8.3-8 through Figure 8.3-10.

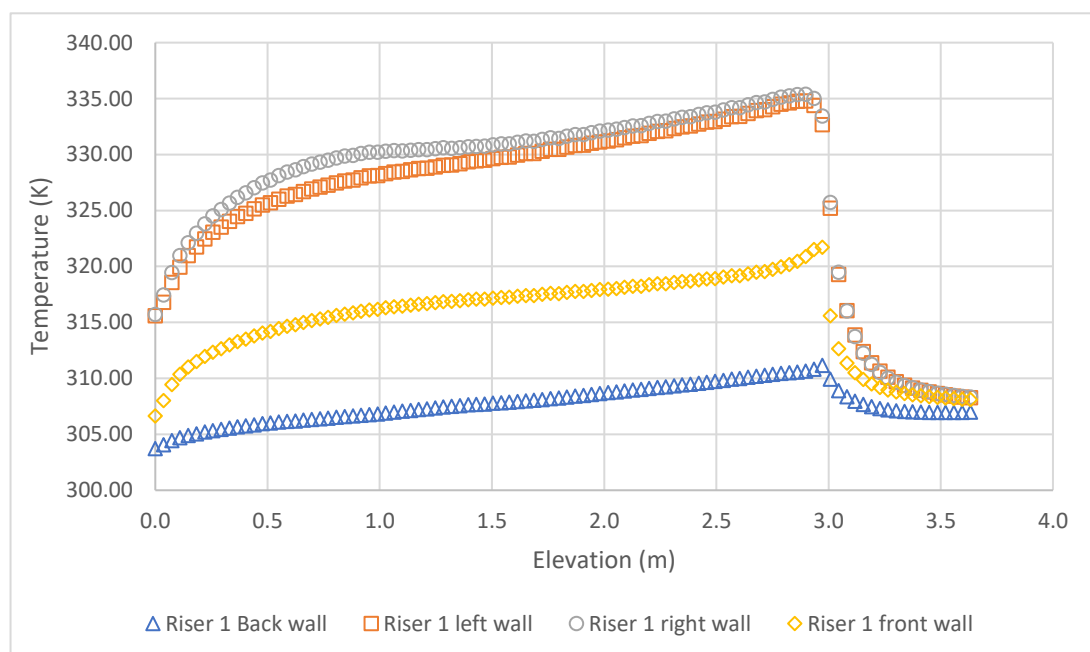


Figure 8.3-8. Wall temperatures in the first riser tube.

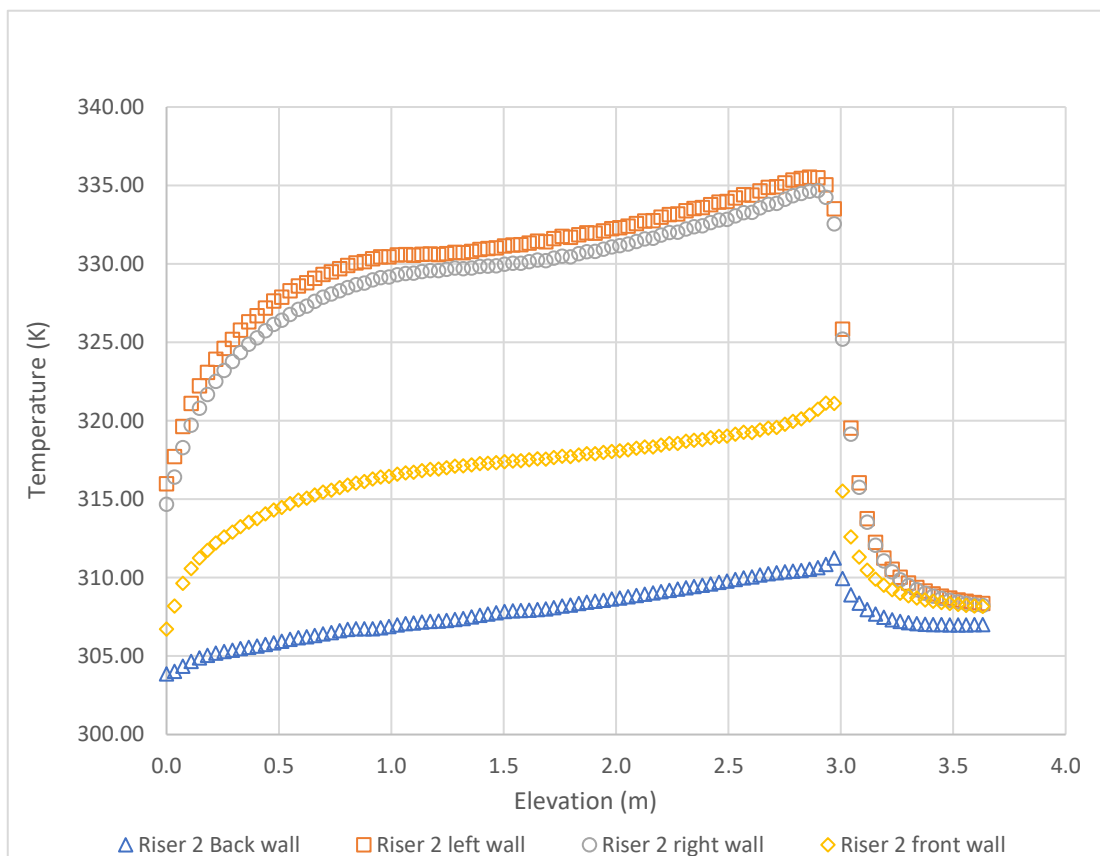


Figure 8.3-9. Wall temperatures in the second riser tubes.

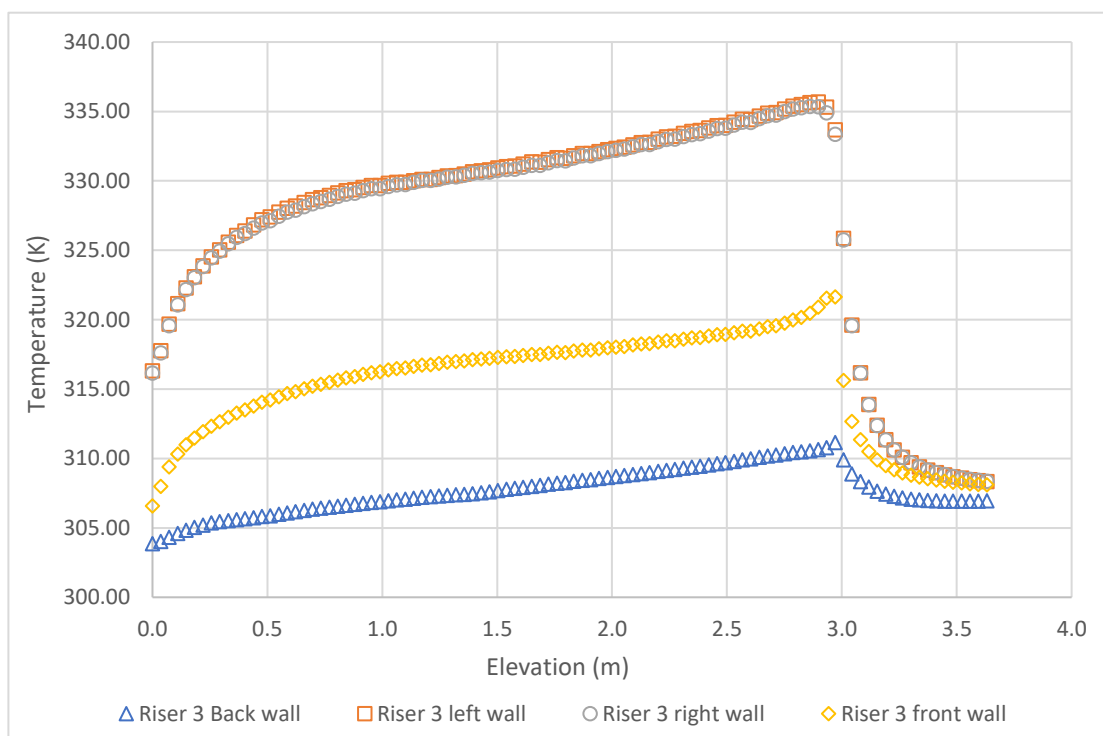


Figure 8.3-10. Wall temperatures in the third riser tube.

The convection heat transfer coefficients for the inner surfaces of the middle riser are shown as a function of elevation in Figure 8.3-11. The coefficients show an overall increase for the front and sides of the tube, with a decrease for the back side. The temperature increase on the former surfaces causes an improvement of the convection regime while for the latter the regime degrades further. Figure 8.3-12 shows the Aicher number as a function of elevation, which shows the back wall moving deeper into the mixed convection regime ($0.05 < \text{Aicher} < 0.2$) with an increase in elevation, while the sides of the riser tube transitions into the free convection regime. The lowest elevation points in Figure 8.3-11 and Figure 8.3-12 represent the lowest elevation that was unheated, therefore was much lower than the subsequent elevations.

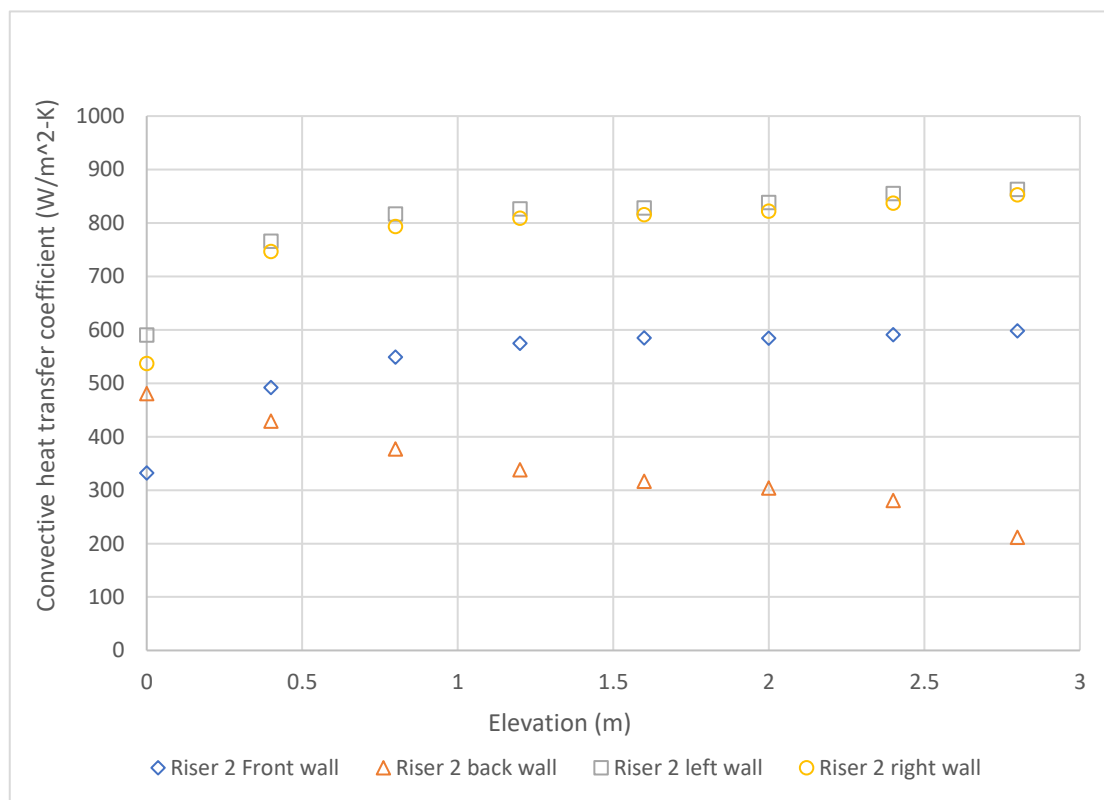


Figure 8.3-11. Convection heat transfer coefficient as a function of elevation.

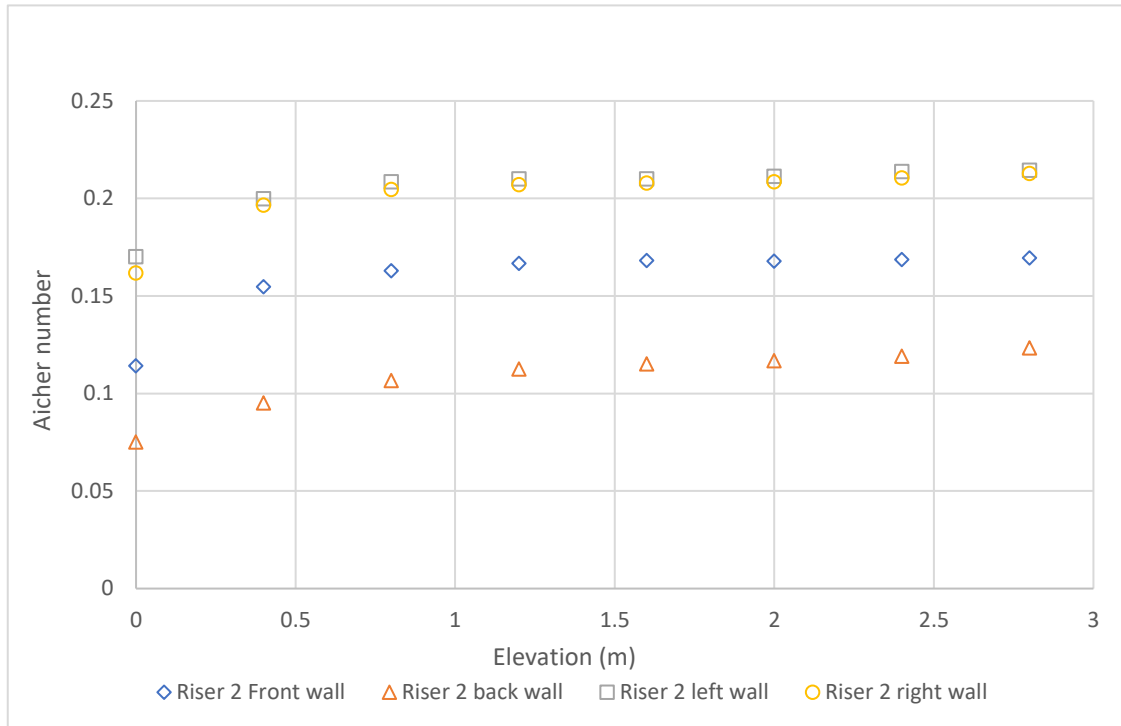


Figure 8.3-12. Aicher number as a function of elevation for Variation C.

In conclusion the Variation C model differed from the Variation B model in that the mass flow rate developed numerically in a natural way. A heat flux was transferred from Fluent to Flownex, rather than the exit temperature transferred in the case of Variation B.

The wall temperature distributions and the water temperature distributions were extracted as results and were subsequently used to calculate the convection heat transfer coefficients on the inner surfaces of the riser tubes. Additionally, the division of the mass flow rate and heat flux between the riser tubes have also been shown.

The Variation C model seemed to converge more efficiently than the Variation B model, possibly due to the heat flux being transferred.

8.3.3 Summary of the 1D/3D coupled results

The results of the 1D/3D coupled cases have been discussed. The results focussed on the inside surfaces of the riser tubes and excluded the results of the surfaces in the heated cavity, since the latter was discussed in detail in the 3D cases. The focus on the coupled cases was the water inside the loop.

Line probes were used to obtain the temperature distributions in the riser tube walls and in the water in the riser tubes. Subsequently the convection heat transfer coefficients were calculated by using the temperatures.

The division of the mass flow rate into the three riser tubes wasn't the same for the two cases. In Variation B the mass flow rate of the middle riser tube was the highest, followed by Riser 1 and 3. In Variation C, the middle riser had the highest mass flow rate and Risers 1 and 3 had approximately the same mass flow rate. The division of the mass flow rate resulted in some differences in the wall temperatures in the risers. In Variation B the heat was centred around Riser 2, while being slightly offset from Riser 2 (between Riser 2 and Riser 1) in Variation C.

The Variation B model was stopped after approximately 14000 iterations, while the Variation C model was run for approximately 11000 iterations. The Variation C model therefore converged more efficiently. This is possibly due to the transferred heat flux, as opposed to the outlet temperature transferred in the Variation B model. It was observed that a transferred heat flux was generally more stable. Both variations however needed bottom and top limits to the transferred variables to prevent numerical flow reversals.

8.4 Comparison between the 3D and coupled 1D/3D results

Figure 8.4-1 shows the comparison of the results between the 3D CFD results (discussed in 8.2.2), Variation B results (8.3.1) and Variation C results (discussed in 8.3.2 and C.17). The results shown focus on the temperature distributions in the walls of the middle riser tube. Similarities between these specific temperature distributions are used as an indicator for the other results. The temperatures show good agreement between the approaches with differences below 2 % for corresponding temperatures.

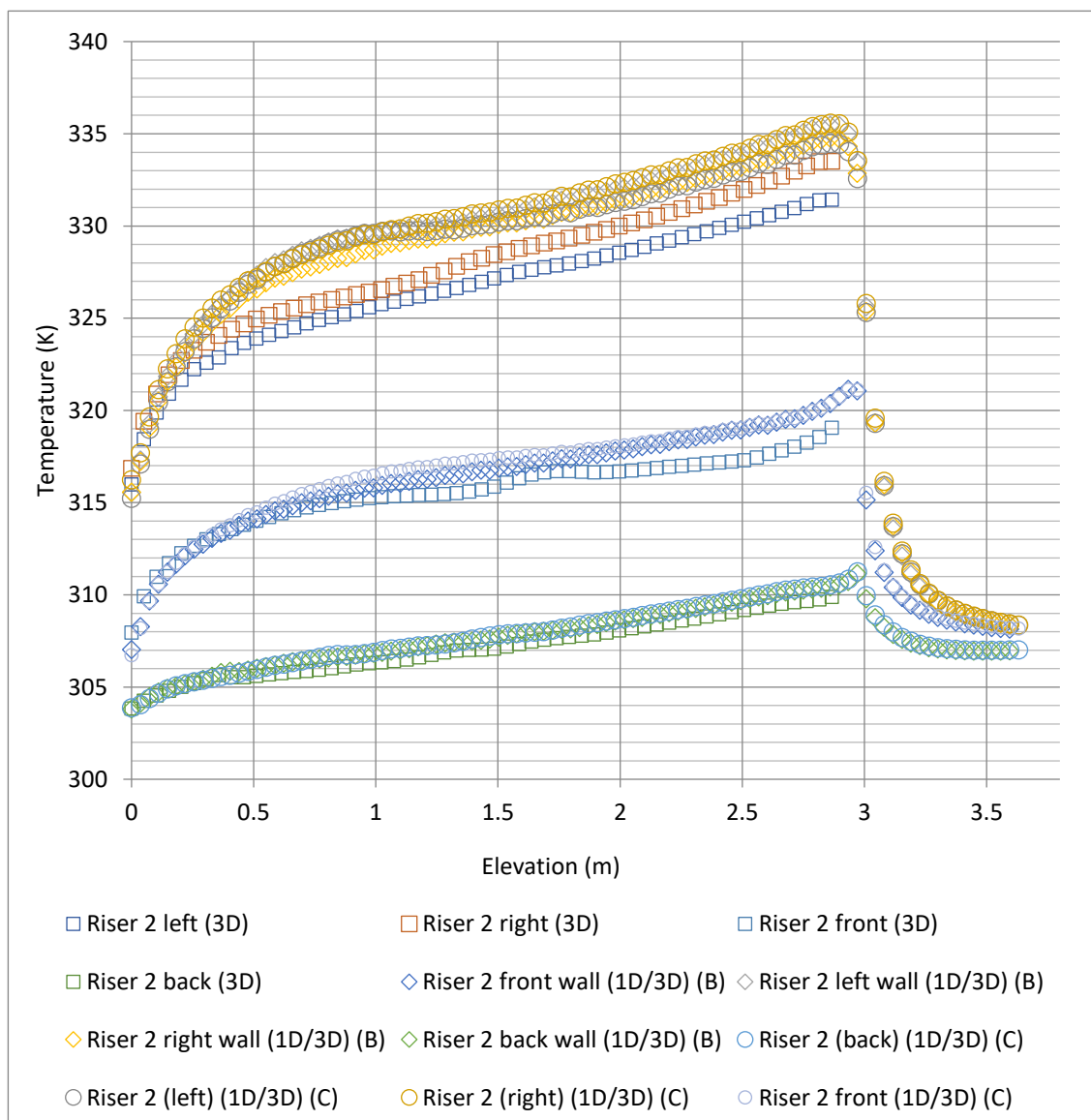


Figure 8.4-1. Comparison between the wall temperatures between the 3D and coupled approaches.

The mass flow rates in the three cases were differently divided between the riser tubes by the codes. In Variation B, Riser 1 had a mass flow rate of (0.194 kg/s), Riser 2 (0.196 kg/s) and Riser 3 (0.190 kg/s). In Variation C, Riser 2 had the highest mass flow rate (0.194 kg/s) with Risers 1 (0.193 kg/s) and 3 (0.193 kg/s) having a similar mass flow rate. In the 3D case, the mass flow rate was the same in the three riser tubes (0.192 kg/s).

Figure 8.4-2 compares the convection heat transfer coefficients that were calculated on four sides on the inner surface of the middle riser tube. The coefficients on the surfaces exhibit similar trends between the methodologies, however there are noticeable differences in the coefficients. Discounting the side walls (until later discussion) the largest difference between corresponding coefficients was approximately 15%. Reason for the difference is as follows: The differences in the mass flow rate division means that the mass flow rate in the Variation B model was slightly higher (0.002 kg/s or 1%). The Reynolds numbers were therefore different. The differences in the mass flow rate resulted in differences in the development of the temperature distributions in the riser tube walls. This in turn led to different temperature gradients between the fluid and wall temperatures for the two cases. The compound effect of the small deviations in wall and fluid temperatures resulted in ever increasing differences in the subsequent variables such as the Grashof number, Rayleigh number, Nusselt number and finally convection heat transfer coefficient.

The largest differences in Figure 8.4-2 are at the left- and right-side walls. The main reasons for the differences in this region was the proximity to the transition from mixed convection to the free convection regime. The side walls in all cases have Aicher numbers approaching a value of 0.2. The convection heat transfer coefficients are noticeably larger in the free convection regime, which causes the 'jumps' in the coefficients on the side walls as soon as the transition occurs. The coupled cases experienced the transition at lower elevations compared with the 3D case.

Although the boundary conditions were roughly the same, the 3D case had a fixed outlet pressure and inlet velocity based on hand calculations. Conversely the coupled cases had a feedback loop in which the pressure and inlet mass flow rates could be corrected per iteration. The 3D case therefore had a fixed total pressure of 115325 Pa while Variation B concluded on a total pressure of 115614 Pa and Variation C concluded on a total pressure of 115624 Pa. The

coefficients calculated by the 3D case would also be slightly different from the coupled cases for that reason.

The 3D case converged after the fewest number of iterations (10400 iterations; see section C.9) while the Variation B model ran for approximately 14000 iterations (see section C.16) and the Variation C model ran for approximately 11000 iterations (see section C.17). The 3D case was the most stable since the boundary conditions were fixed. The transfer of the heat flux was more stable than the outlet temperature, causing the Variation C model to converge more efficiently.

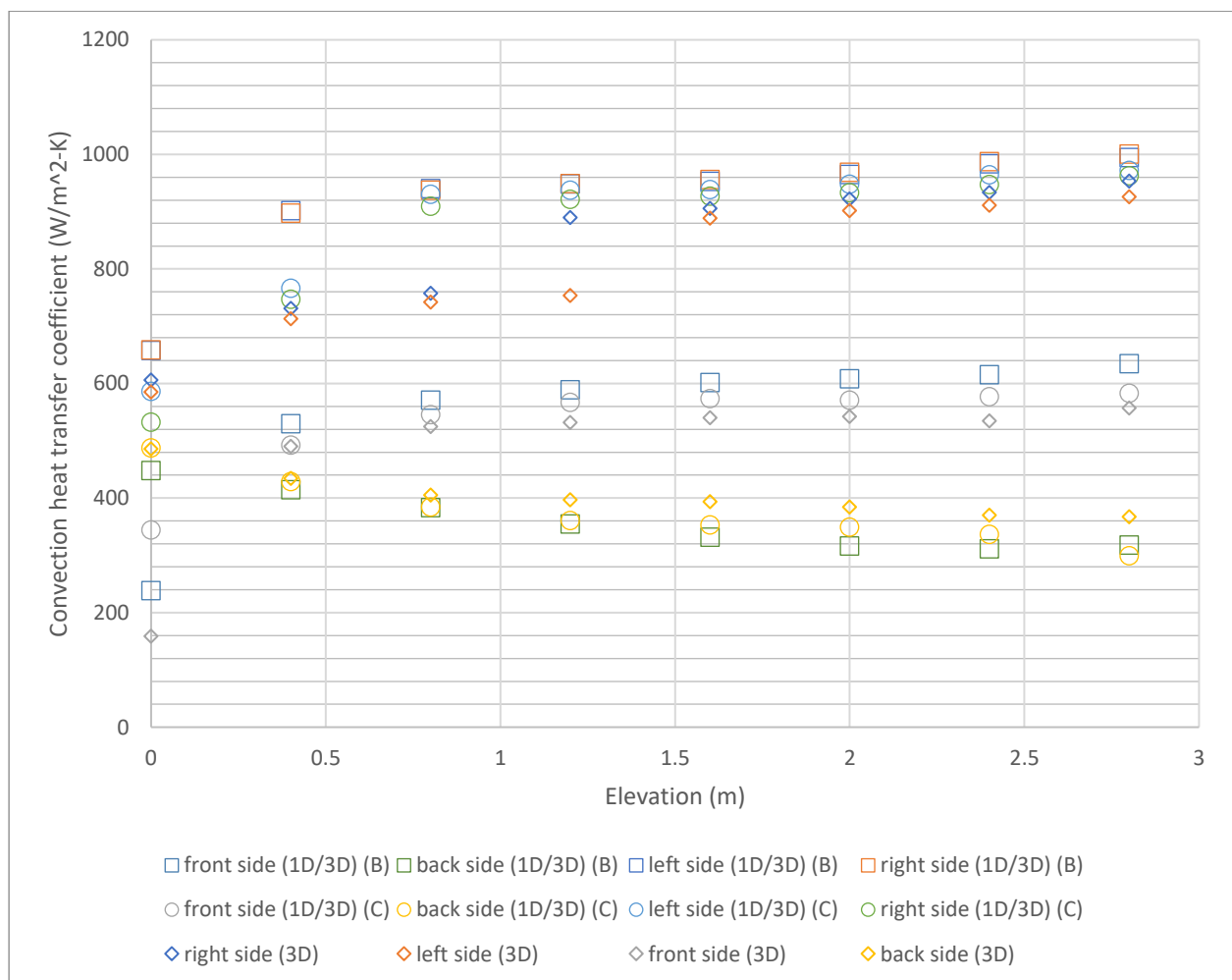


Figure 8.4-2. Comparison between the convection heat transfer coefficients.

In conclusion the results compare reasonably well in terms of the similar trends between the approaches. The temperature distributions compare fairly well, but small differences in temperature distributions compounded in other variables such as Grashof and Nusselt number to result in reduced agreement in the calculated coefficients. The feedback loop for the boundary conditions that is provided by the coupled methodology requires more computational time than a non-coupled methodology. The differences in the results infer the importance of the coupling, since there could be differences in the development in the flow fields between the tested methodologies. The specific differences in the development of the flow is left as future work.

CHAPTER 9 - CONCLUSIONS

9.1 Overview

The reactor cavity cooling system (RCCS) improves the safety of a nuclear reactor by removing decay heat from the reactor cavity. Experimental facilities have been constructed for the purpose of validating the designs as well as simulation codes such as Star-CCM+ and Fluent. Several facilities have been constructed and numerically modelled to this end.

Scaled down facilities are favoured for the lower cost in construction and reduced complexity of the subsequent analysis. Scales of $\frac{1}{2}$ and $\frac{1}{4}$ have been used for the vertical scaling of prototypes into experimental facilities. Lateral scales however are kept to 1:1 ratio. Radial slices of the prototype designs mean that less than 10 of the total riser tubes are built into experimental facilities, which greatly reduces the numerical models' complexity without sacrificing accuracy.

Two of the methods that can be used to simulate the RCCS are full 3D computational fluid dynamics (CFD) and 1D systems CFD. The latter is simpler and uses less computational resources but is not as accurate as the 3D method. By combining the methodologies, the complex 3D phenomena can be solved with high accuracy, while limiting computational resources in regions with relatively simple phenomena.

A facility has been constructed at the University of Wisconsin representing a $\frac{1}{4}$ scale model of the RCCS of the MHTGR. Experiments have been carried out and reported by Lisowski (2013), as well as numerical simulations. The current study aimed to expand on the simulations done by Lisowski (2013). The focus was on one of the single-phase experiments by Lisowski (2013), from which the boundary conditions were derived. The study looked at the two air filled cavities as well as the flow in the riser tubes. The distribution of the heat transfer as well as the mechanisms (radiation/convection) were investigated associated with the surfaces of the riser tubes. In addition, the extracted temperature distributions were used to calculate the convection heat transfer coefficients.

9.2 Conclusion from 3D CFD studies

The system's sensitivity to the geometry was investigated by also simulating different shapes of the riser tubes. The variations simulated, were therefore circular with fins (as the experimental facility), circular tubes without fins, square tubes with fins, square tubes without fins, rectangular

tubes with fins and rectangular tubes without fins. In the cases with rectangular fins, the number of the tubes was doubled. The total cross-sectional area was kept constant when changing the geometry of the riser tubes. The temperature distributions were one of the results that were compared between cases.

The temperature distributions around the cavity was more uniform in the cases without fins. The stresses on the riser tubes due to thermal expansion would be lower in these cases. The back-wall temperature was however lower with the finned configurations (in some cases more than a 200 K difference between the heater and back walls). A lower back wall temperature is preferable in terms of the assurance of the structural integrity of the concrete enclosure.

The convection heat transfer coefficients were calculated for the heater wall, back wall, riser tube outer surfaces and for the riser tube inner surfaces. Where applicable coefficients were also calculated for the surfaces of the fins. The contribution of the study is partly in these coefficients which were also calculated for the back cavity, which was a region that was overlooked in previous works. The parameter referred to as the Aicher number was used to determine the convection regime and determined that mixed convection dominated in the riser tubes for the circular and square tubes and free convection dominated for the rectangular tubes. This implies that the convection heat transfer in the rectangular tubes was the most efficient, as the mixed convection regime resulted in a deterioration of convection heat transfer. In the rectangular case twice the number of tubes were used, each with $\frac{1}{2}$ of the cross-sectional area of a single tube of the other geometries.

9.3 Conclusion from coupled 1D/3D CFD studies

In the coupled simulations two variations were done. In the first variation the mass flow rate was fixed in Flownex. The mass flow rate, inlet temperature and outlet pressure were transferred as boundary conditions from Flownex to Fluent. The outlet temperature was then calculated in Fluent to be transferred back to Flownex. Flownex would then calculate the full system loop and adjust the boundary conditions accordingly. The second variation relied on the coupled codes to calculate the mass flow rate (which was therefore not fixed). Another difference from the other variation was that instead of the outlet temperature, Fluent transferred a heat flux back to Flownex.

The data extracted for comparison from the coupled simulations was the water temperatures in the riser tubes as well as the wall temperatures of the riser tubes. The fluid and wall temperatures were used to calculate the convection heat transfer coefficients at various points.

The wall temperatures of the two coupled variations were compared with each other as well as the corresponding 3D CFD case. The wall temperatures show a good degree of agreement with less than 2% difference between corresponding temperatures.

The convection heat transfer coefficients compare well between the approaches in terms of the trends in the coefficients. Small differences in the temperature distributions as well as small differences in the calculated mass flow rates compounded to result in reduced agreement between the calculated coefficients. This was indicative of differences in the flow development in the riser tubes, between the methodologies. Future work could possibly expand on the flow development resulting from the different methods.

9.4 Recommendations

In the actual facility the fins were cut to allow for thermal expansion. In this work the fins were simplified to exclude the cutting of the fins. The flow patterns would be influenced by this geometry and could be investigated.

The true 5° slice geometry could also be investigated, since the flow patterns and radiation heat transfer would be slightly different for a system in which the fin and tube assembly is curved to follow the curved wall of the reactor cavity.

The flow development could be studied in more detail, especially comparing the fixed velocity vs. fixed mass flow rate vs. numerically developing mass flow rate.

More comprehensive and extended studies could be done on 1D/3D coupling of reactor cavity cooling systems.

BIBLIOGRAPHY

Aicher, T. & Martin, H. 1997. New correlations for mixed turbulent natural and forced convection heat transfer in vertical tubes. *International Journal of Heat and Mass Transfer*, 40(15):3617-3626.

Ampofo, F. & Karayiannis, T.G. 2003. Experimental benchmark data for turbulent natural convection in an air filled square cavity. *International Journal of Heat and Mass Transfer*, 46(19):3551-3572.

Ansys. 2017a. Ansys documentation: Fluent theory guide.

Ansys. 2017b. Ansys documentation: Meshing user's guide.

Ansys. 2021a. Computational fluid dynamics (CFD) software program solutions. <https://www.ansys.com/products/fluids/ansys-cfx>. Date of Access: 17/02/2021.

Ansys. 2021b. Ansys Fluent. <https://www.ansys.com/products/fluids/ansys-fluent>. Date of access: 17/02/2021.

Bae, Y.Y., Hong, S.D. & Kim, M.H. 2012. Scaling analysis of reactor cavity cooling system for PMR200. Paper presented at the Proceedings of the HTR, Tokyo, Japan, October 28 - November 1.

Bechtel National INC, Combustion Engineering INC, Department of Energy, EG&G Idaho INC, GA Technologies INC, Gas-cooled Reactor Associates, General Electric Company, Oak Ridge National Laboratory & Stone and Webster Engineering Corp. 1986. Preliminary safety information document for the standard MHTGR.

Behzadmehr, A., Galanis, N. & Laneville, A. 2003. Low Reynolds number mixed convection in vertical tubes with uniform wall heat flux. *International Journal of Heat and Mass Transfer*, 46(2003):4826-4833.

CD-adapco. 2006. CCM user guide; Star-CD Version 4.02. <https://www.scribd.com/doc/193836790/Star-CCM-User-Guide>. Date of access: 17/02/2021.

- Celata, G.P., Dannibale, F., Chiaradia, A. & Cumo, M. 1998. Upflow turbulent mixed convection heat transfer in vertical pipes. *International Journal of Heat and Mass Transfer*, 41(24):4037-4054.
- Corrandini, M. 2012. Experimental studies of NGNP reactor cavity cooling system with water.
- Du Toit, C.G., Rousseau, P.G., Jun, J.S. & Noh, J.M. 2014. 1D and 3D numerical simulation of the reactor cavity cooling system of a very high temperature reactor. (In. 15th International heat transfer conference organised by Kyoto, Japan. p. p13).
- Frisani, A. & Hassan, Y.A. 2014. Computation fluid dynamics analysis of the Reactor Cavity Cooling System for Very High Temperature Gas-Cooled Reactors. *Annals of Nuclear Energy*, 72:257-267.
- Haque, H., Feltes, W. & Brinkmann, G. 2006. Thermal response of a modular high temperature reactor during passive cooldown under pressurized and depressurized conditions. *Nuclear Engineering and Design*, 236(5):475-484.
- IAEA. 2000. Heat transport and afterheat removal for gas cooled reactors under accident conditons. Vienna, Austria.
- Incropera, F.P., Dewitt, D.P., Bergman, T.L. & Lavine, A.S. 2013. Principles of heat and mass transfer. 7 th John Wiley & Sons.
- Jackson, J.D., Cotton, M.A. & Axcell, B.P. 1989. Studies of mixed convection in vertical tubes. *International Journal of Heat and Fluid Flow*, 10(1):2-15.
- Ji, Y. 2014. CFD Modelling of Natural Convection in Air Cavities. *CFD Letters*, 6.
- Keshmiri, A. 2014. Verification and Validation of Three Different CFD Codes in Simulating Mixed Convection Flows Using Two Advanced Eddy-Viscosity Models.
- Kim, J.H., Bae, Y.Y., Hong, S.D., Kim, C.S., Cho, B.H. & Kim, M.H. 2014. Results of the preliminary test in the 1/4-scale RCCS of the PMR200 VHTR. Paper presented at the Korean Nuclear Society Autumn Meeting, Pyeongchang, Korea, October 30-31.
- Klein, S.A. 2018. Engineering Equation Solver Manual (Version V10.412).
- Kööp, K. 2018. Application of a system thermal-hydraulics code to development of validation process for coupled STH-CFD codes. KTH Royal Institute of Technology.

- Lim, H.S., Tak, N.-i., Lee, S.N. & Jo, C.K. 2018. Water-jacket reactor cavity cooling system concept to mitigate severe accident consequence of high temperature gas-cooled reactor. *Nuclear Engineering and Design*, 340:156-165.
- Lisowski, D.D. 2013. Thermal hydraulic analysis of an experimental reactor cavity cooling system with water: performance and stability. University of Wisconsin - Madison.
- Lisowski, D.D., Haskin, T.C., Tokuhiko, A., Anderson, M.H. & Corradini, M.L. 2012. Study on the behaviour of an asymmetrically heated reactor cavity cooling system with water in single phase. *Nuclear Technology*, 183:75-87.
- Lisowski, D.D., Kraus, A.R., Bucknor, M.D., Hu, R. & Farmer, M.T. 2016. Experimental observations of natural circulation flow in the NSTF. *Nuclear Engineering and Design*, 306:124-132.
- Lommers, L.J., Mays, B.E. & Shahrokhi, F. 2014. Passive heat removal impact on AREVA HTR design. *Nuclear Engineering and Design*, 271:569-577.
- Lomperski, S., Pointer, W.D., Tzanos, C.P., Wei, T.Y.C. & Krauss, A.R. 2011. Generation IV nuclear energy system initiative: air-cooled option RCCS studies and NSTF preparation.
- M-Tech Industrial. Flownex library manual (Version 8.10.0.0.3800).
- M-Tech Industrial. 2019a. Flownex library manual (Version 8.10.0.0.3800).
- M-Tech Industrial. 2019b. Flownex simulation environment: Flownex theory manual (Version 8.10.0.0.3800).
- Munson, B.R., Young, D.F. & Okiishi, T.H. 2006. Fundamentals of fluid mechanics. 5th ed. Hoboken, NJ: J. Wiley & Sons.
- Neylan, A.J., Shenoy, A., Silady, F.A. & Dunn, T.D. 1994. GT-MHR design, performance, and safety. (*In*. Development Status of Modular High Temperature Reactors and Their Future Role organised by Pettern, Netherlands.
- Oh, C.H., Park, G.C. & Davis, C. 2009. RCCS Experiments and Validation for High-Temperature Gas-Cooled Reactor. *Nuclear Technology*, 167(1):107-117.
- Pal, E., Kumar, M., Nayak, A.K. & Joshi, J.B. 2016. Experimental and CFD simulations of fluid flow and temperature distribution in a natural circulation driven Passive Moderator Cooling System of an advanced nuclear reactor. *Chemical Engineering Science*, 155:45-64.

- Prasad, G.V.D., Pandey, M. & Kalra, M.S. 2007. Review of research on flow instabilities in natural circulation boiling systems. *Progress in Nuclear Engineering*, 49(2007):429-451.
- Raghav, R. 2018. Near wall treatment. <https://studentcommunity.Ansys.com/thread/heat-wall-treatment/>. Date of access: 02/10/2019.
- Rajamohan, G., Ramesh, N. & Kumar, P. 2011. Study on mixed convection heat transfer in vertical ducts with radiation effects. *World Academy of Science, Engineering and Technology*, 57:671-675.
- Rousseau, P.G., du Toit, C.G., Jun, J.S. & Noh, J.M. 2015. Code-to-code comparison for analysing the steady-state heat transfer and natural circulation in an air-cooled RCCS using GAMMA+ and Flownex. *Nuclear Engineering and Design*, 291:71-89.
- Scheele, G.F., Rosen, E.M. & Hanratty, T.J. 1960. Effect of natural convection on transition to turbulence in vertical pipes. *The Canadian Journal of Chemical Engineering*, 38(3):67-73.
- Shin, D.-H., Kim, C.S., Park, G.-C. & Cho, H.K. 2017a. Experimental analysis on mixed convection in reactor cavity cooling system of HTGR for hydrogen production. *International Journal of Hydrogen Energy*, 42(34):22046-22053.
- Shin, D.H., Kim, S.Y., Kim, C.S., Cho, H.K. & Park, G.C. 2017b. Experimental study on the mixed convection phenomena in the air-cooled RCCS riser. Paper presented at the The Korean Nuclear Society Spring Meeting, Jeju, Korea, May 18 - 19.
- Siemens PLM Software. 2019. Simcenter Star-CCM+® documentation. Version 2019.1.
- Slabber, J., Van der Merwe, L., Theron, W. & Matzner, H.D. 2006. Technical description of the PBMR demonstration power plant.
- Symolon, P., Neuhaus, W. & Odell, R. 2004. Mixed convection heat transfer experiments in a smooth and rough vertical tubes. Niskayuna, New York.
- Tian, J. & Karayiannis, T.G. 2001. Low turbulence natural convection in an air filled square cavity, part 1: the thermal and fluid flow fields. *Journal of Heat and Mass Transfer*, 43: 849-866.
- Tzanos, C.P. & Farmer, M.T. 2006. Feasibility study for use of the natural convection shutdown heat removal test facility (NSTF) for VHTR water-cooled RCCS shutdown.
- Versteeg, H.K. & Malalasekera, W. 2007. Introduction to computational fluid dynamics. 2nd Pearson Education Limited.

- Wagner, W. & Pruss, A. 1993. International Equations for the Saturation Properties of Ordinary Water Substance. Revised According to the International Temperature Scale of 1990. Addendum to J. Phys. Chem. Ref. Data 16, 893 (1987). *Journal of Physical and Chemical Reference Data*, 22(3):783-787.
- Zhao, H., Dong, Y., Zheng, Y., Ma, T. & Chen, X. 2017. Numerical simulation on heat transfer process in the reactor cavity of modular high temperature gas-cooled reactor. *Applied Thermal Engineering*, 125(Supplement C):1015-1024.
- Zhao, H., Zheng, Y., Chen, X., Ma, T. & Dong, Y. 2019. A one-dimensional code of the passive residual heat removal system for the modular high temperature gas-cooled reactor. *Progress in Nuclear Energy*, 110:374-383.
- Zhou, B., Yuan, J., Lu, J. & Hong, F. 2017. Investigation on transient behavior of residual heat removal pumps in 1000MW nuclear power plant using a 1D-3D coupling methodology during start-up period. *Annals of Nuclear Energy*, 110:560-569.
- Zitzmann, T., Cook, M., Pfrommer, P., Rees, S. & Marjanovic, L. 2005. *Simulation of steady-state natural convection using CFD*. Paper delivered at the Ninth International IBSA Conference, Montreal, Canada.
- http://www.ibpsa.org/proceedings/bs2005/bs05_1449_1456.pdf. Date of access: 08/03/2021.

APPENDIX A MATERIAL PROPERTIES

A.1 Material properties

The material properties of principal importance were the thermal conductivity, specific heat capacity, density and viscosity. Fair comparison between the simulation codes was assured by setting the material properties to be calculated in the same manner. The properties for air and water were calculated by specifying a polynomial as a function of temperature. The properties of the steel were assumed to be constant.

The property curves of air and water were plotted by using Engineering Equation Solver (EES). The properties of water used by EES (Klein, 2018) are based on the work of (Wagner & Pruss, 1993). A polynomial curve fit of the third order was applied in all cases.

The properties that were calculated were the dynamic viscosity, density, specific heat capacity and conductivity. The temperature range was chosen based on the temperatures that would occur during the experiments. The temperature range for water would be well within 300 – 369 K and the range for air would be within 300 - 900 K.

The graph of water viscosity as a function of temperature is shown in Figure A-1, with the polynomial equation that was fit to the curve. Similarly, the water conductivity is shown in Figure A-2, specific heat capacity is shown in Figure A-3 and density is shown in Figure A-4.

The polynomial equations fitted to the properties of water are listed in Equations A-1 to A-4.

$$\mu_{\text{water}} = 0.05484 - 4.41963 \times 10^{-4} T + 1.20461 \times 10^{-6} T^2 - 1.10433 \times 10^{-9} T^3 \quad \text{A-1}$$

$$k_{\text{water}} = -1.33364 + 0.013213 T - 2.85425 \times 10^{-5} T^2 + 2.03013 \times 10^{-8} T^3 \quad \text{A-2}$$

$$C_{p_{\text{water}}} = 5544.45 - 10.2781 T + 0.0236363 T^2 - 1.51138 \times 10^{-5} T^3 \quad \text{A-3}$$

$$\rho_{\text{water}} = 463.984 + 4.69595 T - 0.0126127 T^2 + 9.59093 \times 10^{-6} T^3$$

A-4

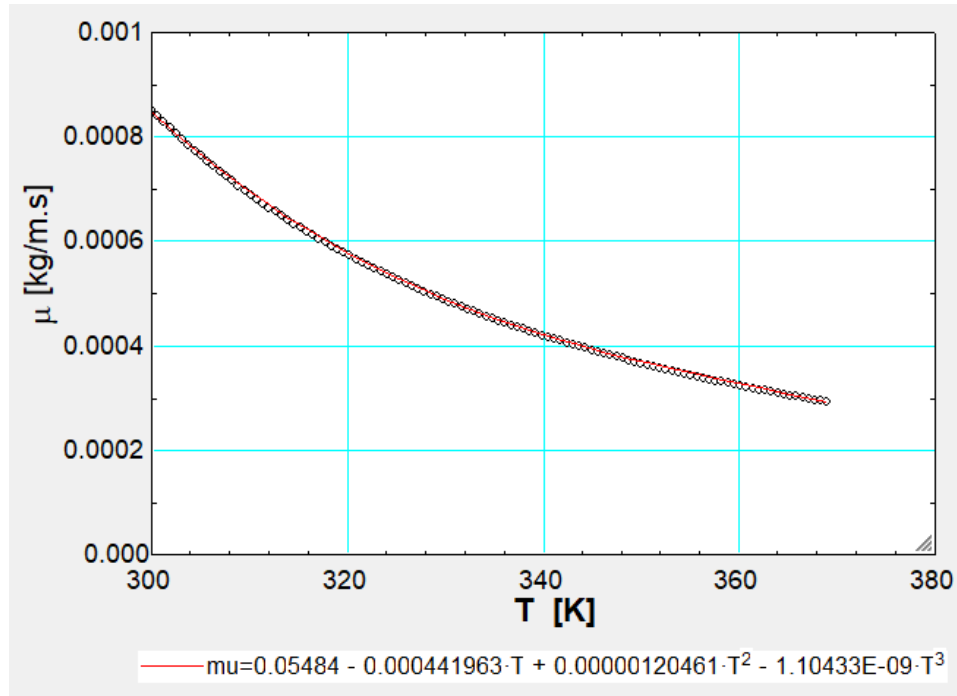


Figure A-1. Dynamic viscosity of water with the polynomial function.

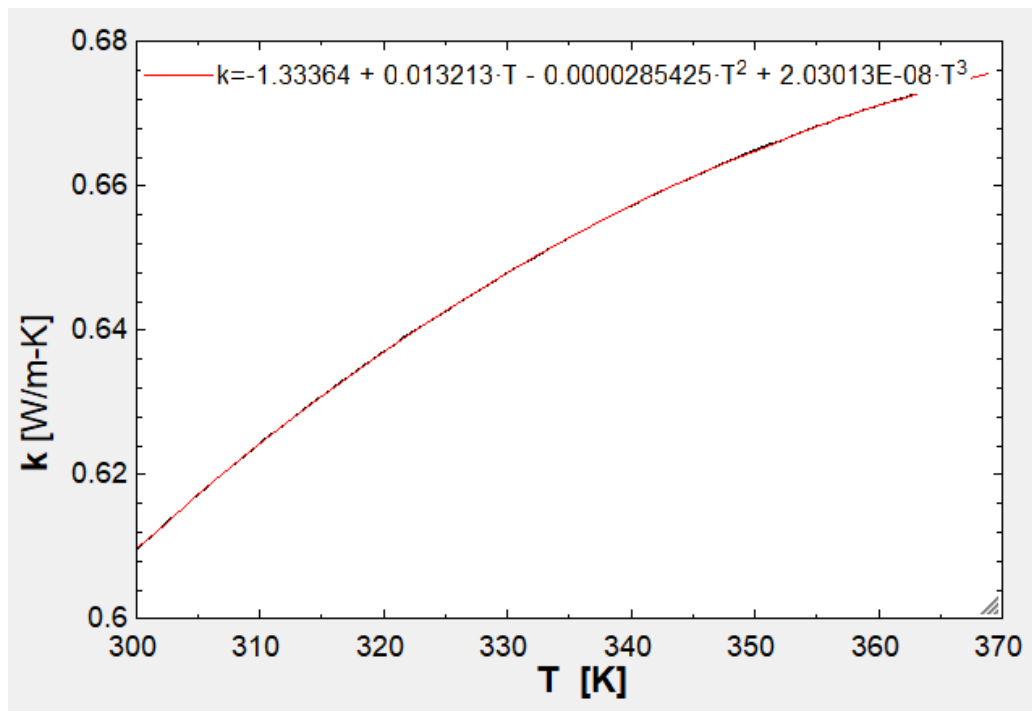


Figure A-2. Polynomial of conductivity in terms of temperature of water.

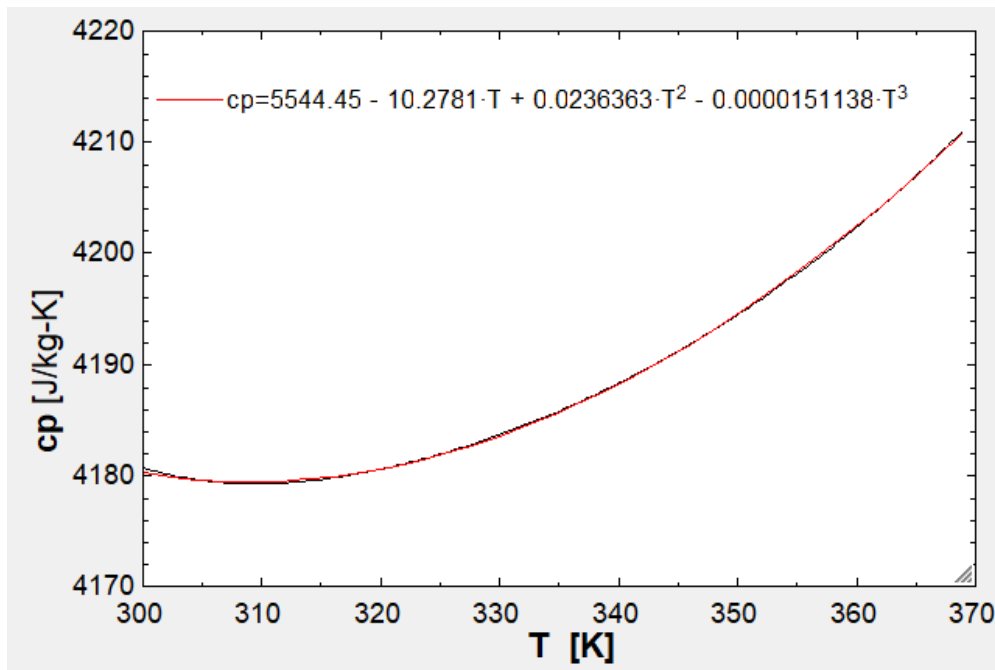


Figure A-3. Polynomial function of specific heat capacity in terms of temperature for water.

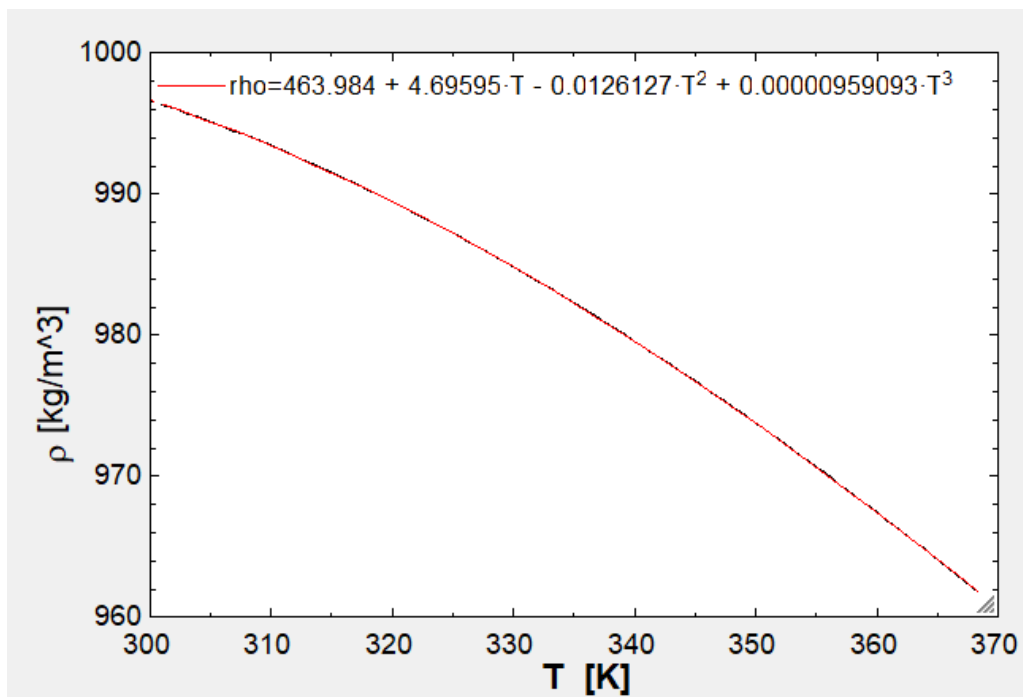


Figure A-4. Polynomial of density in terms of temperature for water.

Properties for air along with the polynomials fitted to the curves are shown in Figure A-5 to Figure A-8 consisting of the specific heat capacity, thermal conductivity, density and finally dynamic viscosity. The equations of the polynomial fitted curves corresponding to the figures are listed in Equation A-5 to Equation A-8.

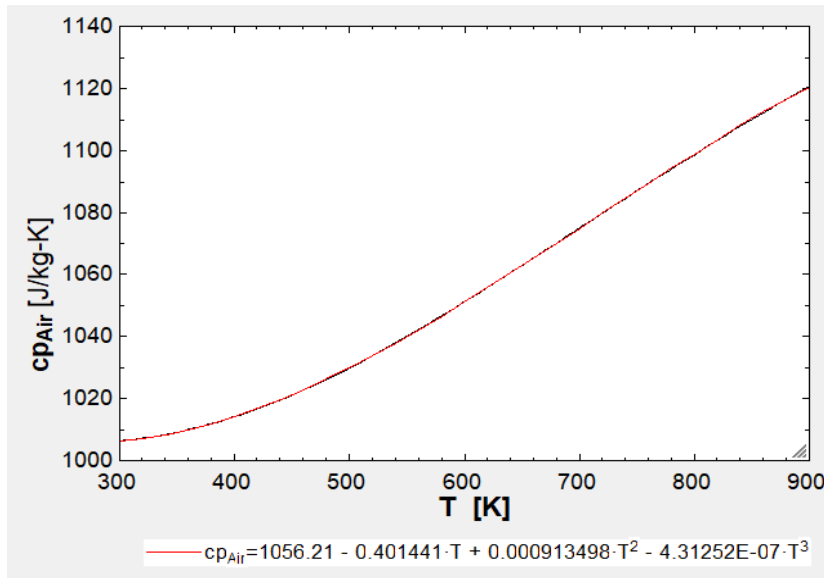


Figure A-5. Polynomial of specific heat capacity in terms of temperature for air.

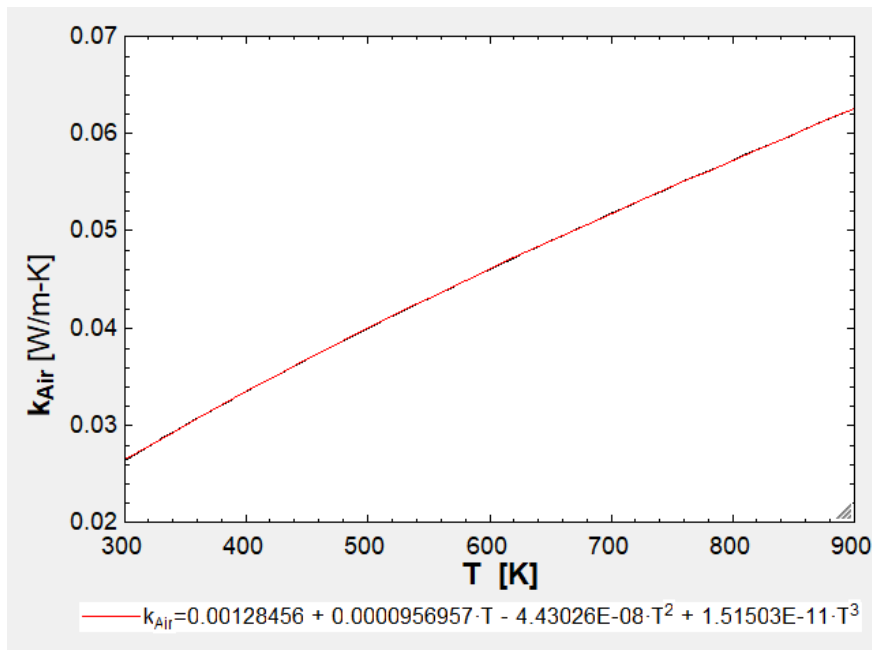


Figure A-6. Conductivity polynomial in terms of temperature for air.

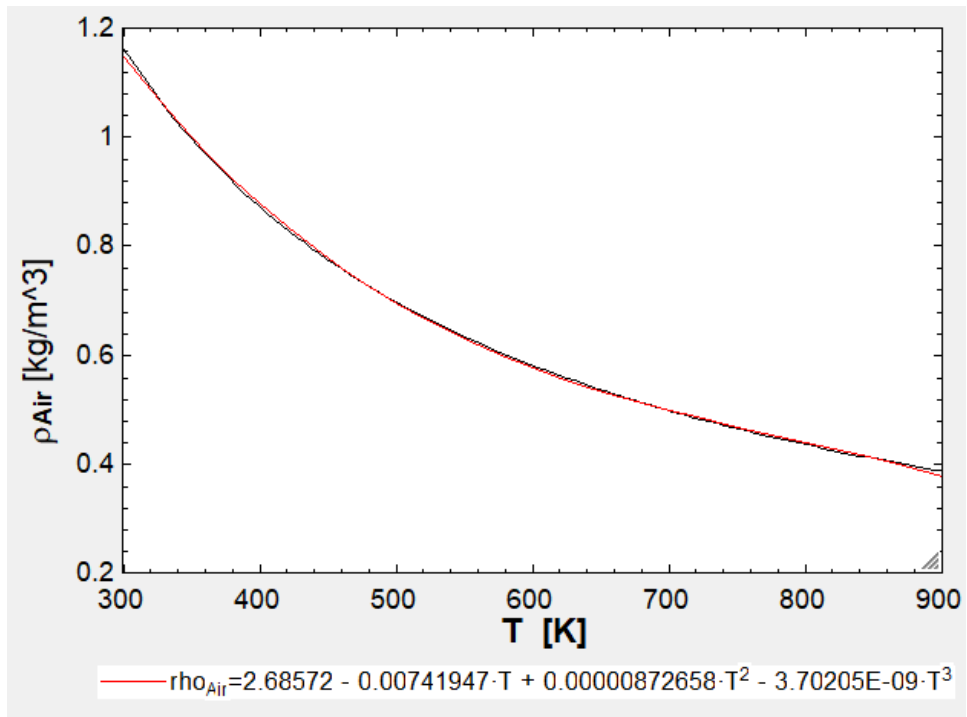


Figure A-7. Polynomial of density in terms of temperature for air.

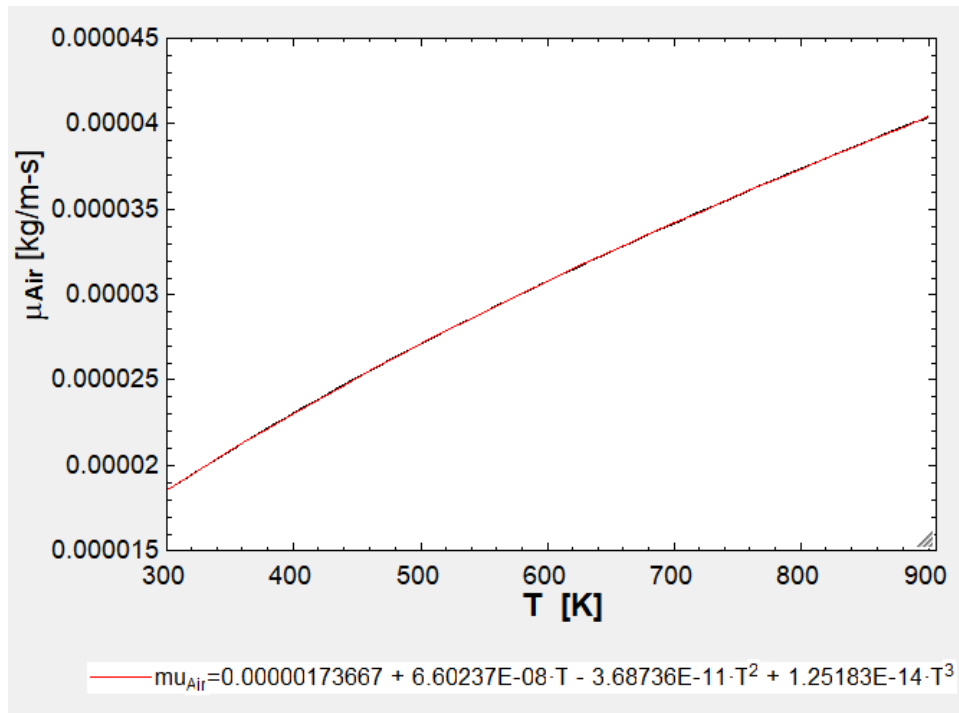


Figure A-8. Polynomial of viscosity in terms of temperature for air.

$$Cp_{air}=1056.21-0.401441 T+9.13498 \times 10^{-4} T^2-4.31252 \times 10^{-7} T^3 \quad A-5$$

$$k_{air}=0.00128456+9.56957 \times 10^{-5} T-4.43026 \times 10^{-8} T^2+1.51503 \times 10^{-11} T^3 \quad A-6$$

$$\rho_{air}=2.68572-0.00741947 T+8.72658 \times 10^{-6} T^2-3.70205 \times 10^{-9} T^3 \quad A-7$$

$$\mu_{air}=1.73667 \times 10^{-6}+6.60237 \times 10^{-8} T-3.68736 \times 10^{-11} T^2+1.25183 \times 10^{-14} T^3 \quad A-8$$

In conclusion the calculation of the material properties of the water and air was discussed. The specific heat capacity, dynamic viscosity, thermal conductivity and density were plotted against temperature by using EES. A polynomial fit was applied to the graphs plotted by EES and was subsequently implemented in the CFD codes to calculate the fluid properties.

APPENDIX B MESH QUALITIES

B.1 Mesh quality

The mesh qualities are given in this chapter in more detail. The mesh parameters considered were the cell skewness, orthogonal quality and aspect ratio. An optimum cell skewness would be a value close to 0 with skewness of 1 considered as very poor quality. A value of 0.75 would still be considered fair quality. (Ansys, 2017b).

The orthogonal quality should be as close as possible to a value of 1. The lowest acceptable value for a good quality mesh would be 0.333. (Ansys, 2017b).

The aspect ratio should have a value lower than 100. (Ansys, 2017b).

B.2 Mesh dependence study

The mesh dependence study in section 5.1.3 comprised of three meshes with varying base sizes. The meshes were numbered 1 to 3 from least number of elements to the greatest number of elements. These meshes along with their qualities are presented in sections B.3, B.4 and B.5.

B.3 Mesh number 1 (section 5.1.3)

The first mesh was generated by specifying maximum face and tetrahedron sizes of 7 mm, 5 prism layers of growth ratio 1,2 and transition ratio of 0.5. The mesh obtained by these settings is shown in Figure B-1. The mesh contained 3 925 944 nodes and 3 548 028 elements.

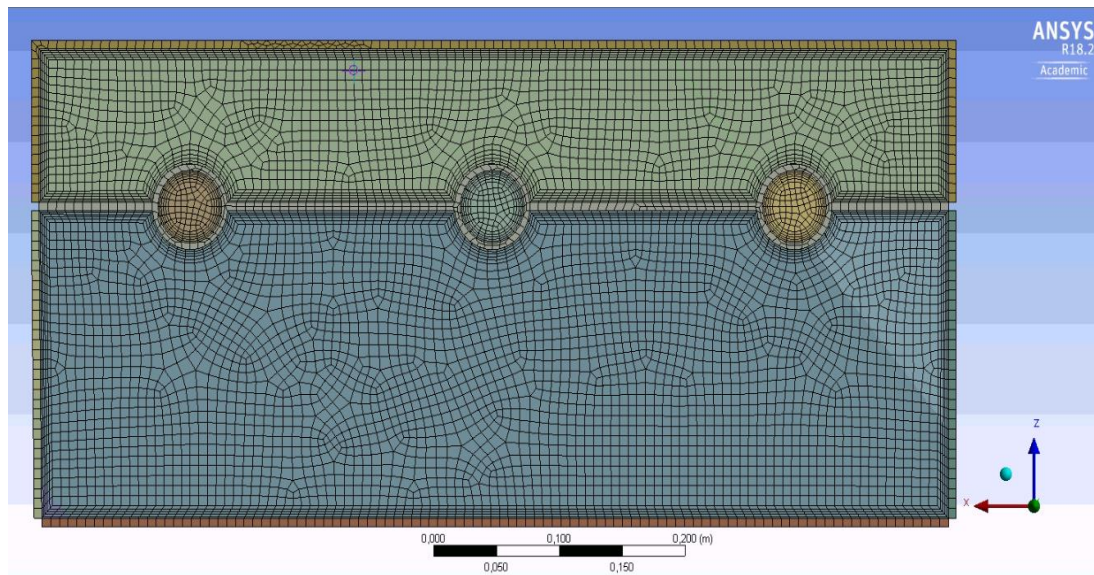


Figure B-1. Mesh number 1 generated by Ansys.

The mesh skewness was between 1.3058×10^{-10} and 0.71625. The average skewness was 0.1574 with a standard deviation of 0.1475. The cell skewness distribution is shown in Figure B-3.

The mesh orthogonal quality was between 0.49233 and 1, with an average of 0.94992 and standard deviation of 7.8479×10^{-2} . The cell orthogonal quality distribution is shown in Figure B-2.

The aspect ratio was calculated to average 2.4637 with a standard deviation of 1.6099 with minimum and maximum values of 1.0052 and 6.5038 respectively. The cell aspect ratio distribution is shown in Figure B-4.



Figure B-2. Orthogonal quality of mesh number 1.

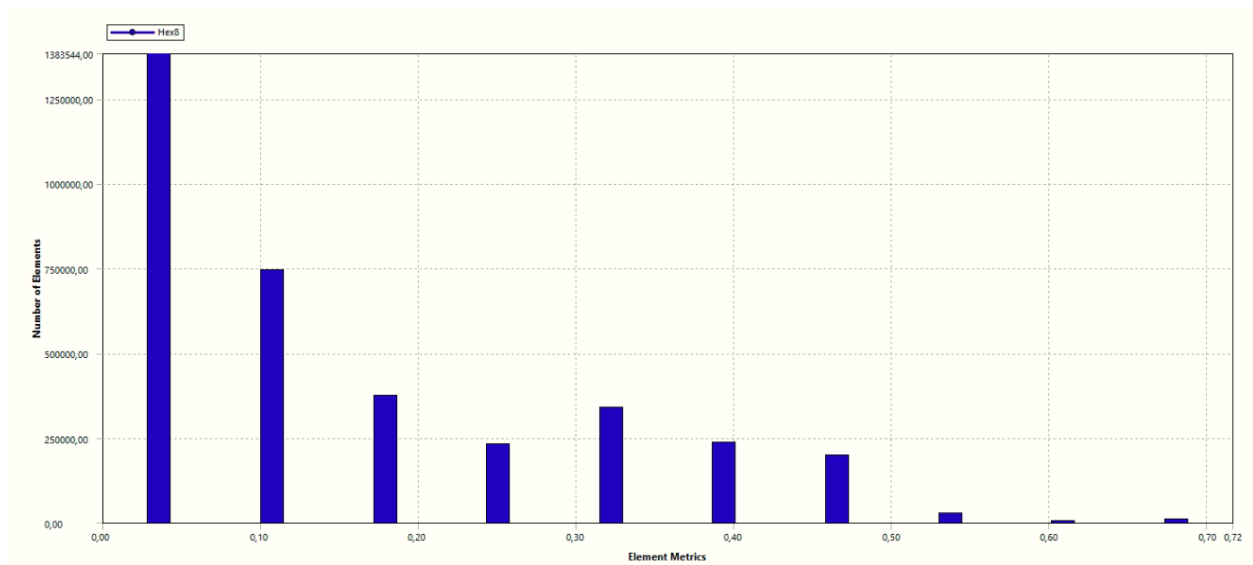


Figure B-3. Skewness of mesh number 1.

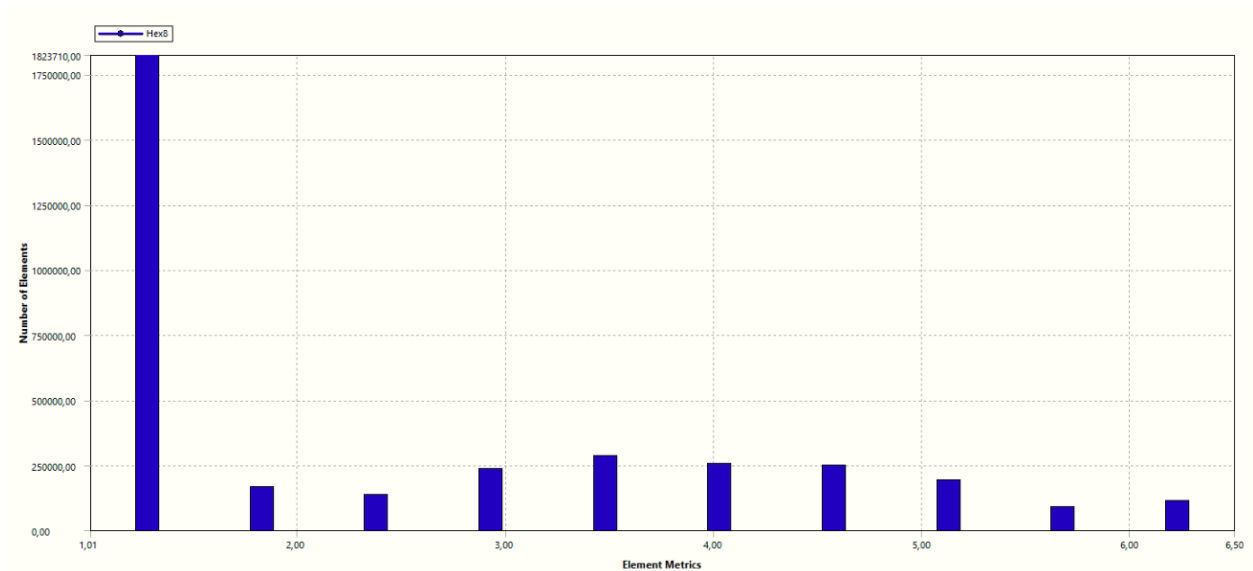


Figure B-4. Cell aspect ratio for mesh number 1.

Since the above-mentioned parameters were within acceptable limits the mesh was assumed to be of sufficient quality.

B.4 Mesh number 2 (section 5.1.3)

Mesh number 2 was generated by specifying maximum face and tetrahedral size of 5 mm. Eight prism layers were specified with a transition ratio of 0.5 and growth rate of 1.2. The mesh settings yielded a mesh (shown in Figure B-5) containing 10 593 593 nodes and 9 866 360 elements.

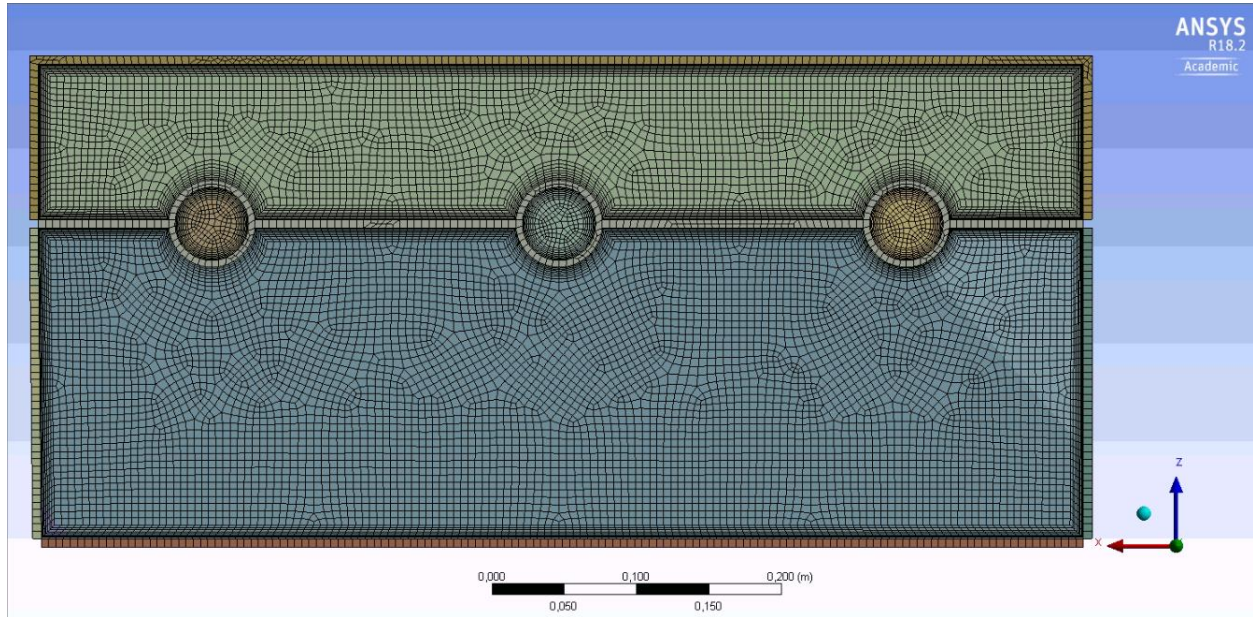


Figure B-5. Mesh generated for mesh number 2.

The average mesh skewness was 0.15664 with a standard deviation of 0.14582 with minimum and maximum values of 1.3058×10^{-10} and 0.90907, respectively. The cell skewness distribution is shown in Figure B-6.

The average orthogonal quality was 0.95002 with a standard deviation of 7.6407×10^{-2} and minimum and maximum values of 0.59515 and 1, respectively. The cell orthogonal quality distribution is shown in Figure B-7.

The average aspect ratio was 2.8484 with a standard deviation of 2.1555, with minimum and maximum values of 1.0018 and 9.9895 respectively. The cell aspect ratio distribution is shown in Figure B-8.

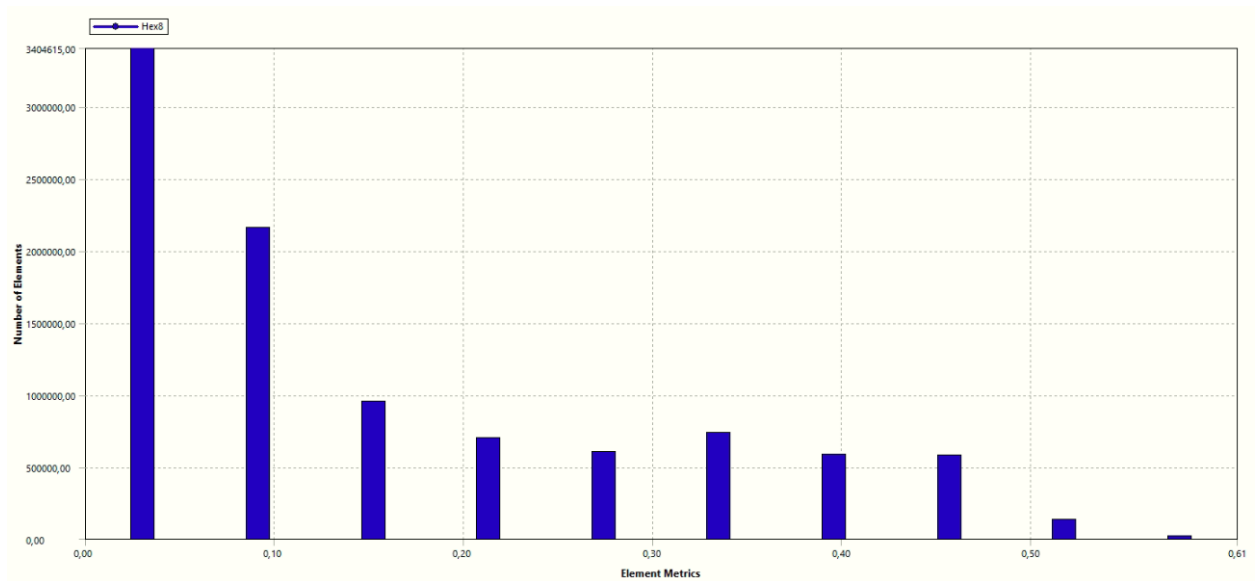


Figure B-6. Mesh skewness for mesh number 2.

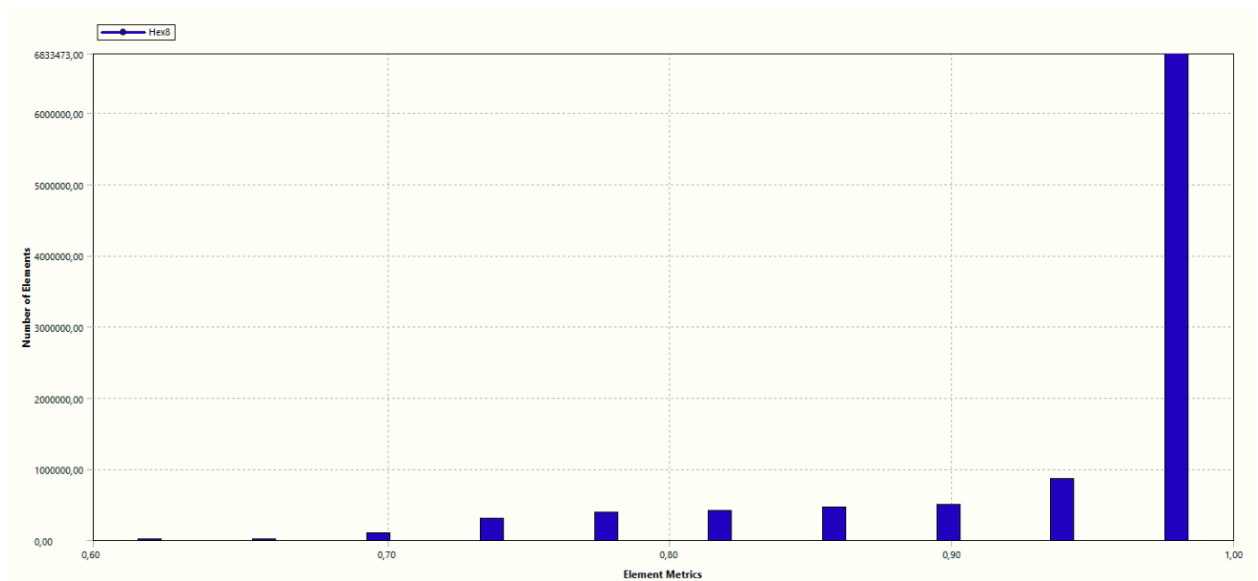


Figure B-7. Orthogonal quality for mesh number 2.

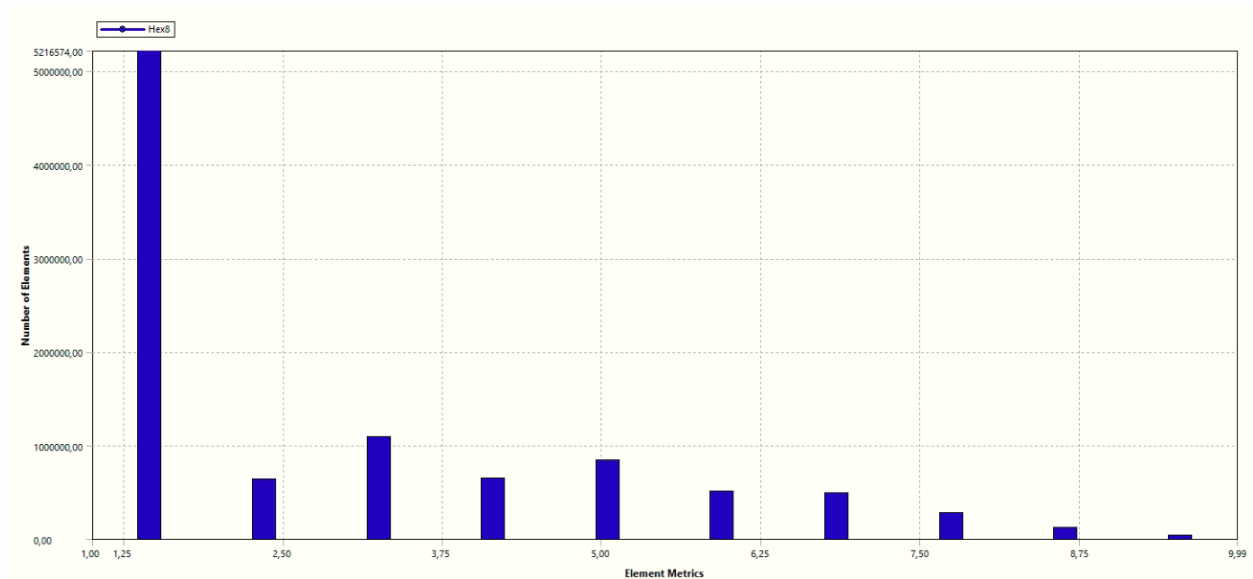


Figure B-8. Cell aspect ratio for mesh number 2.

B.5 Mesh number 3 (section 5.1.3)

Figure B-9 was generated by specifying maximum face and tetrahedron size of 3 mm. Eleven prism layers were used with a transition ratio of 0.5 and growth rate of 1.2. The settings yielded a mesh containing 44 083 043 nodes and 42 074 876 elements.

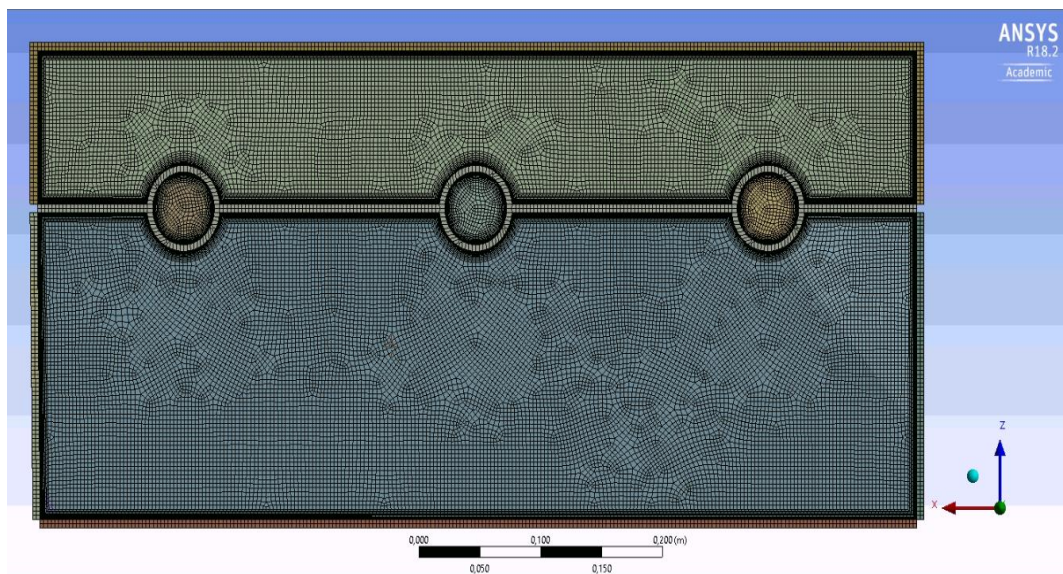


Figure B-9. Mesh generated for mesh number 3.

The average mesh skewness was 0.13156 with a standard deviation of 0.14373. The minimum and maximum values were $8.9722\text{e-}9$ and 0.5564 respectively. The cell skewness distribution is shown in Figure B-10.

The average mesh orthogonal quality was 0.95833 with a standard deviation of $7.1872\text{e-}7$. The minimum and maximum values were 0.70148 and 1 respectively. The cell orthogonal quality distribution is shown in Figure B-11.

The minimum and maximum aspect ratios were 1.001 and 10.132 respectively. The average aspect ratio was 2.807 with a standard deviation of 2.4624. The cell aspect ratio distribution is shown in Figure B-12. Since the above-mentioned values were within acceptable limits the mesh was considered of a sufficient quality.

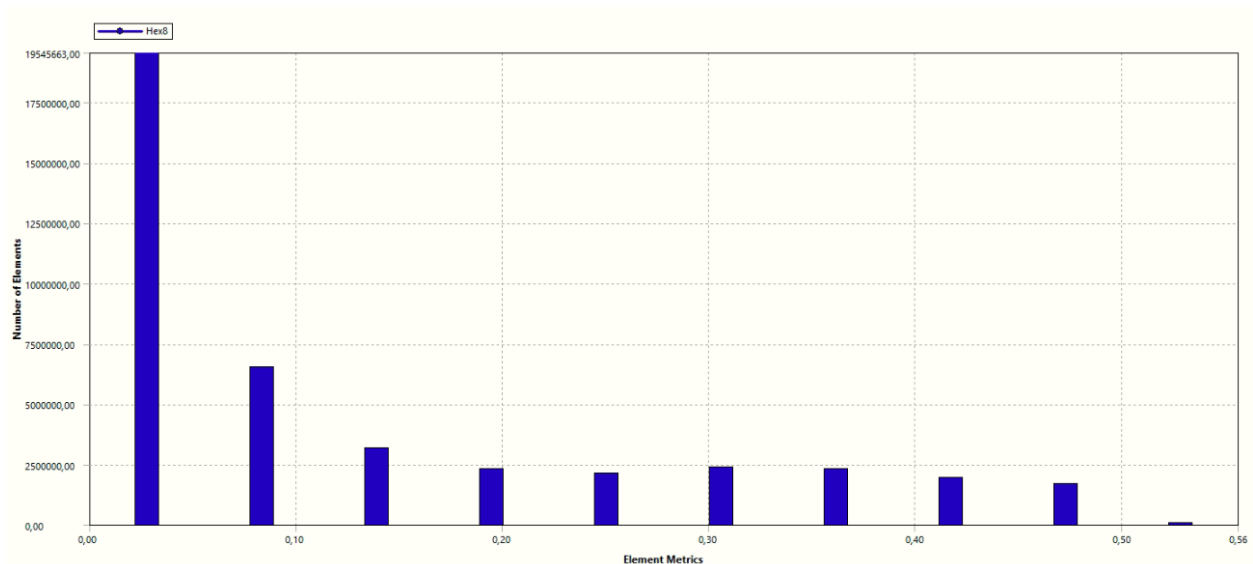


Figure B-10. Mesh skewness for mesh number 3.

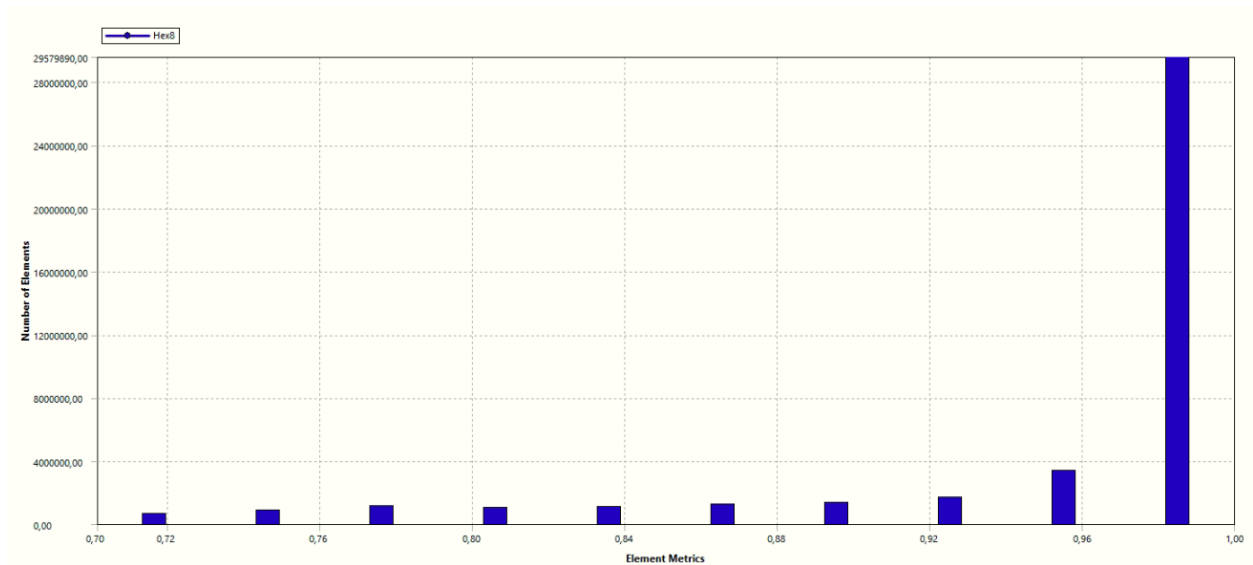


Figure B-11. Mesh orthogonal quality for mesh number 3.

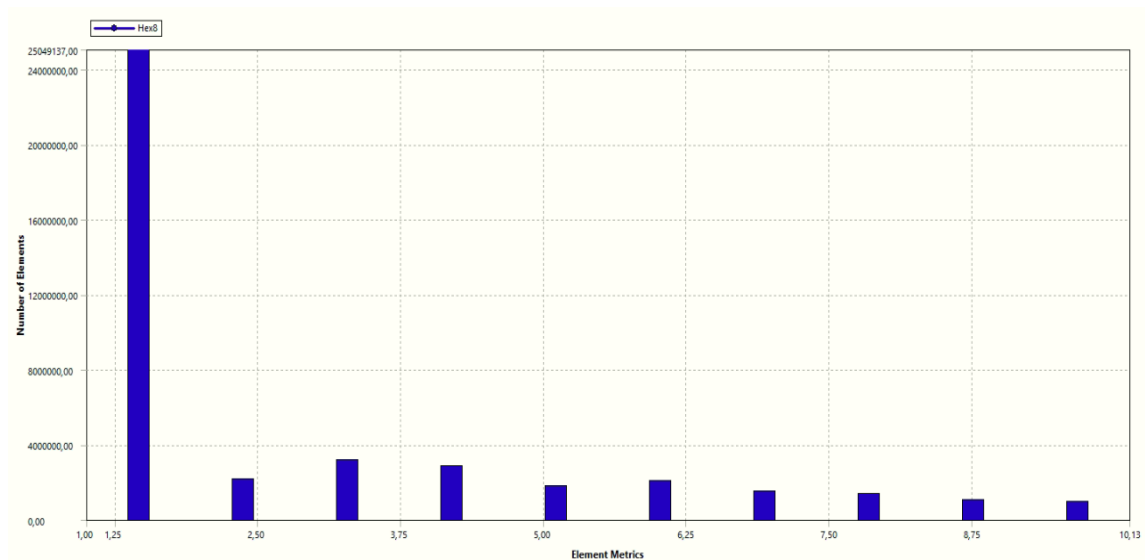


Figure B-12. Cell aspect ratio for mesh number 3.

The wall y^+ values were also checked (and is shown in Figure B-13), with the maximum being below 5. The maximum values were found at the inlet.

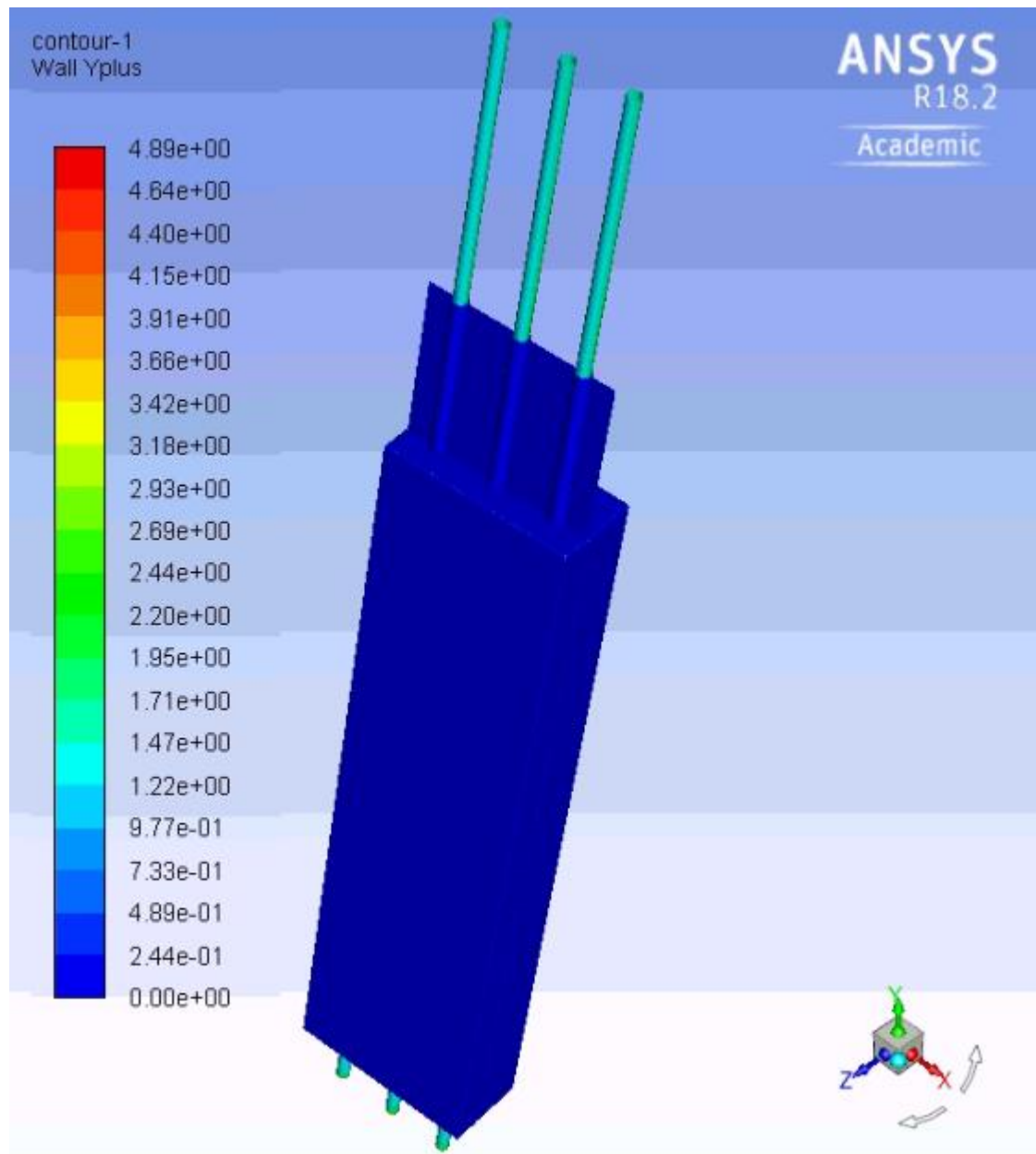


Figure B-13. Contour of wall y+ values for the finest mesh.

B.6 Mesh qualities for the 3D CFD

B.7 Round tubes with fins (section 6.2.1)

The mesh was obtained by specifying a 'proximity and curvature' size function, with 'max face' and 'max tet size' specified at 5 mm. The proximity size function was set to depend on both faces and edges. Inflation layers were added to obtain prism layers on the appropriate surfaces, with a maximum of 8 prism layers, with a growth rate of 1.2 and transition ratio of 0.5. A hexagonal mesh of 15 596 882 nodes and 13 882 460 elements was obtained by this approach. A refinement was added in which the cell size in the solids was decreased such as to obtain at least three cells across the width of the solids. The mesh is shown in Figure B-14.

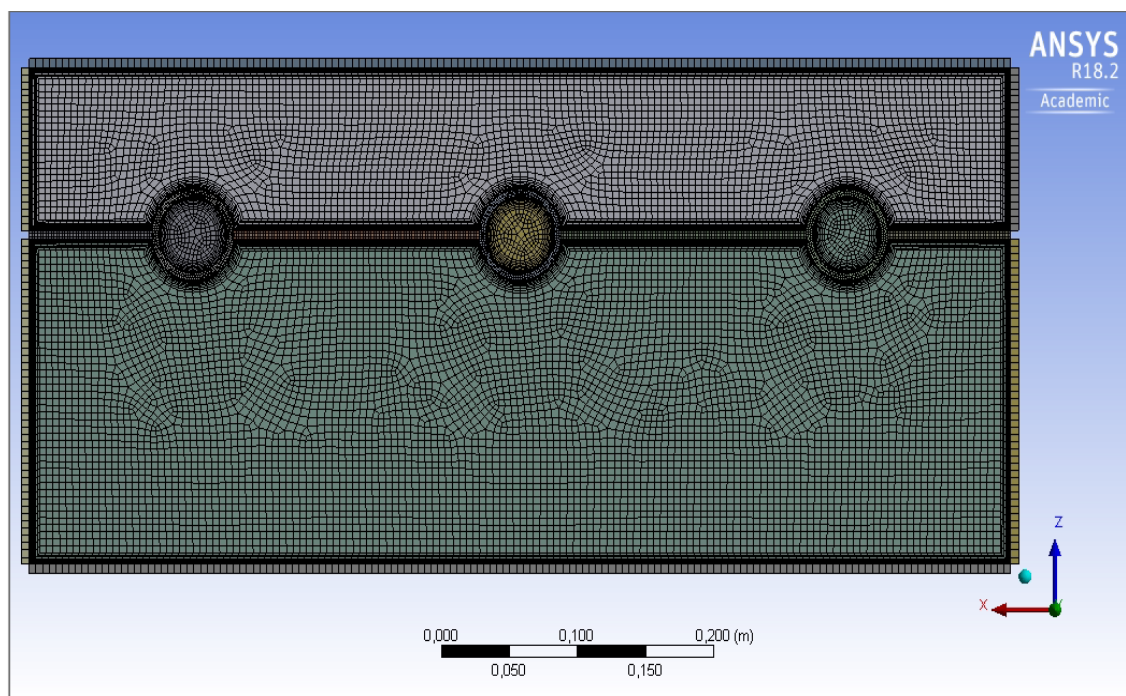


Figure B-14. Mesh used for the round tubes with fins case.

The distribution of the mesh skewness is shown in Figure B-15. The average skewness was 0.15688 with a standard deviation of 0.15489 and minimum and maximum values of 1.3058×10^{-10} and 0.68496 respectively.

Figure B-16 shows the mesh orthogonal quality distribution. The average was 0.93601 with a standard deviation of 9.8282×10^{-2} , minimum of 0.46659 and maximum of 1.

The average aspect ratio was 2.5967 with a standard deviation of 2.3165, with minimum and maximum values of 1.0016 and 14.745 respectively. Figure B-17 shows the cell aspect ratio distribution.

The above parameters fell within the acceptable limits and the mesh was therefore considered of sufficient quality to continue.

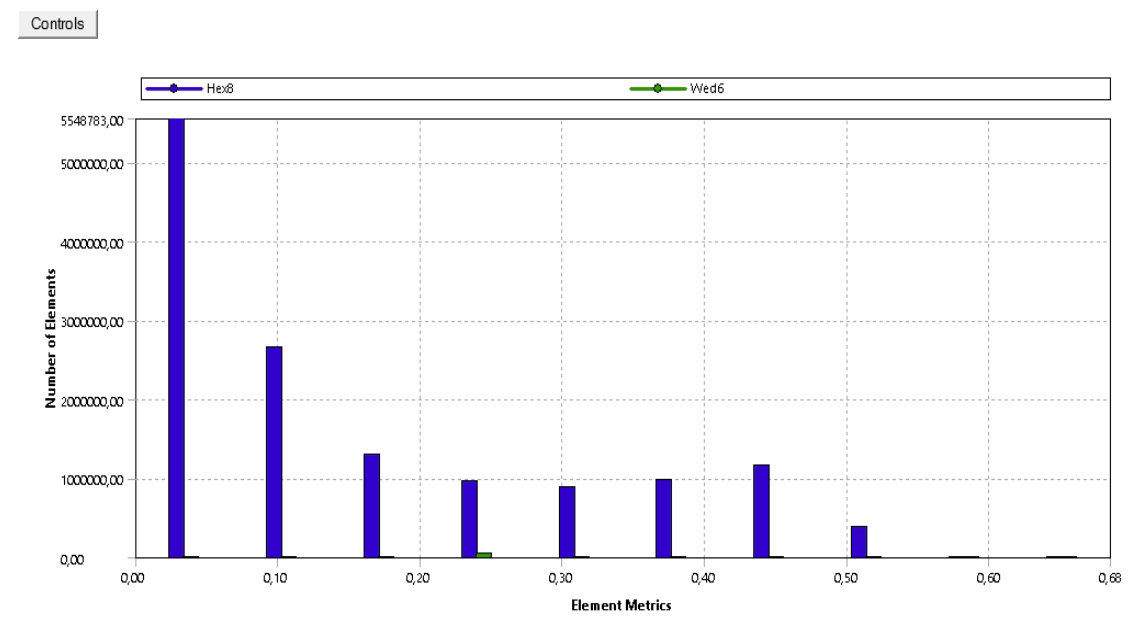


Figure B-15. Mesh skewness for the case with round tubes with fins.

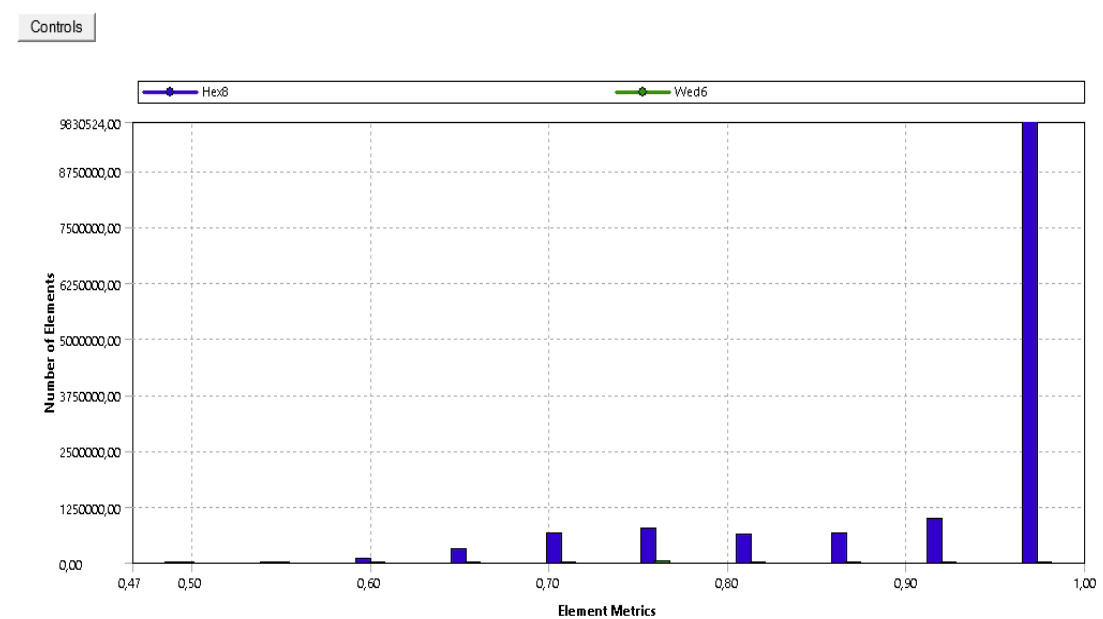


Figure B-16. Mesh orthogonal quality for the case with round tubes with fins.

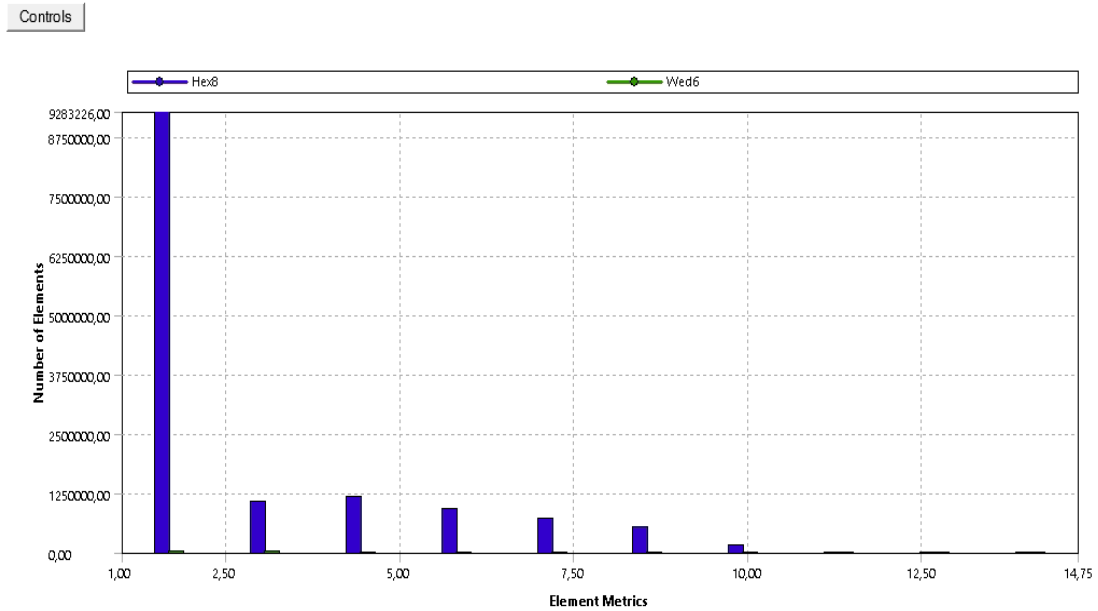


Figure B-17. Cell aspect ratio distribution for the case with round tubes with fins.

B.8 Round tubes without fins (section 6.2.1)

The maximum face and tetrahedron size were set to 5 mm, with the sizing function set around 'proximity and curvature', with 8 prism layers with growth rate of 1.2 and transition ratio of 0.5. The settings yielded a mesh containing 18 582 580 nodes and 15 790 419 elements. The mesh is shown in Figure B-18.

The average skewness was $7.4478e-2$ with a standard deviation of 0.11717 with minimum and maximum values of $1.3093e-10$ and 0.50001 respectively. The cell skewness distribution is shown in Figure B-19.

The average cell orthogonal quality was 0.98049 with a standard deviation of $5.2454e-2$, with minimum and maximum values of 0.70711 and 1 respectively. The orthogonal quality distribution is shown in Figure B-20.

The average aspect ratio was 2.0174 with a standard deviation of 1.9936, with minimum and maximum values of 1 and 10.128 respectively. The aspect ratio distribution is shown in Figure B-21.

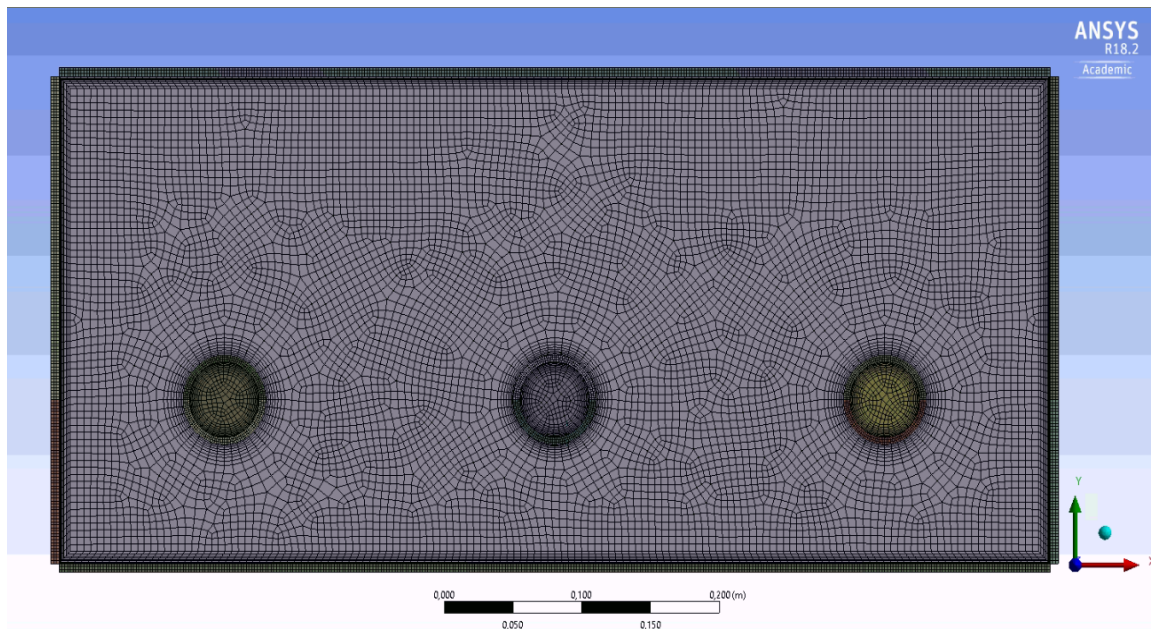


Figure B-18. The mesh generated for the case of the circular riser tubes without fins.

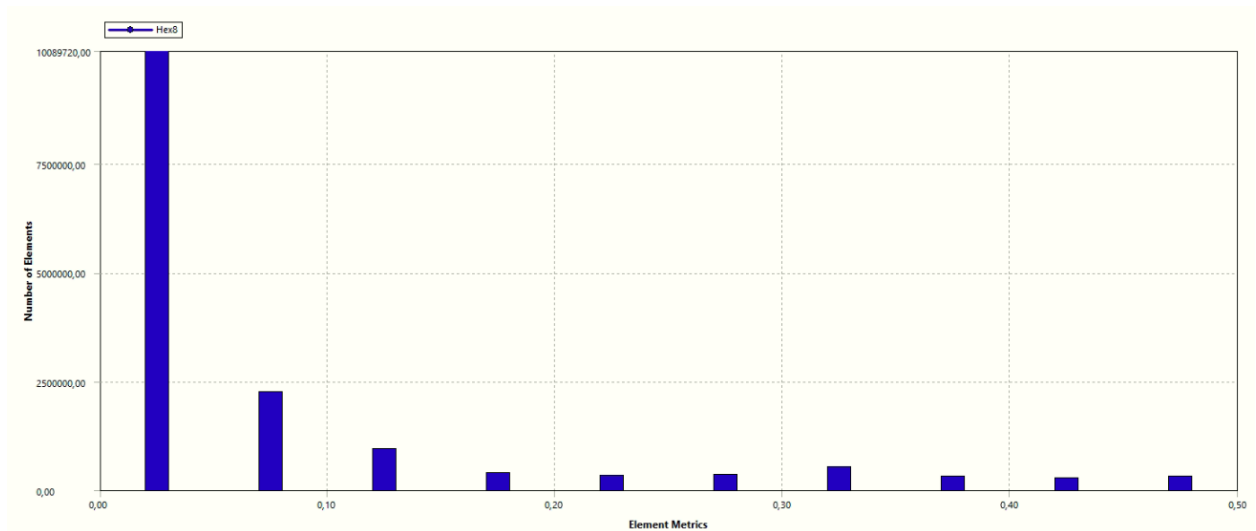


Figure B-19. Cell skewness distribution for the case with round tubes without fins.

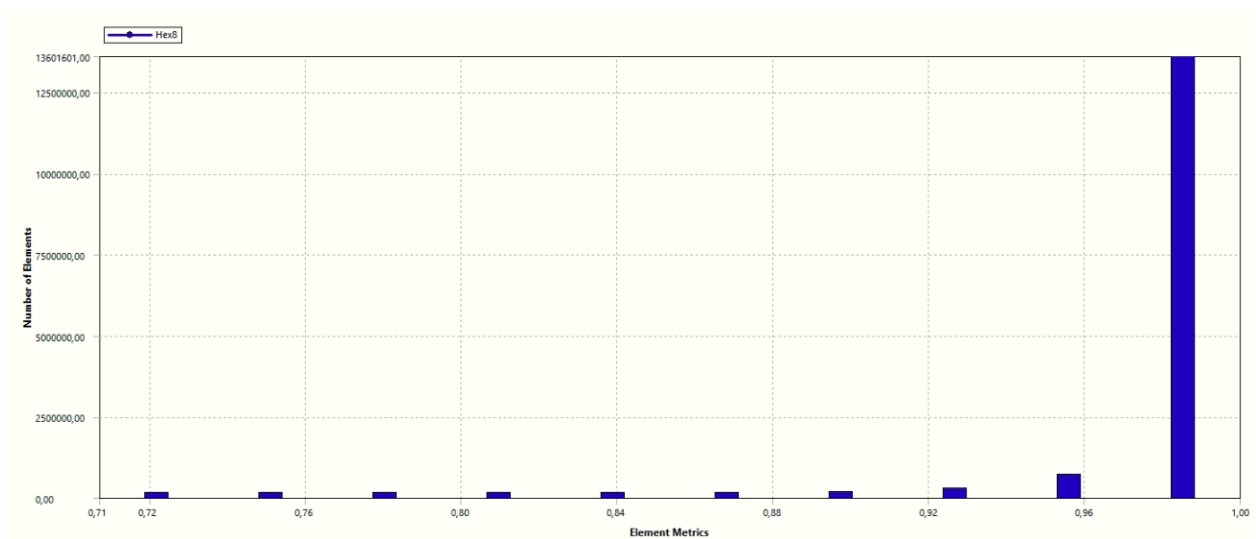


Figure B-20. Cell orthogonal quality distribution for the case round tubes without fins.

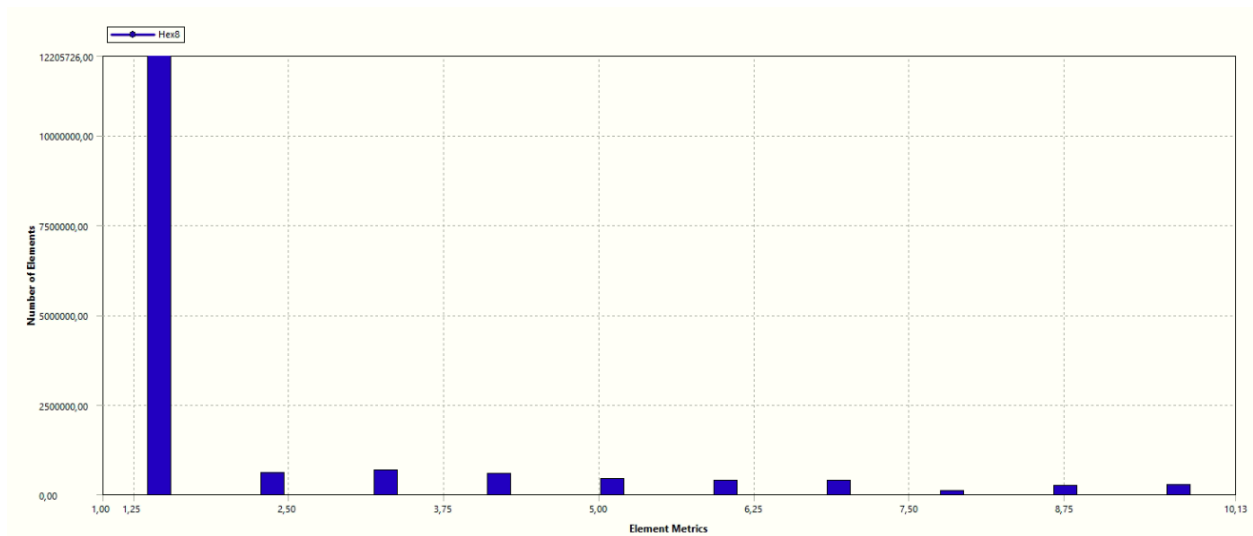


Figure B-21. Cell aspect ratio distribution for the case with round tubes without fins.

B.9 Square tubes with fins (section 6.2.1)

The maximum face and tetrahedron size were set to 5 mm, with sizing function based on proximity and curvature, with 8 prism layers with a growth rate of 1.2 and transition ratio of 0.5. The mesh settings yielded 20 769 904 nodes and 17 639 691 elements. The mesh is shown in Figure B-22.

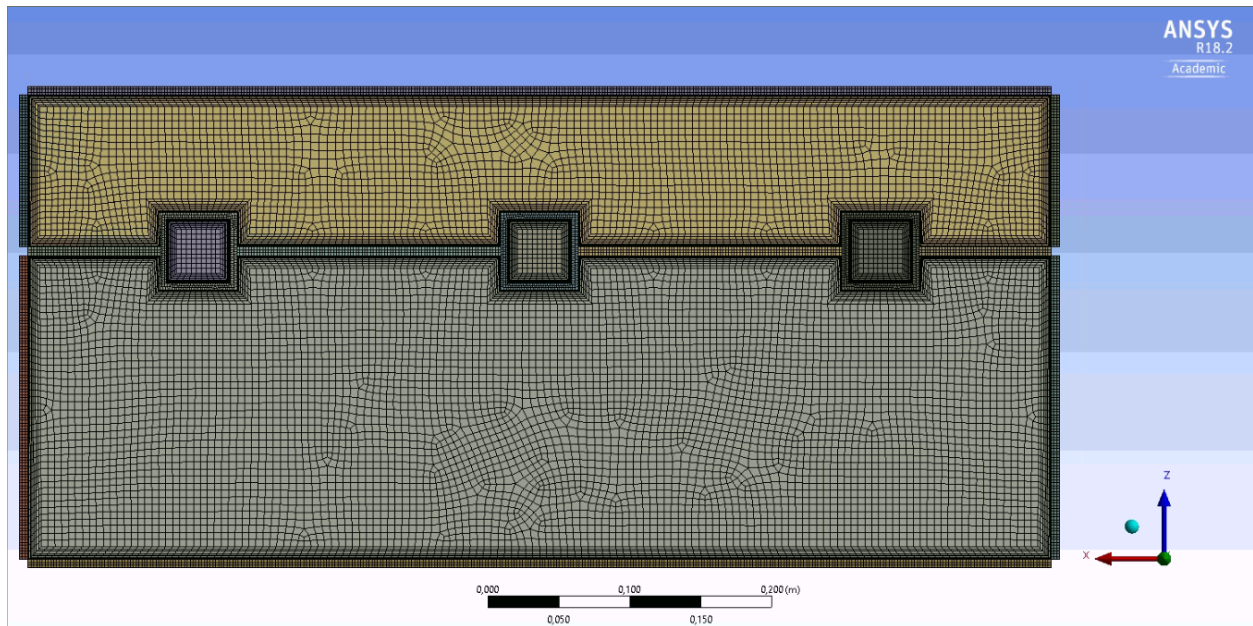


Figure B-22. Mesh for the case with square tubes with fins.

The average skewness was 0.10646 with a standard deviation of 0.15428, with minimum and maximum values of 1.3103e-10 and 0.54571 respectively (see Figure B-23).

The average orthogonal quality was 0.95882 with a standard deviation of 8.1718e-2, with minimum and maximum values of 0.65619 and 1 respectively (see Figure B-24).

The average aspect ratio was 2.1983 with a standard deviation of 2.0631, with minimum and maximum values of 1.0001 and 10.125 respectively (see Figure B-25).

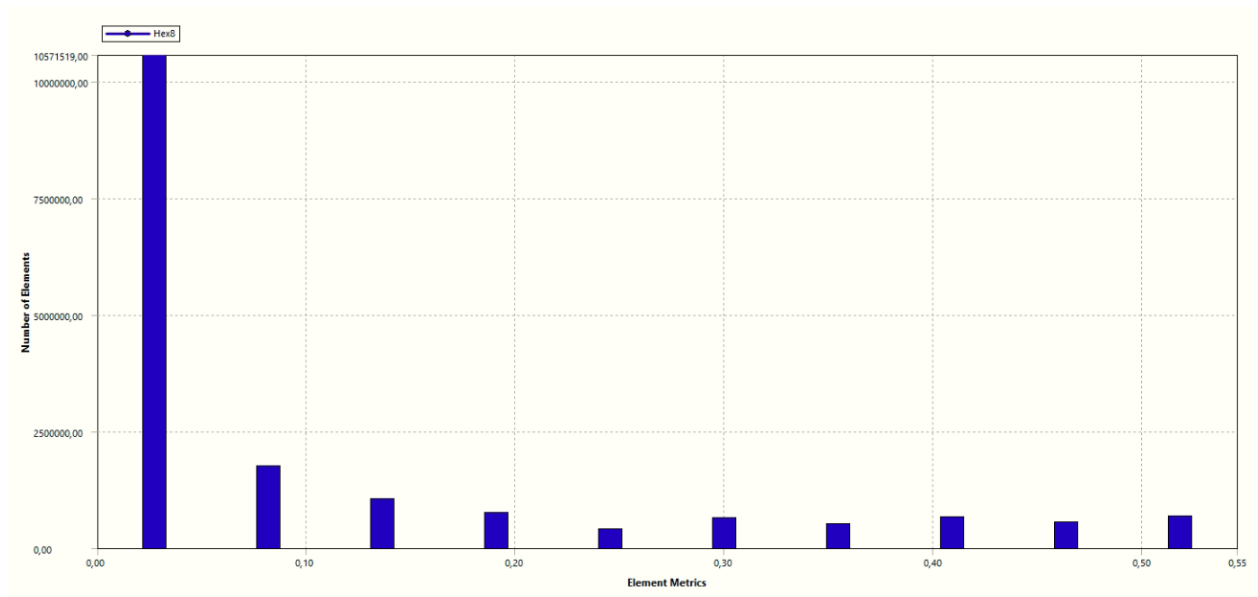


Figure B-23. Skewness distribution for the case with square tubes with fins.

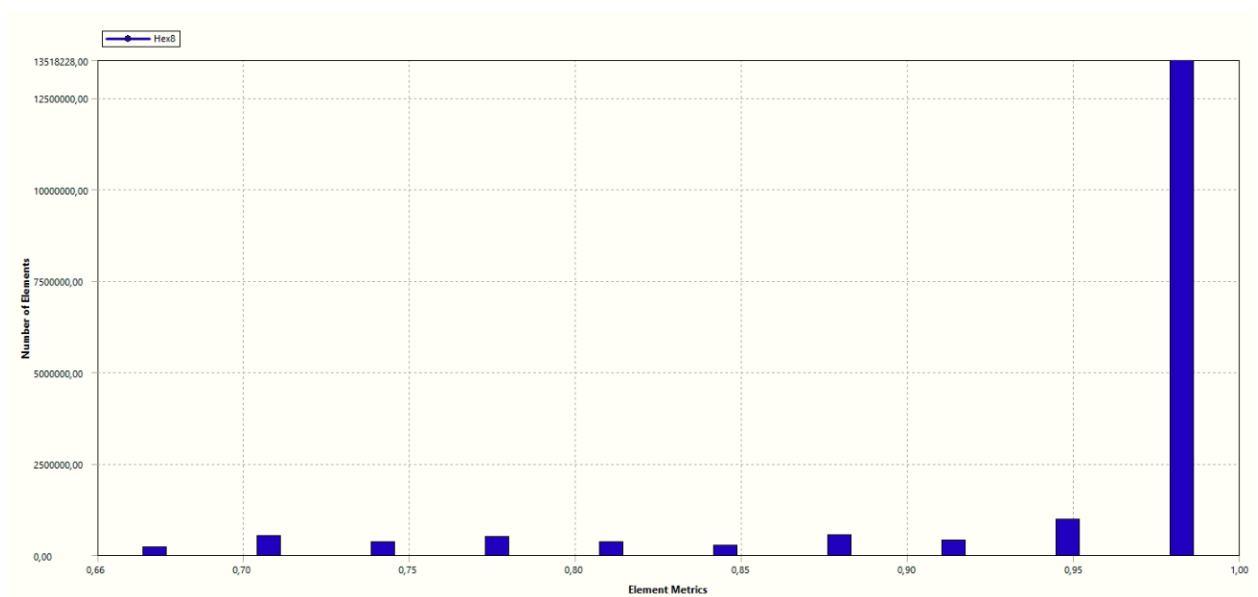


Figure B-24. Orthogonal quality distribution for the case with square tubes with fins.

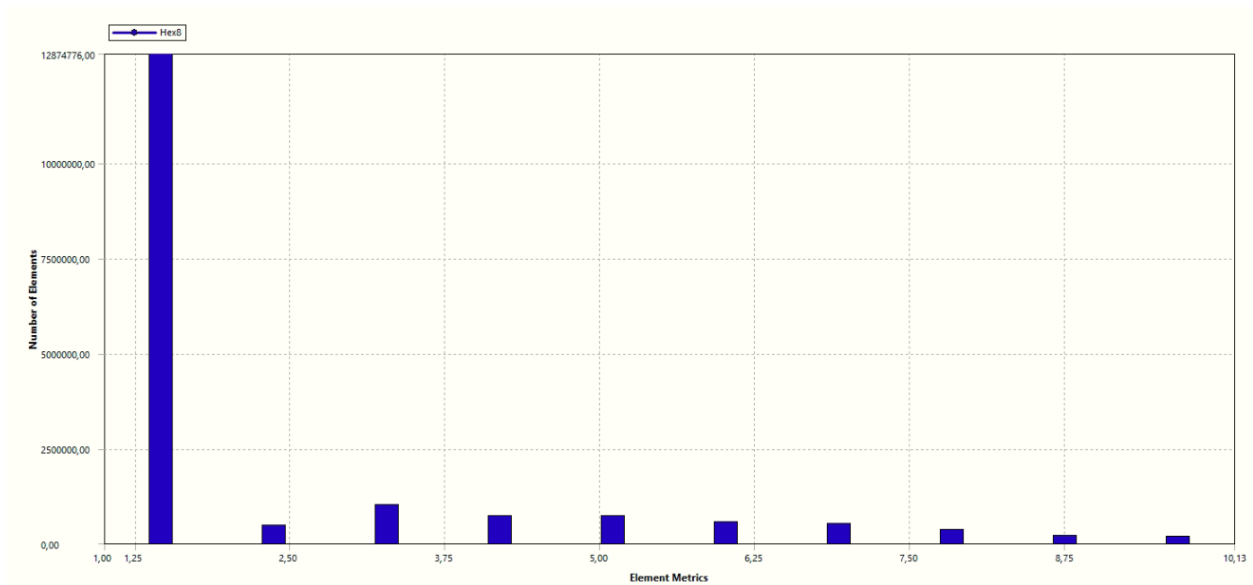


Figure B-25. Cell aspect ratio distribution for the case square tubes with fins.

B.10 Square tubes without fins (section 6.2.1)

The maximum face and tetrahedron sizes were set to 5 mm, with 8 prism layers with transition ratio 0.5 and growth rate of 1.05 for the cavity, with 16 prism layers in the riser tubes with the same transition ratio and growth rate. The mesh settings yielded a mesh containing 13 882 667 nodes and 12 389 080 elements. The mesh is shown in Figure B-26.

In addition to the above settings the maximum skewness had to be improved, which was done by specifying a target skewness of 0.25. Subsequently the average skewness obtained was 0.13074 with a standard deviation of 0.16522, with minimum and maximum values of 1.3058e-10 and 0.50001 respectively (see Figure B-27).

The average orthogonal quality was 0.9473, with a standard deviation of 8.801e-2, with minimum and maximum values of 0.70721 and 1 respectively (see Figure B-28).

The average aspect ratio was 2.3823 with a standard deviation of 1.9599, with minimum and maximum values of 1.0003 and 7.4005 respectively (Figure B-29).

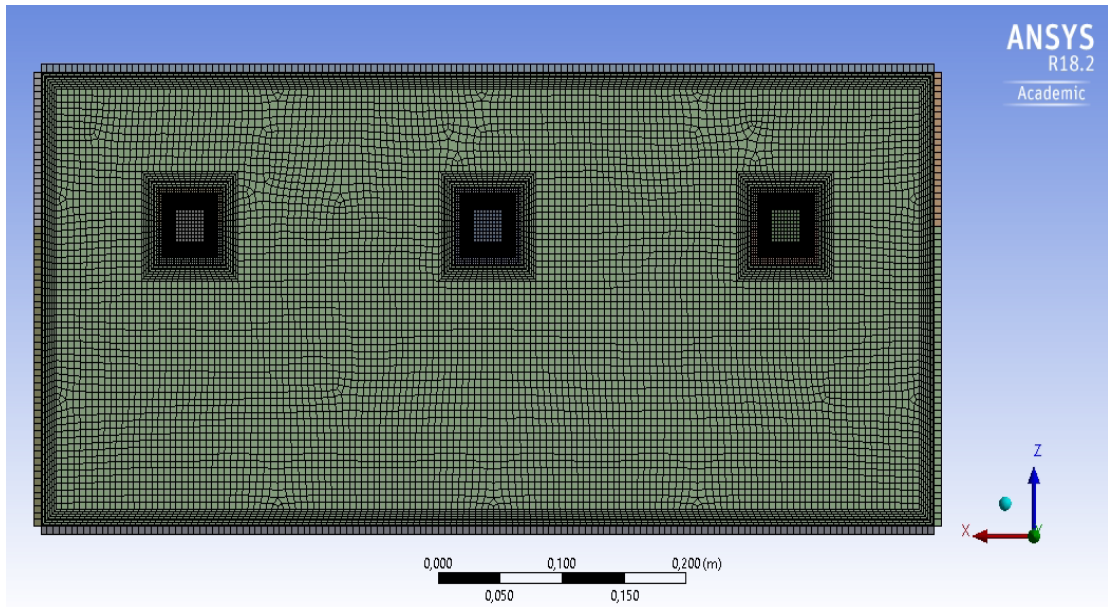


Figure B-26. Mesh generated for the case square tubes without fins.

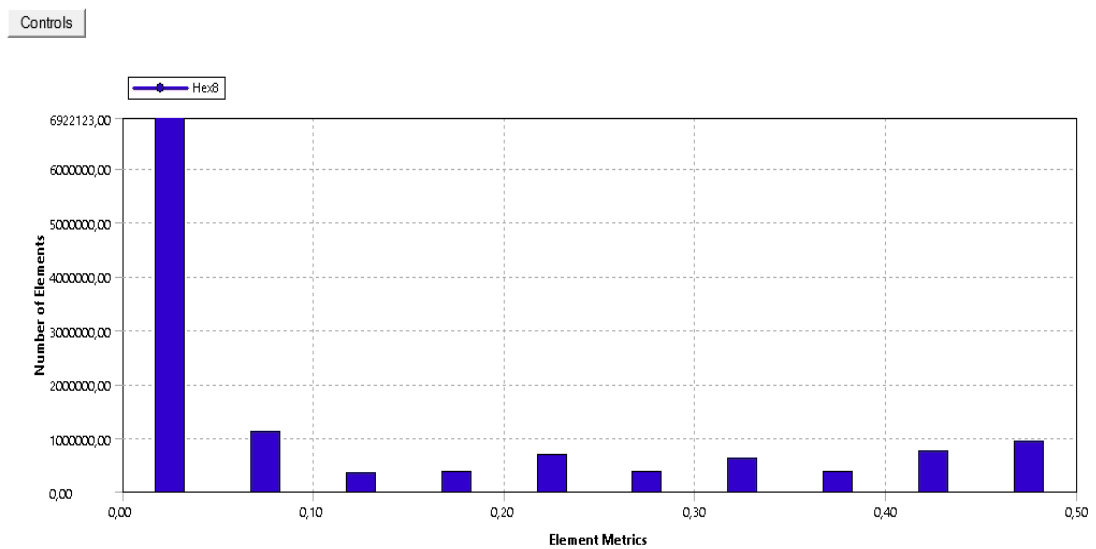


Figure B-27. Cell skewness distribution for the case with square tubes without fins.

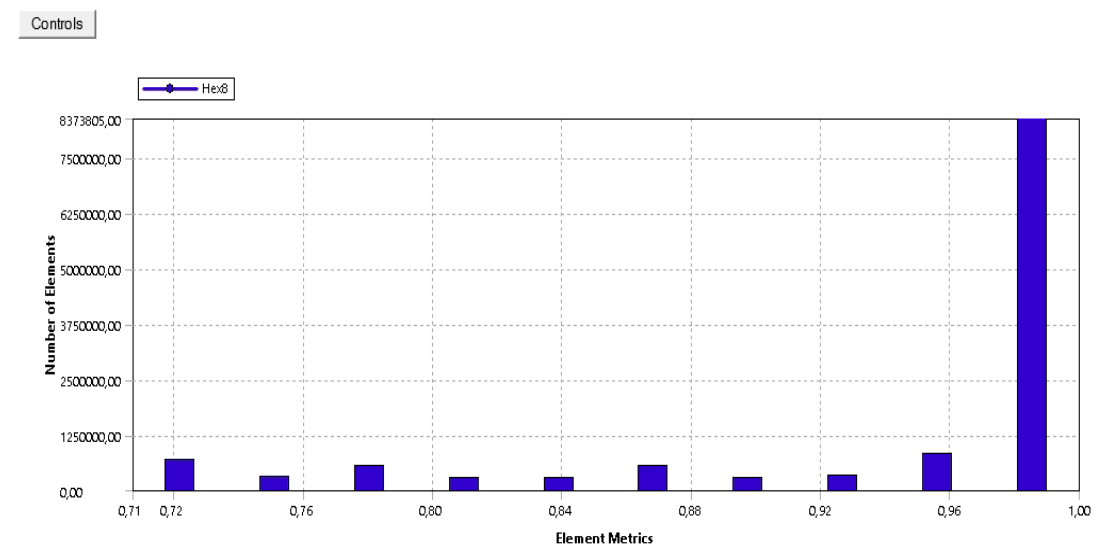


Figure B-28. Orthogonal quality distribution for the case with square tubes without fins.

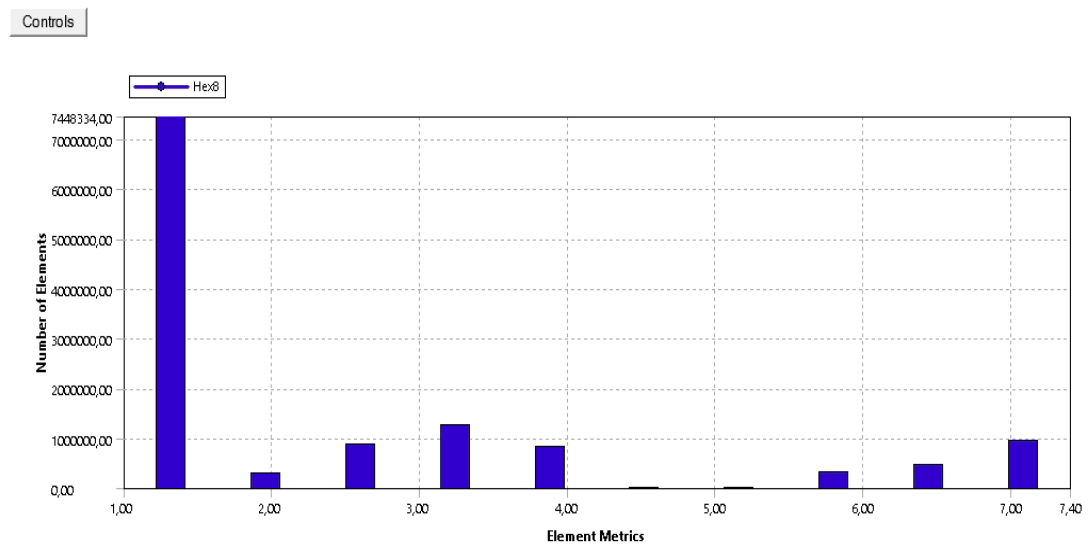


Figure B-29. Cell aspect ratio distribution for the case square tubes without fins.

B.11 Rectangular tubes with fins (section 6.2.1)

The mesh generated for the case with 6 rectangular riser tubes is shown in Figure B-30. The maximum face and tetrahedral size was set to 5 mm, with 16 prism layers in the riser tubes and 8 layers in the cavity with a transition ratio of 0.5 and growth rate of 1.05. The mesh contained 21 736 039 nodes and 18 904 545 elements.

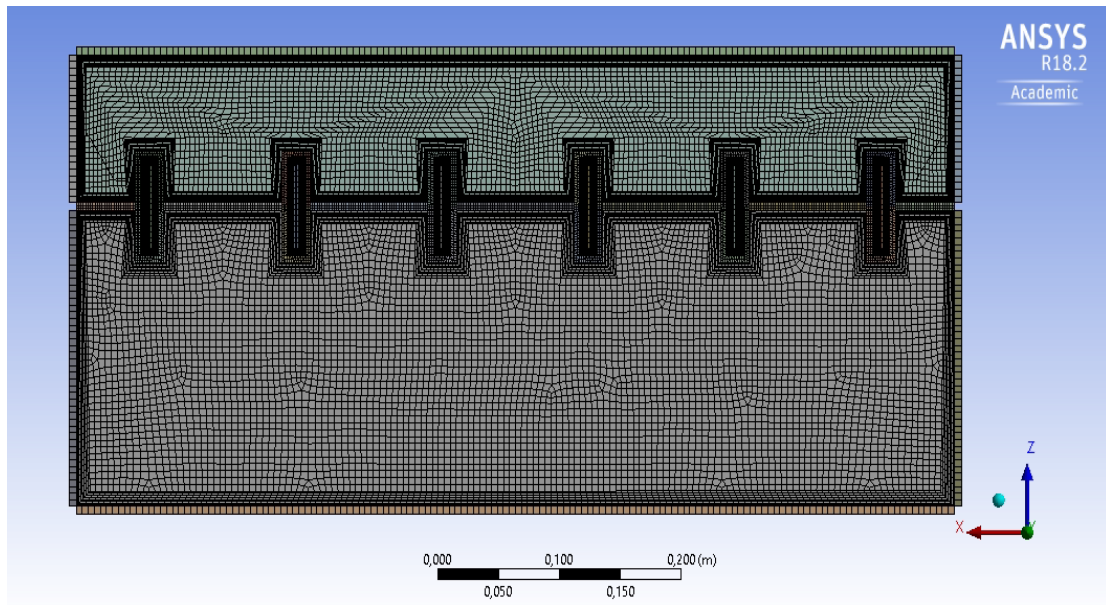


Figure B-30. The mesh generated for the case with rectangular tubes with fins.

The average cell skewness (Figure B-31) was 0.19075 with a standard deviation of 0.18675 with minimum and maximum values of 1.3065e-10 and 0.96591.

The average cell orthogonal quality (Figure B-32) was 0.91232 with a standard deviation of 0.11706 with minimum and maximum values of 0.25726 and 1.

The average aspect ratio (Figure B-33) was 4.4132 with standard deviation of 5.0467 with minimum and maximum values of 1.0017 and 19.752.

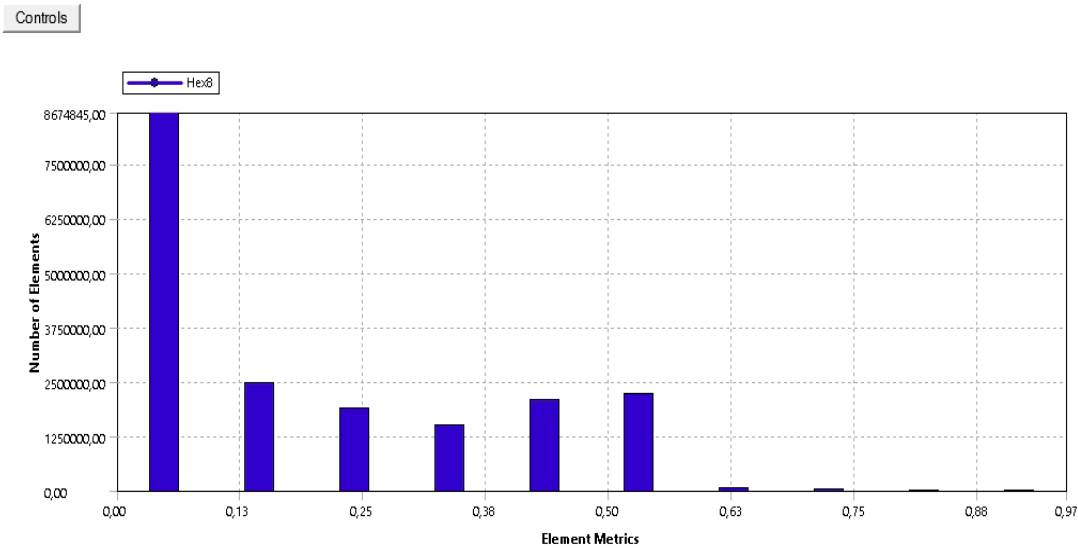


Figure B-31. Skewness distribution for the case with rectangular tubes with fins.

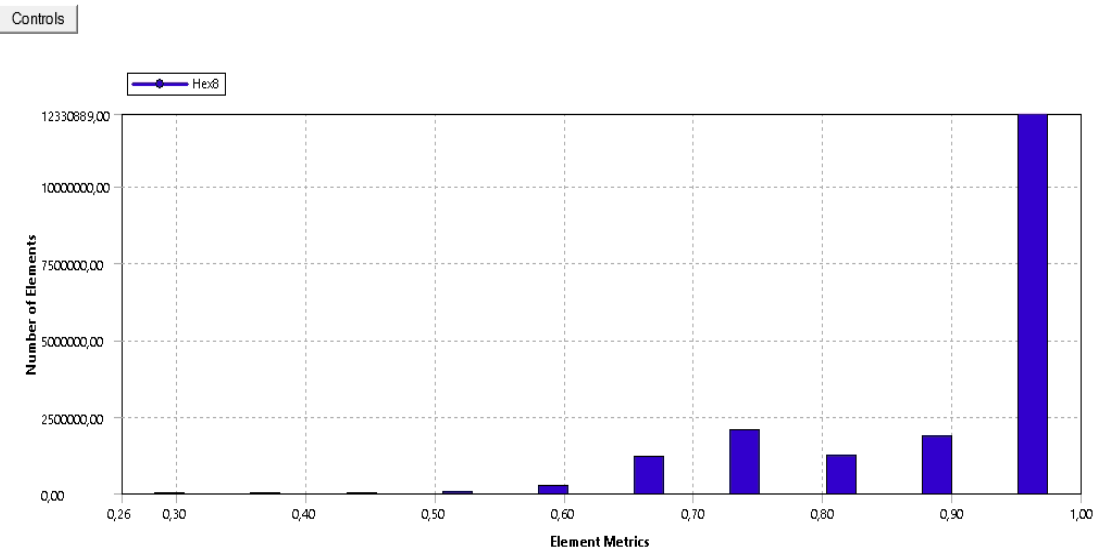


Figure B-32. Orthogonal quality distribution of the case with rectangular tubes with fins.

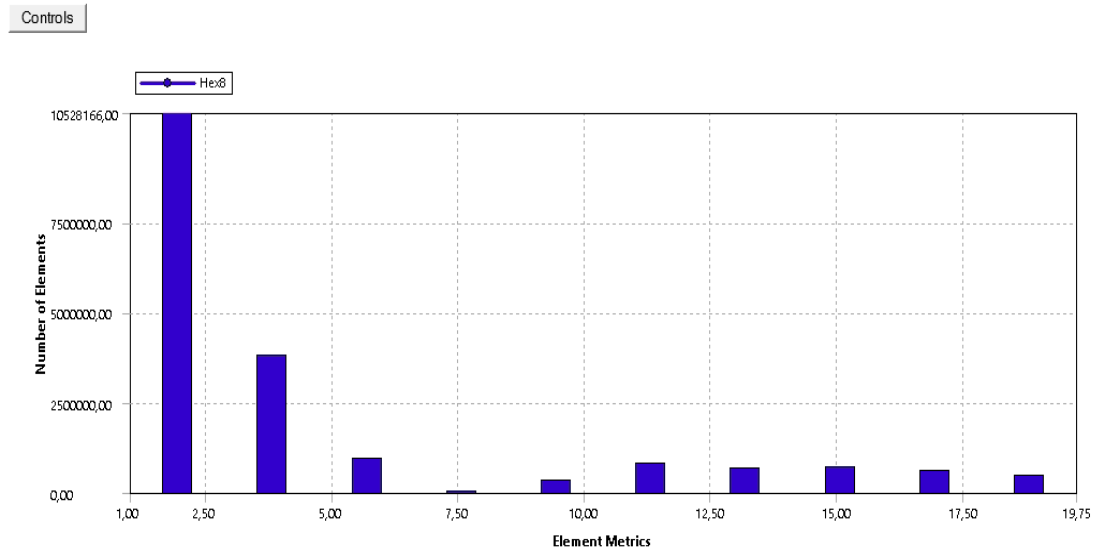


Figure B-33. Aspect ratio distribution of the case with rectangular tubes with fins.

B.12 Rectangular tubes without fins (section 6.2.1)

The mesh settings resulted in the generation of a mesh containing 19 027 507 nodes and 16 798 015 elements. The mesh is shown in Figure B-34.

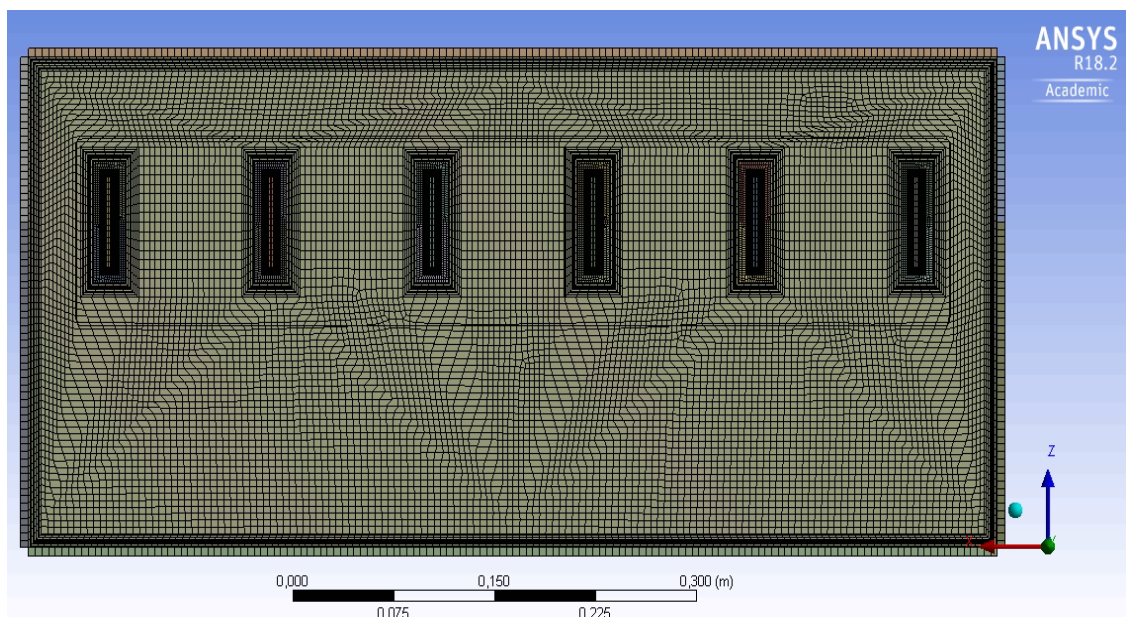


Figure B-34. The mesh generated for the case with rectangular tubes without fins.

The average cell skewness was 0.20167 with a standard deviation of 0.17809 with minimum and maximum values of 1.3064e-10 and 0.88161. The cell skewness distribution is shown in Figure B-35.

The average orthogonal quality was 0.90856 with a standard deviation of 0.11244 with minimum and maximum values of 0.24364 and 1. The orthogonal quality distribution is shown in Figure B-36.

The average aspect ratio was 4.6786 with a standard deviation of 5.2381 with minimum and maximum values of 1.0084 and 19.752. The aspect ratio distribution is shown in Figure B-37.

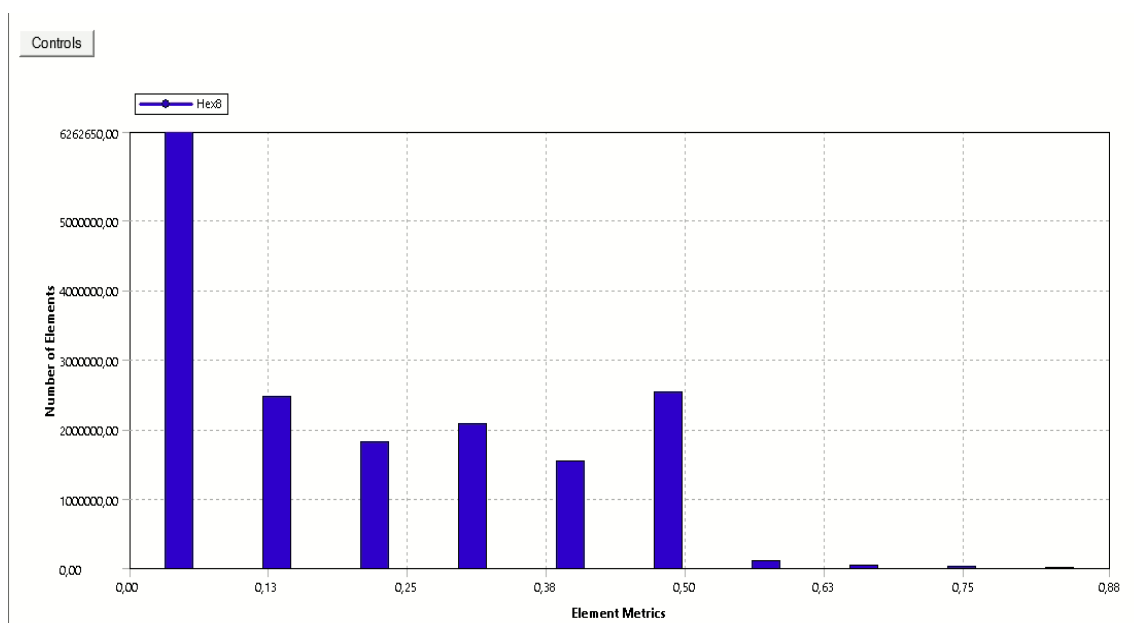


Figure B-35. Cell skewness distribution for the case with rectangular tubes without fins.

Controls

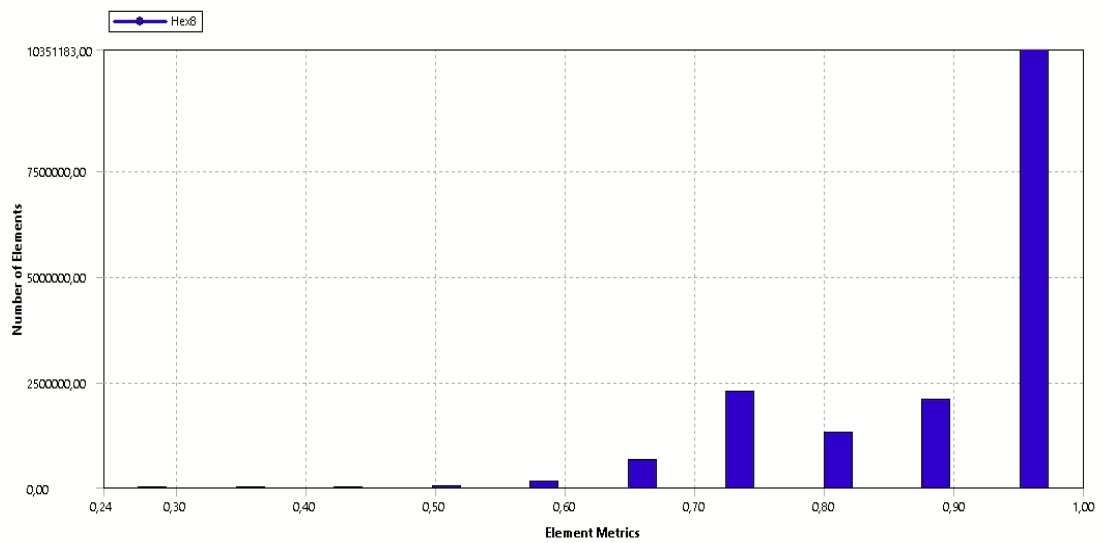


Figure B-36. Orthogonal quality distribution for the case rectangular tubes without fins.

Controls

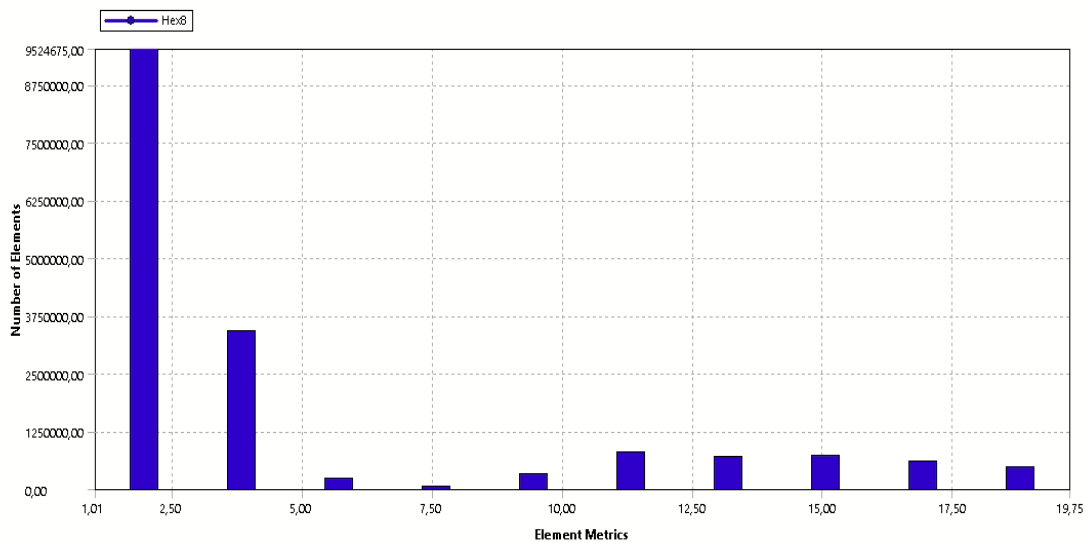


Figure B-37. Aspect ratio distribution for the case with rectangular tubes without fins.

The mesh was considered to be of adequate quality based on the above parameters.

B.13 Mesh qualities for the coupled systems

The mesh qualities for the coupled simulations discussed in section 7.3 and 7.4 are discussed in B.14 and B.15.

B.14 Variation B (section 7.3.2)

The mesh contained 20 755 696 nodes and 18 728 050 elements. The skewness was monitored and is shown in Figure B-38. The average skewness was 0.2222 with a standard deviation of 0.15428. The minimum and maximum skewness were 1.342e-10 and 0.83549 respectively.

The mesh orthogonal quality was also monitored as a measure of quality assurance. The distribution of mesh quality obtained is shown in Figure B-39. The average was 0.9021 with a standard deviation of 0.10817. The minimum and maximum was 0.31628 and 1 respectively.

The maximum aspect ratio was 26.188, with a minimum value of 1.0033. The average and standard deviation of the aspect ratio was 3.4455 and 3.5541 respectively. The distribution is shown in Figure B-40.

The mesh quality was considered as adequate based on the above values, since most of the cells were within acceptable criteria.

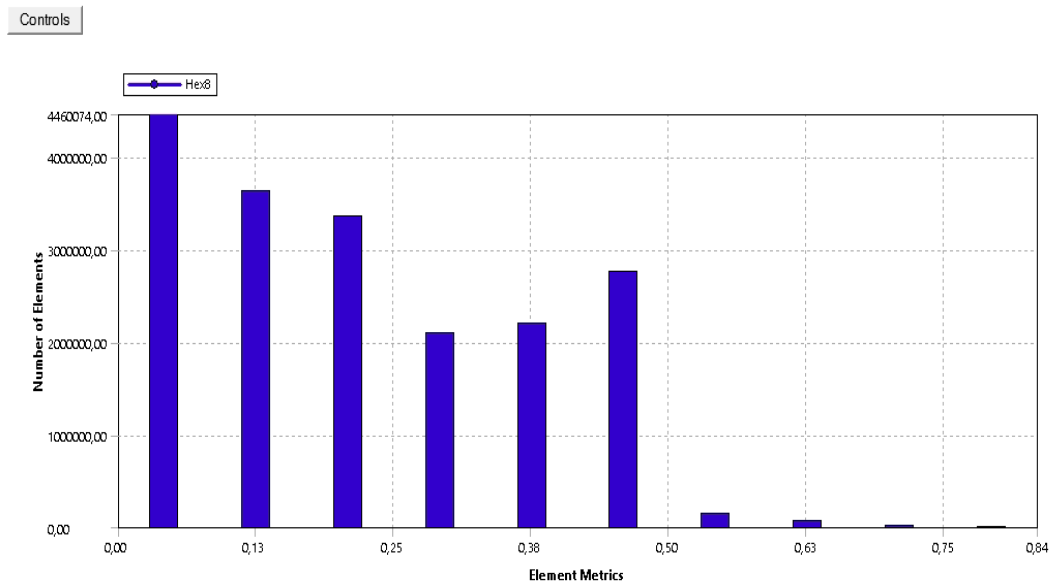


Figure B-38. Mesh skewness for the mesh generated for variation B.

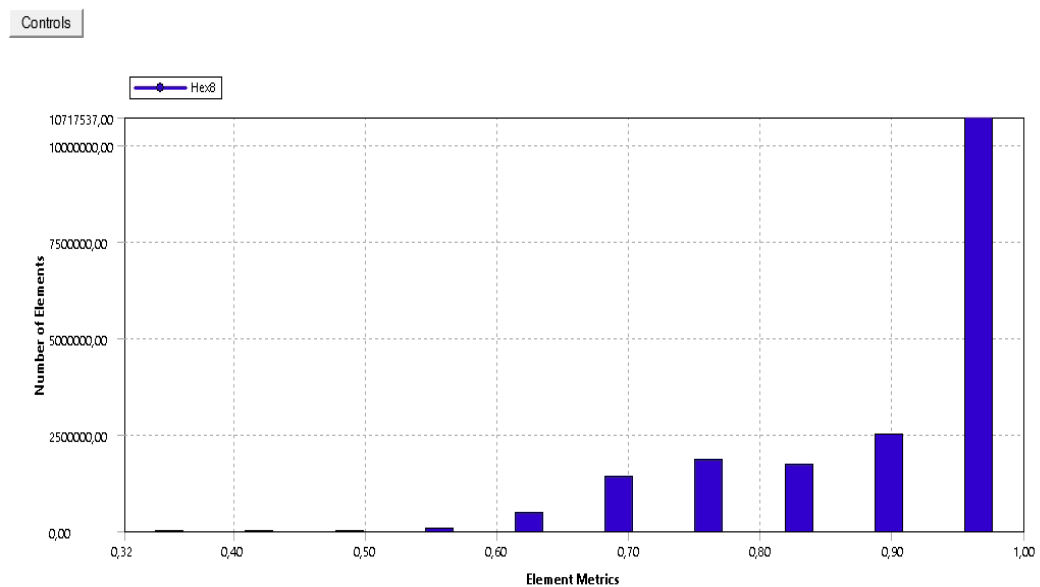


Figure B-39. Mesh orthogonal quality for the mesh generated for variation B.

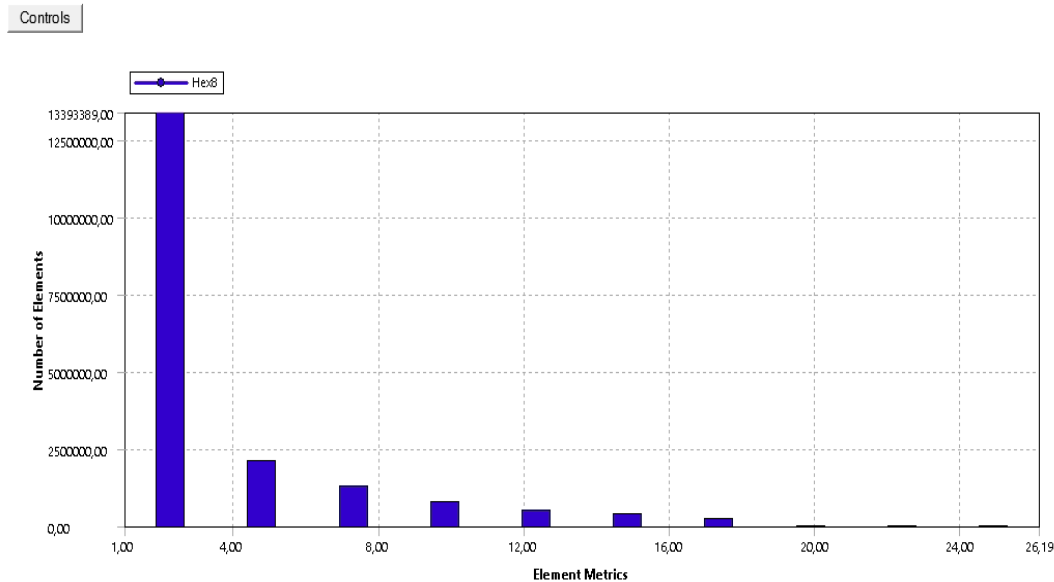


Figure B-40. Aspect ratio distribution for the mesh generated for Variation B.

B.15 Variation C (section 7.4.2)

The maximum face and tetrahedron sizes were set to 4 mm, with the size function set to proximity and curvature, which resulted in a mesh containing 17 616 314 nodes and 15 592 618 elements. 16 inflation layers were used within the riser tubes, with a transition ratio of 0.5 and a growth rate of 1.05. The front cavity and back cavity had 8 inflation layers with a transition ratio of 0.5 and growth rate of 1.05.

The average and standard deviation of the mesh skewness was 0.1493 and 0.14731 with minimum and maximum values of 1.3418e-10 and 0.67931 respectively. The full distribution is shown in Figure B-41.

The average and standard deviation of the mesh orthogonal quality was 0.9434 and 9.3827e-2 with minimum and maximum values of 0.46661 and 1 respectively. The distribution of orthogonal quality is shown in Figure B-42.

The average and standard deviation of the aspect ratio was 2.4496 and 1.87. The minimum and maximum values were 1.0039 and 9.1354. The distribution of aspect ratio of the mesh elements is shown in Figure B-43.

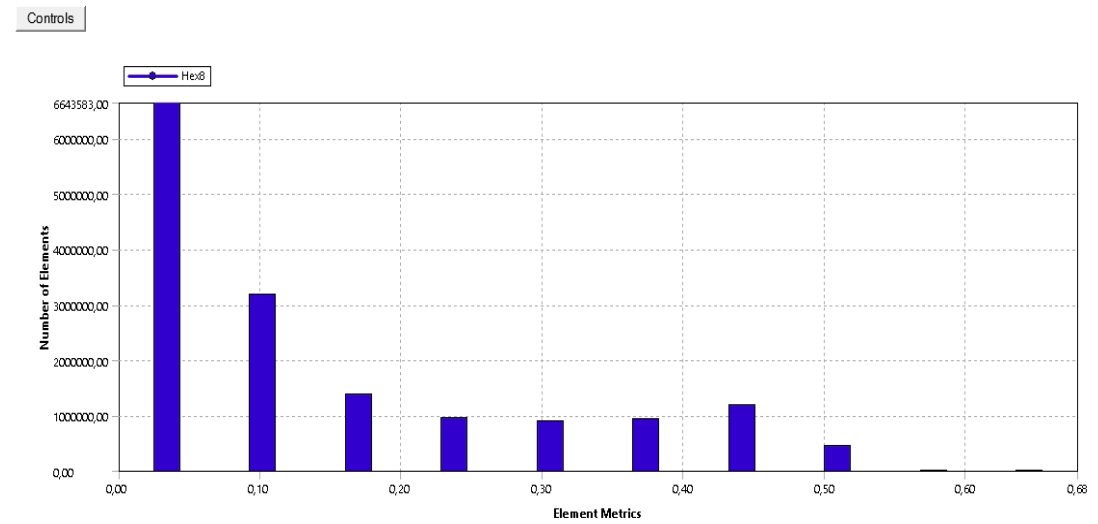


Figure B-41. Mesh skewness distribution for Variation C.

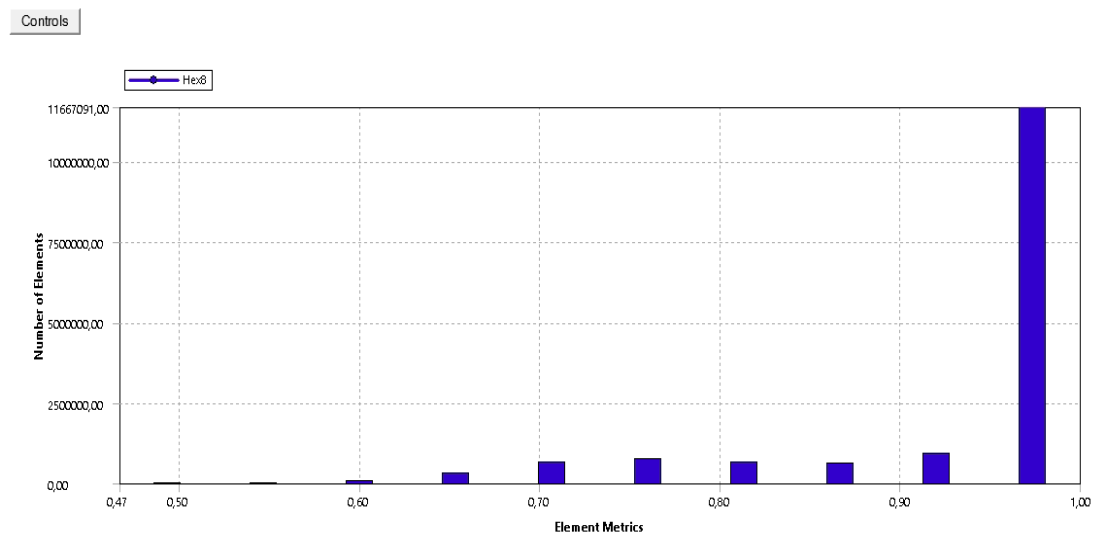


Figure B-42. Orthogonal quality for the mesh for Variation C.

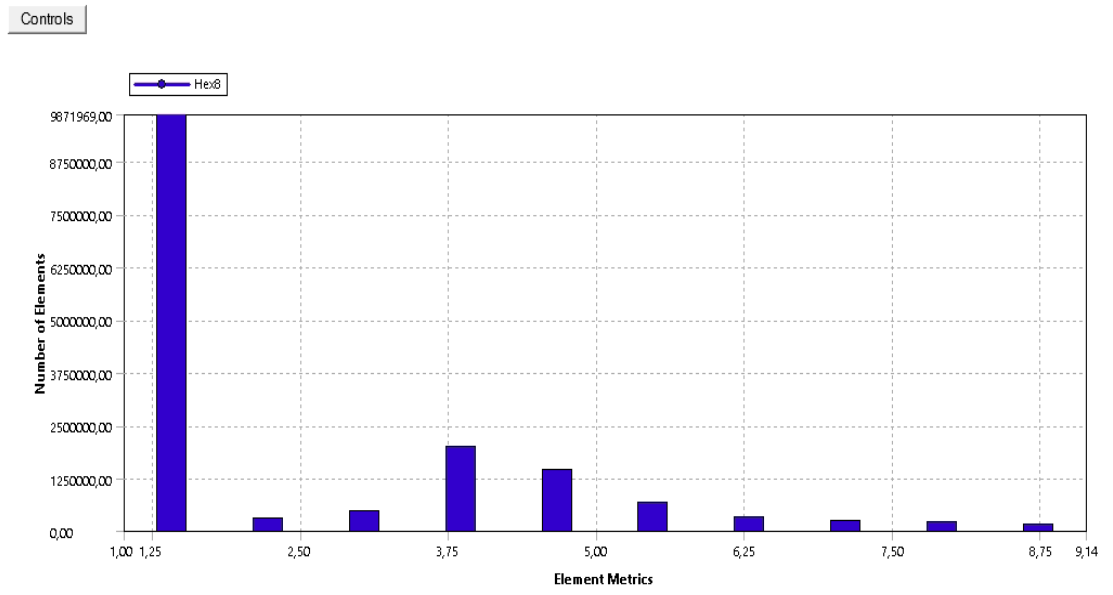


Figure B-43. Aspect ratio distribution for variation C.

Since the majority of the cells fall within the criteria limits the mesh quality was considered as adequate.

APPENDIX C SOLUTION MONITORING

C.1 Introduction

The simulations were run until the specified convergence criteria were reached. In general, the criteria were that the residuals should be below 10^{-4} and the water outlet temperature had stabilized. The residuals and the monitored parameters for the work that was simulated is presented in this chapter.

C.2 Isothermal water region (section 5.1.3)

The figures below show the solution monitoring for the case (discussed in 5.1.3) where the water region (manifolds and three riser tubes) was modelled without heat input. The mass flow rate was specified at the inlet and the outlet pressure was also specified. Five cases were simulated, namely with a mass flow rate of 0.37 kg/s, 0.45 kg/s, 0.52 kg/s, 0.58 kg/s and 0.67 kg/s, corresponding to the cases examined experimentally by Lisowski (2013). The residuals for the case 0.58 kg/s was shown during the verification chapter and is therefore not repeated here. The case 0.67 kg/s was chosen to indicate the difference in the residual criteria for the standard k- ϵ (Figure C-4) and k- ω (Figure C-5). The criteria reached values below 10^{-4} for the k- ϵ model but not for the k- ω model, which still had values above 10^{-3} after the same number of iterations and with the same mesh. All cases show stable operation with gradual decreases in the residuals.

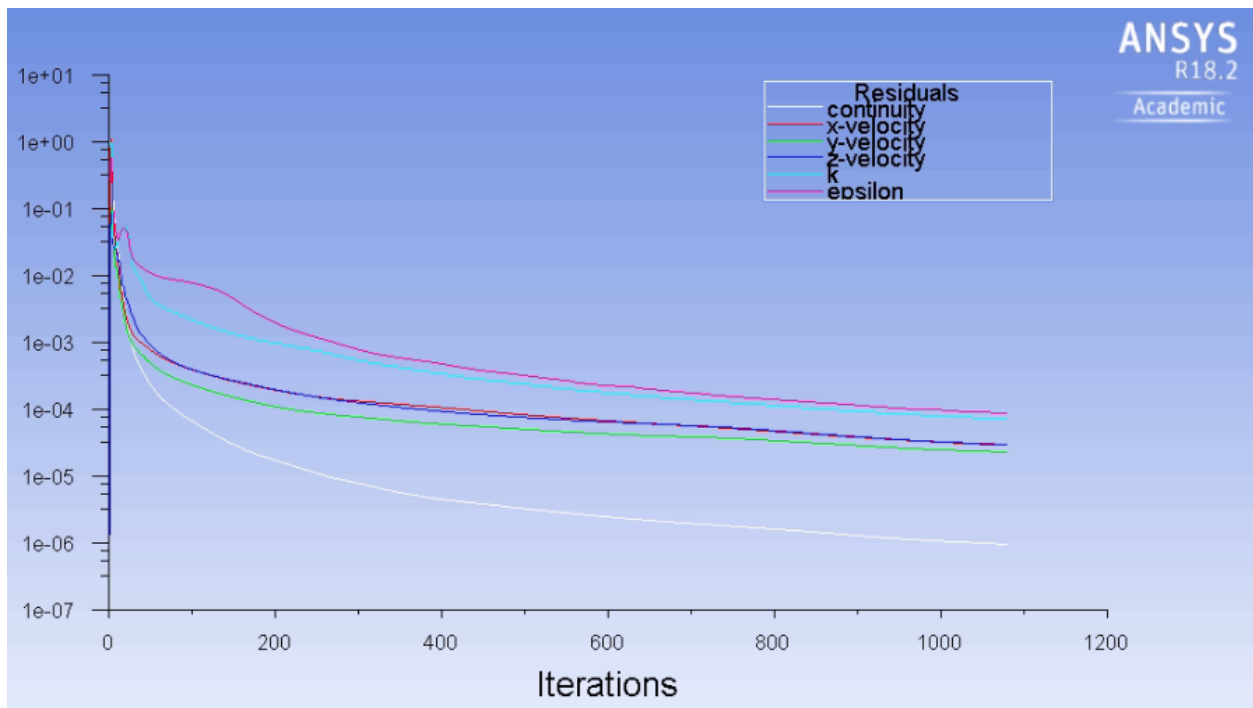


Figure C-1. Residual for 0.37 kg/s case (k-ε turbulence).

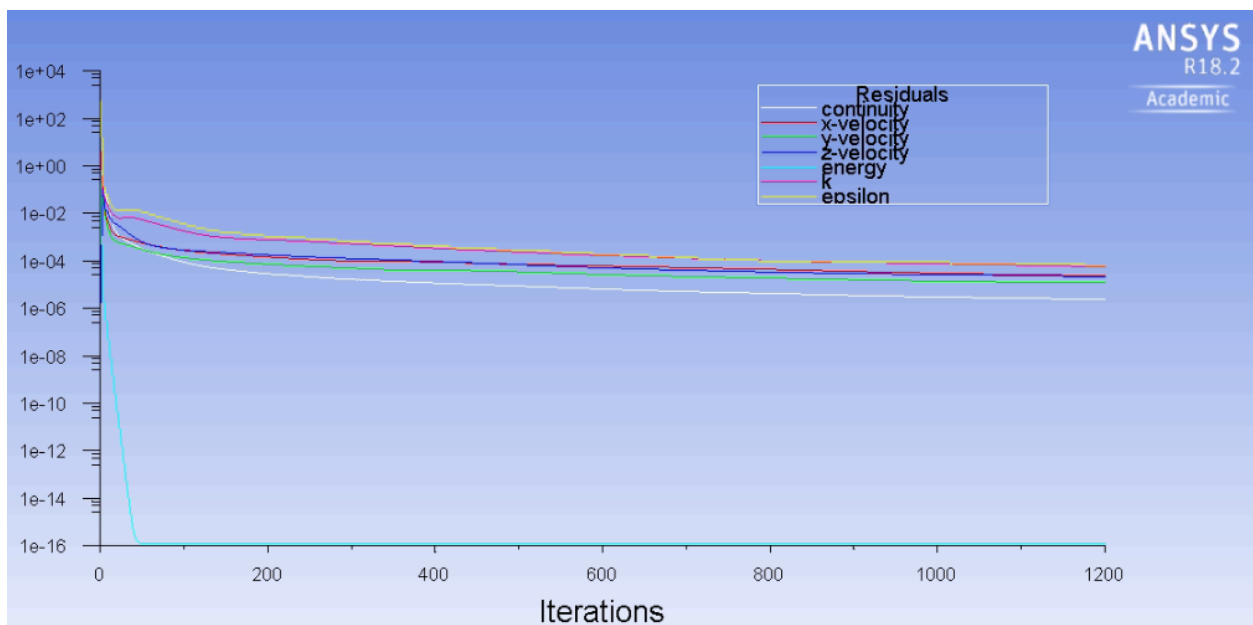


Figure C-2. Residuals for the 0.45 kg/s case (k-ε turbulence).

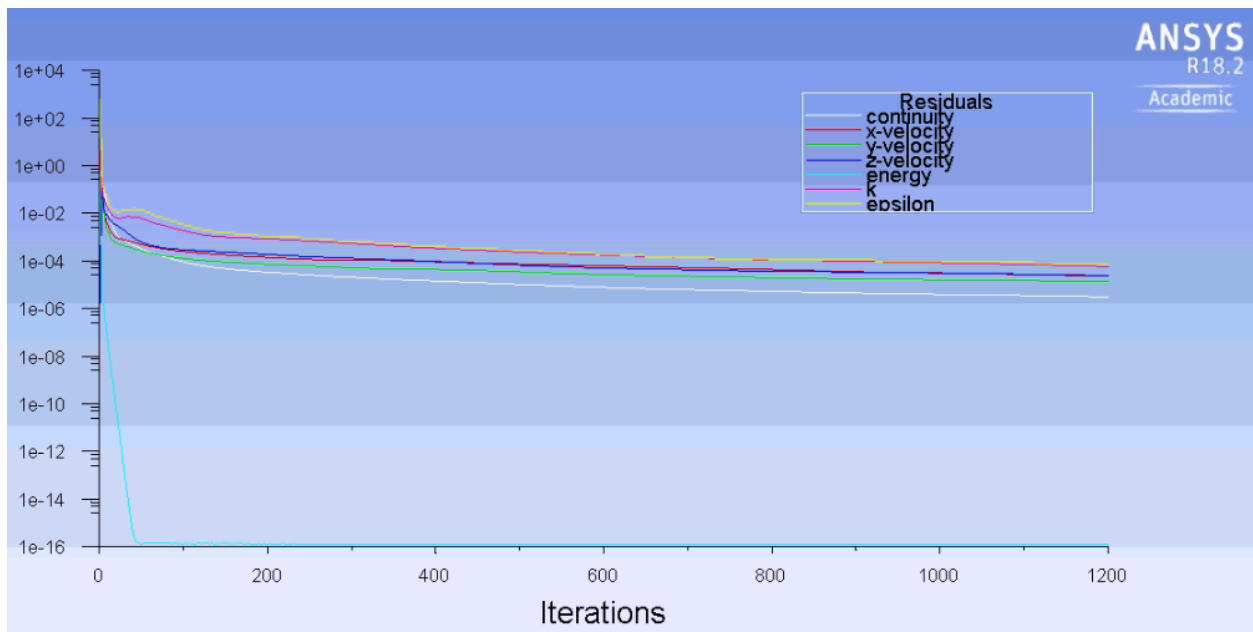


Figure C-3. Residuals for the 0.52 kg/s case (k- ϵ turbulence).

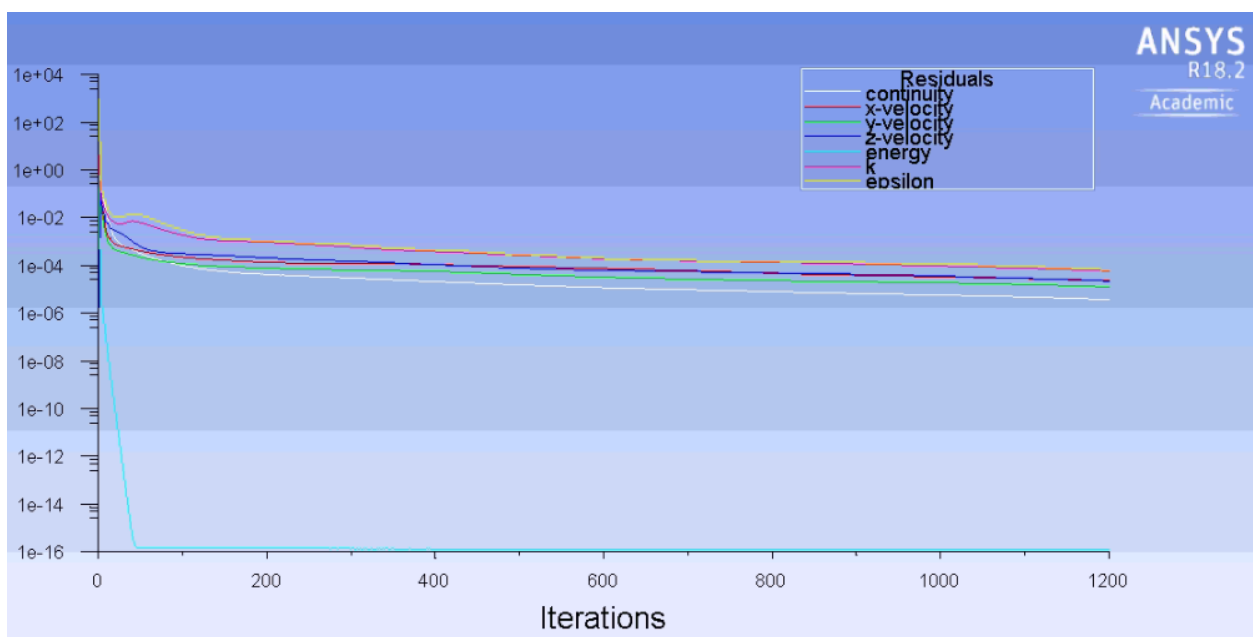


Figure C-4. Residuals for the 0.67 kg/s case (k- ϵ turbulence).

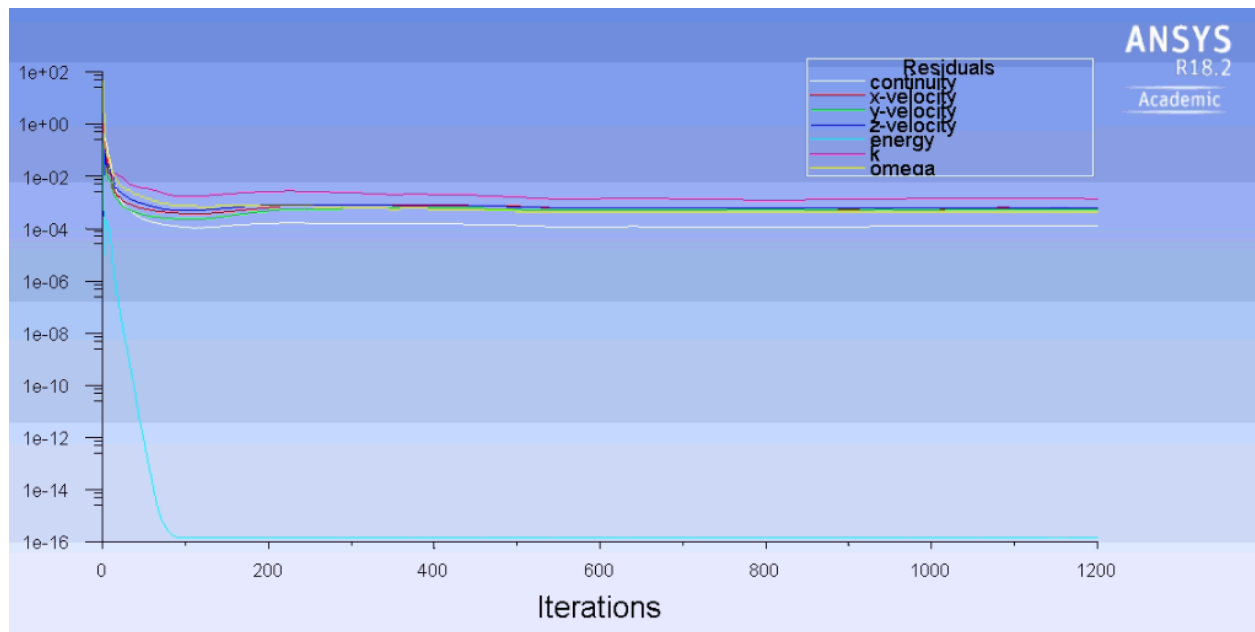


Figure C-5. Residuals for the 0.67 kg/s case (k- ω turbulence model).

C.3 Mesh dependence study

The solution monitoring of the simulations done for the mesh dependence study (see section 5.1.3) is shown in C.4, C.5 and C.6.

C.4 Mesh number 1

The first mesh that was generated consisted of the least number of elements. The solution monitoring of the Fluent model is shown in Figure C-6. After approximately 6500 iterations the residuals were below 10^{-4} . The peaks in the residuals at approximately 5750 iterations was due to the simulation being restarted from an autosave after a crash had occurred.

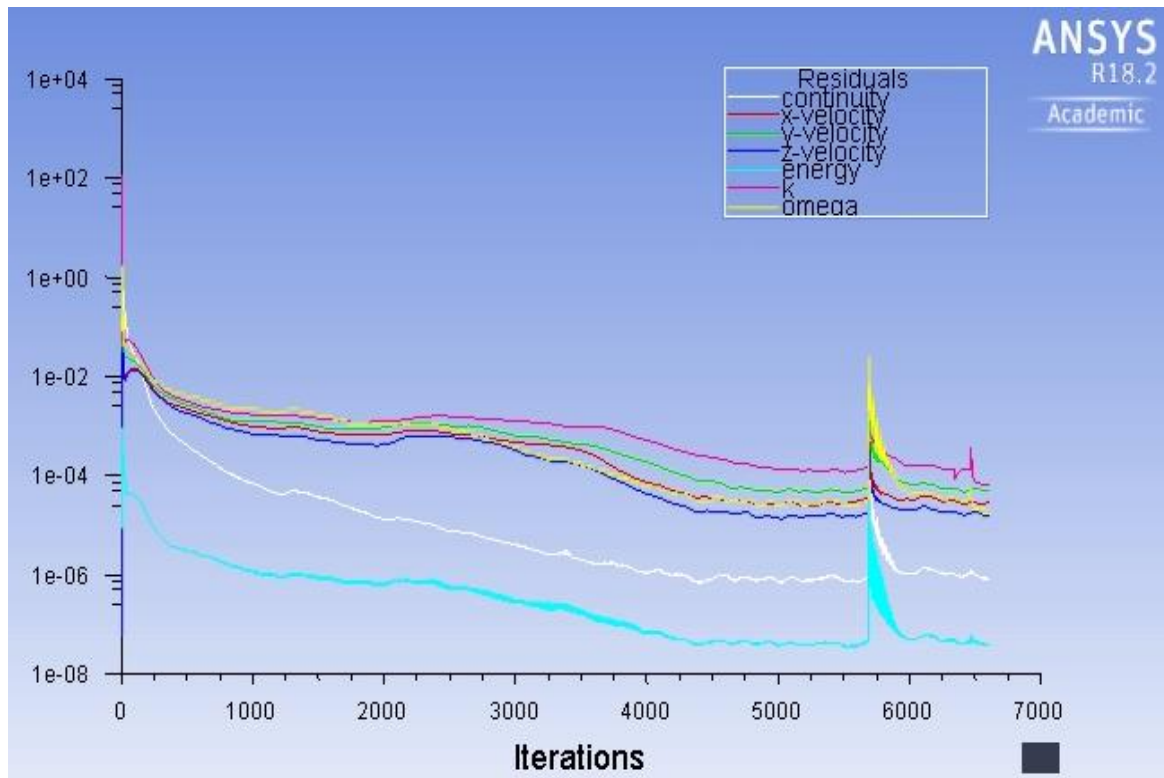


Figure C-6. Ansys residuals for Mesh number 1.

The area averaged outlet temperature of the middle riser was also monitored by Fluent and is shown in Figure C-7. The outlet temperature stabilized after relatively few iterations.

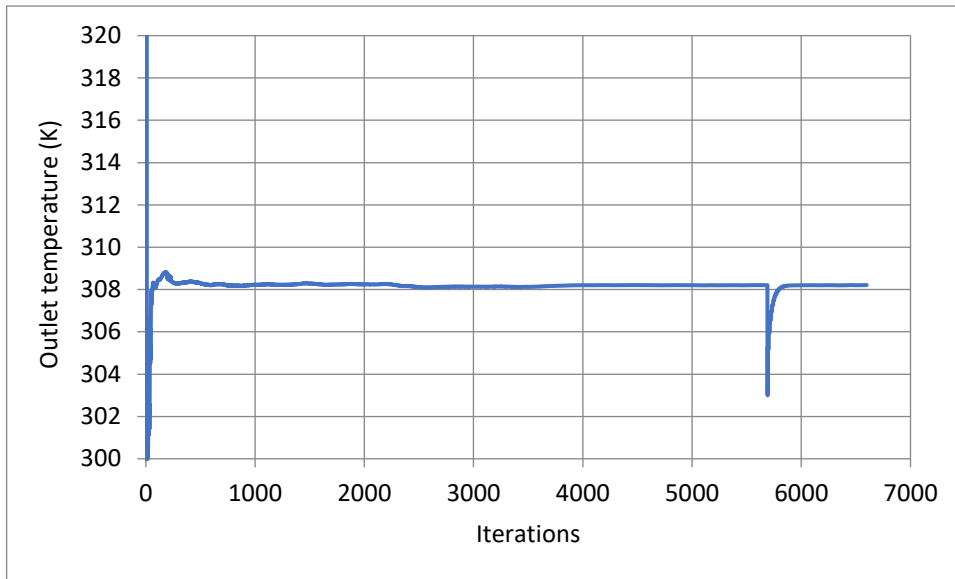


Figure C-7. Surface average of the outlet temperature of Mesh 1.

The residual criteria obtained by using Star-CCM+ for Mesh number 1 is shown in Figure C-8. The simulation ran for 10000 iterations after which all of the criteria were below 10^{-5} . The oscillations that persisted were regarded as negligible.

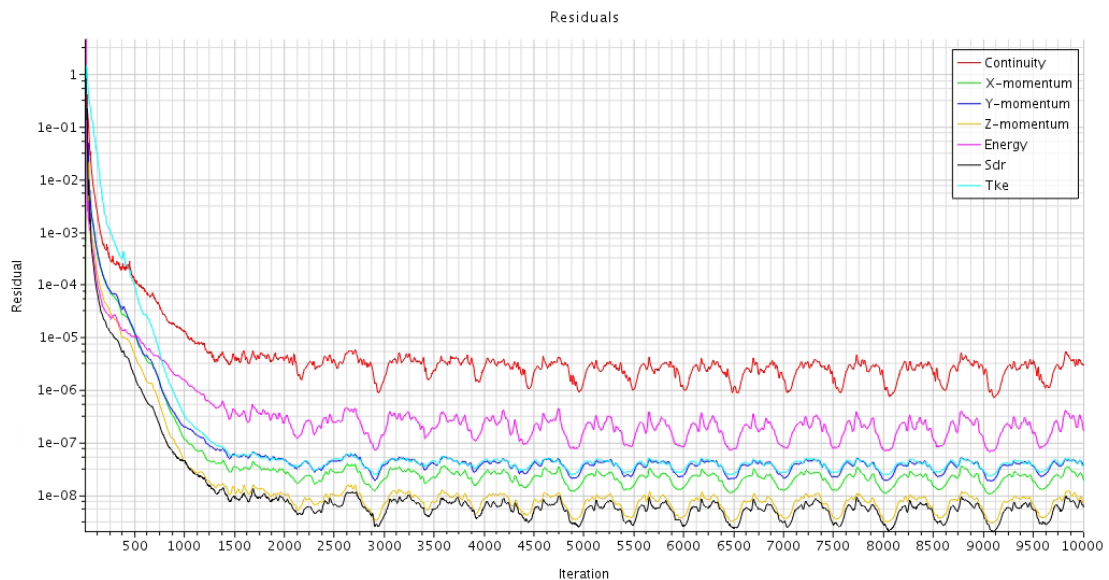


Figure C-8. Star-CCM+ residual criteria for Mesh number 1.

Figure C-9 shows the monitor of surface average of the outlet temperature from the middle riser as obtained by using Star-CCM+. The temperature had stabilized after approximately 1500 iterations.

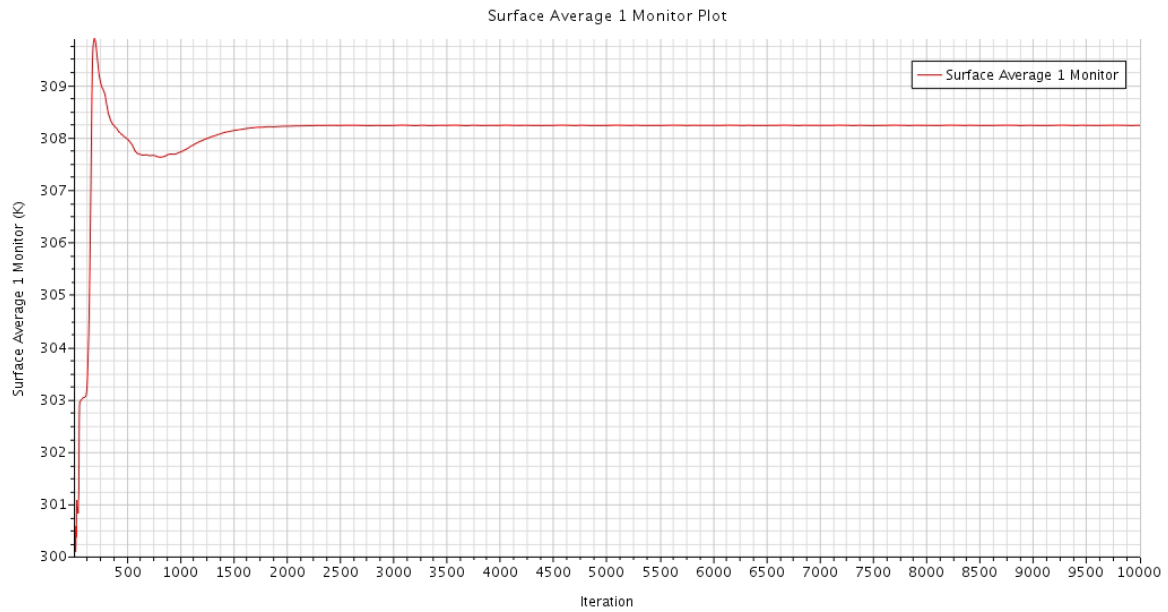


Figure C-9. Star-CCM+ surface averaged outlet temperature monitor for Mesh 1.

C.5 Mesh number 2

The second mesh that was used in section 5.1.3 contained more elements than Mesh number 1 but fewer than Mesh number 3. The residual criteria obtained by using Fluent for Mesh number 2 is shown in Figure C-10. The criteria were below 10^{-4} when the simulation was stopped after 4000 iterations. Figure C-11 shows the surface average of the outlet temperature of the middle riser tube. The outlet temperature had stabilized after approximately 1000 iterations.

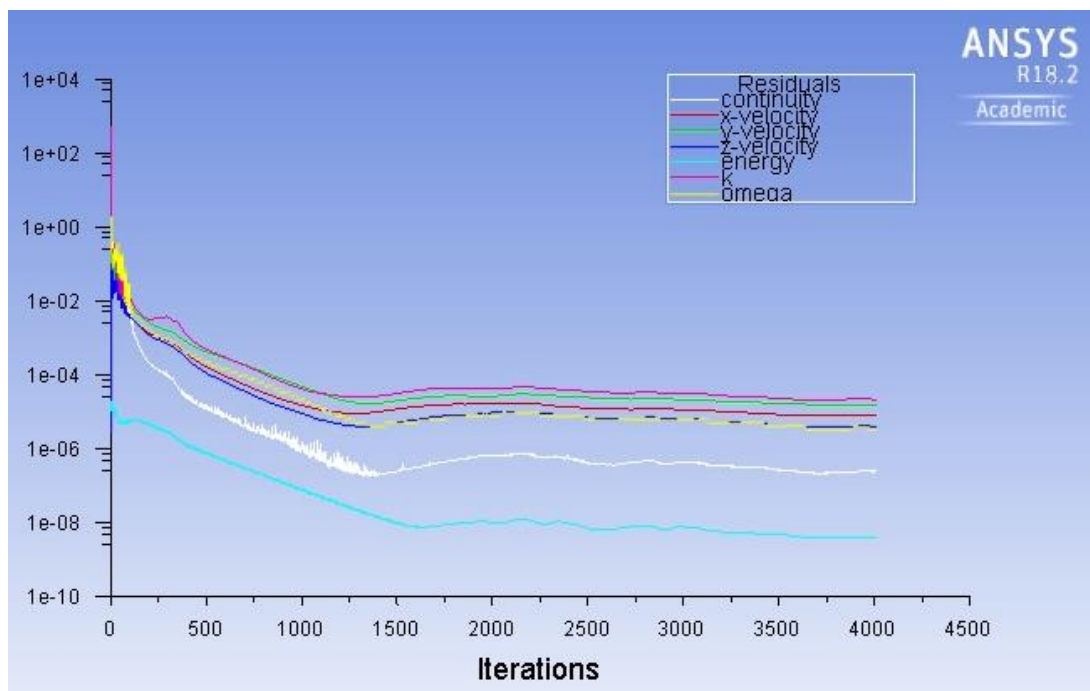


Figure C-10. Ansys residuals for Mesh 2.

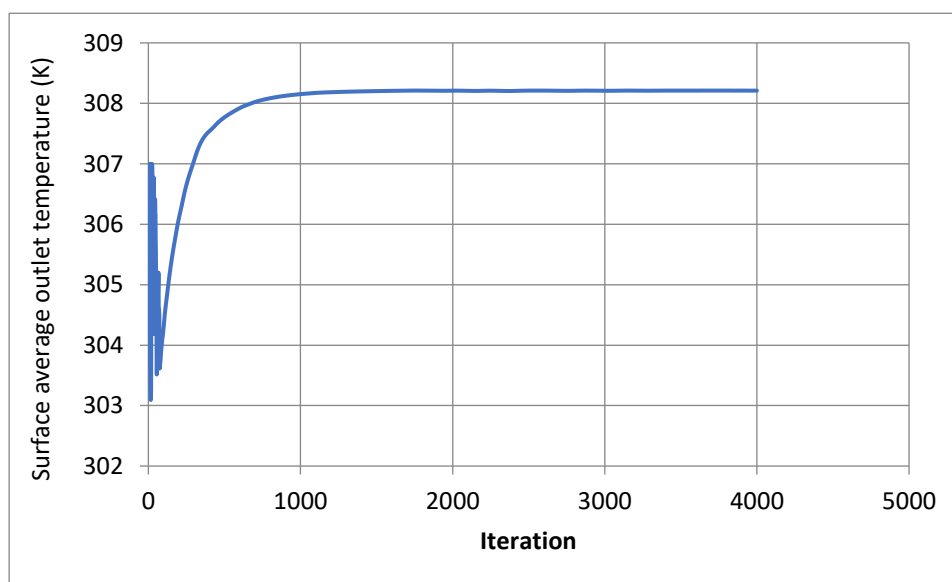


Figure C-11. Surface average of outlet temperature obtained by Fluent for Mesh number 2.

The residual criteria obtain by Star-CCM+ for Mesh number 2 is shown in Figure C-12. The simulation was run for 17000 iterations, after which the residual criteria were below 10^{-4} . The corresponding outlet temperature is shown in Figure C-13, which shows a stabilized temperature after approximately 4000 iterations. The spikes in the figure are due to the simulation being continued from an autosaved point after a crash had occurred.

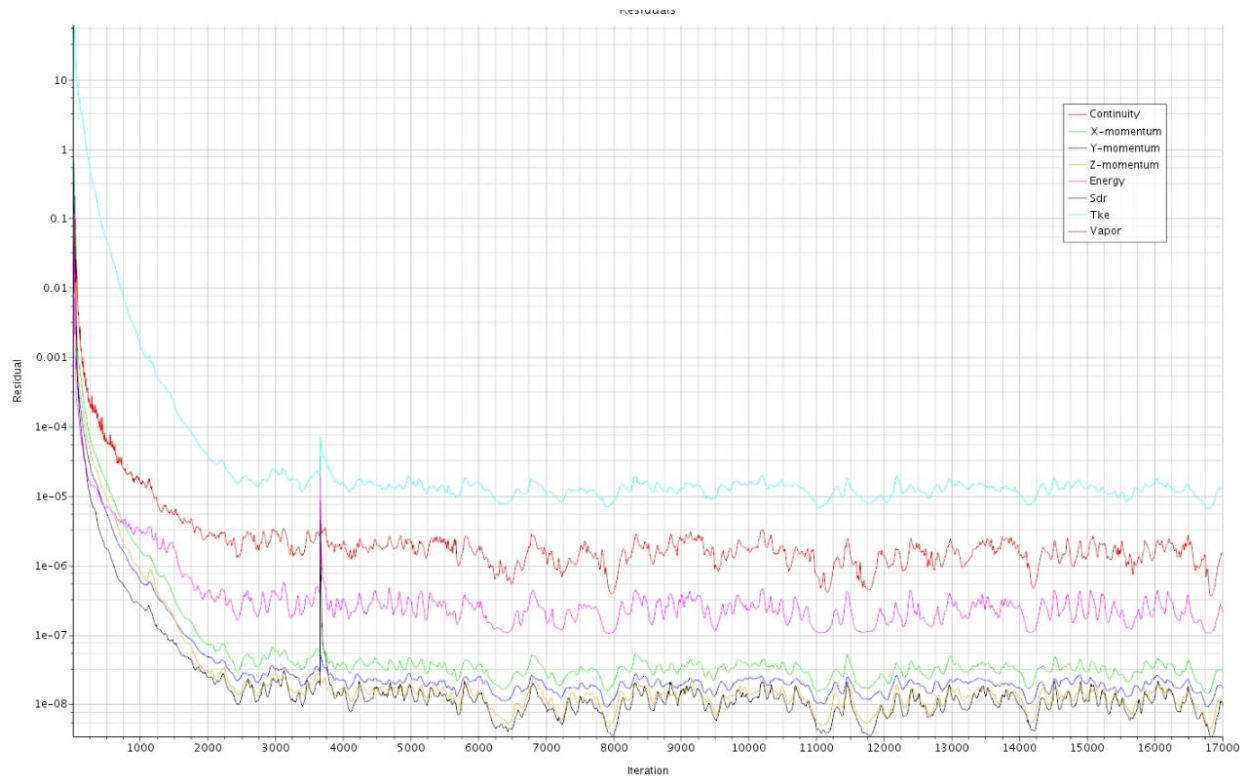


Figure C-12. Star-CCM+ residuals for Mesh 2.



Figure C-13. Outlet temperature monitor for Mesh 2.

C.6 Mesh number 3

The third mesh that was tested (see section 5.1.3) was refined further to contain more elements than Mesh number 2. Figure C-14 shows the residuals obtained for Mesh 3 by using Fluent. The spikes at ≈ 3200 were due to changes in the relaxation parameters. The parameter with the most profound effect on the graph was the parameter for energy, which was increased from 0.9 to 1. Although oscillations continued, the criteria were below 10^{-4} , which was deemed as satisfactory.

Figure C-15 shows the area weighted average of the temperature at the outlet of the middle riser. The outlet temperature appears stable for the iterations that it was monitored.

The integral of the heat flux on the riser tube surfaces is shown in Figure C-16. The jump followed the increase of the energy relaxation factor. The graph shows that the integral of the flux was stable over the range that it was monitored.

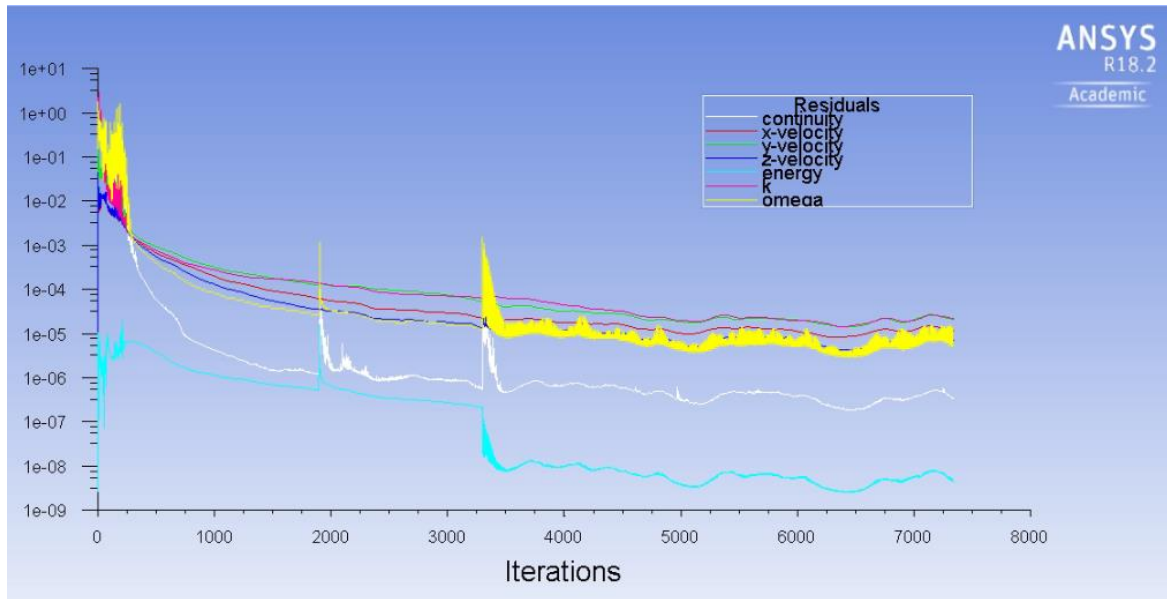


Figure C-14. Residuals for Mesh 3.

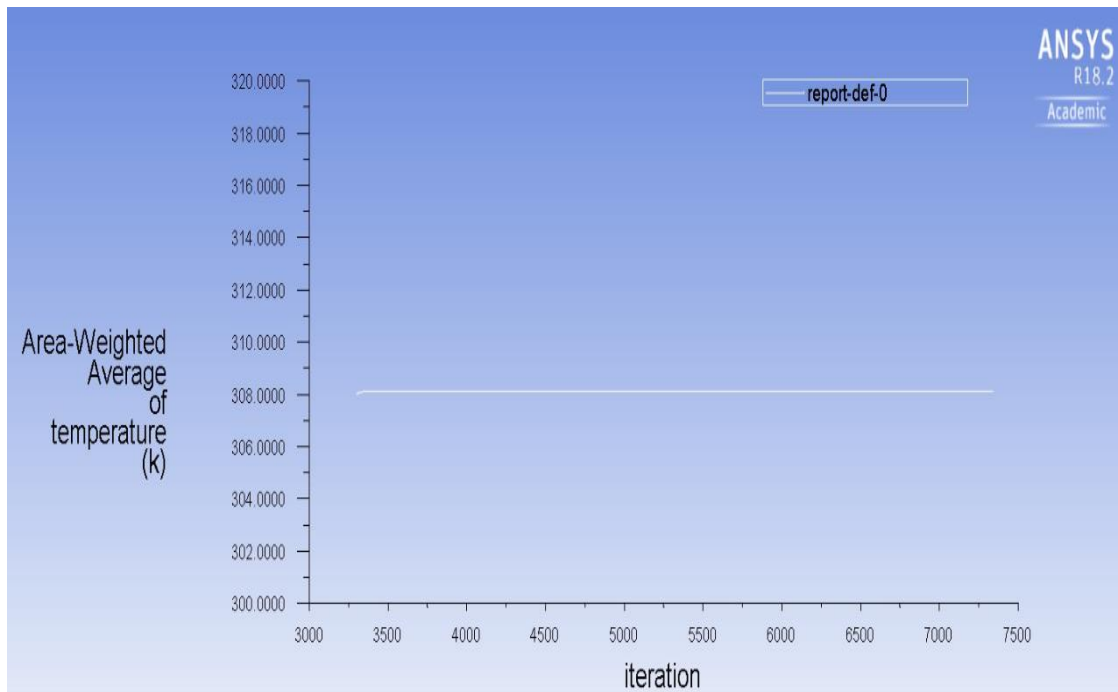


Figure C-15. Outlet temperature per iteration for Mesh 3.

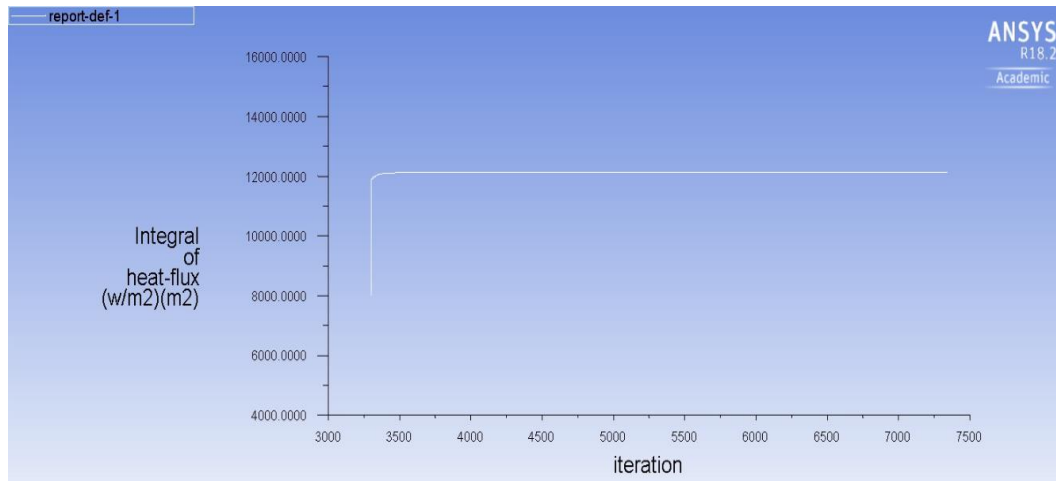


Figure C-16. Integral of heat flux on the riser tube inner surfaces (Mesh 3).

The residuals for Mesh 3 as obtained by using Star-CCM+ is shown in Figure C-17. The criteria were below 10^{-4} , which was considered as satisfactory. The surface average of the outlet temperature is shown in Figure C-18. The temperature had stabilized after approximately 200 iterations and showed little instability over the rest of the run.

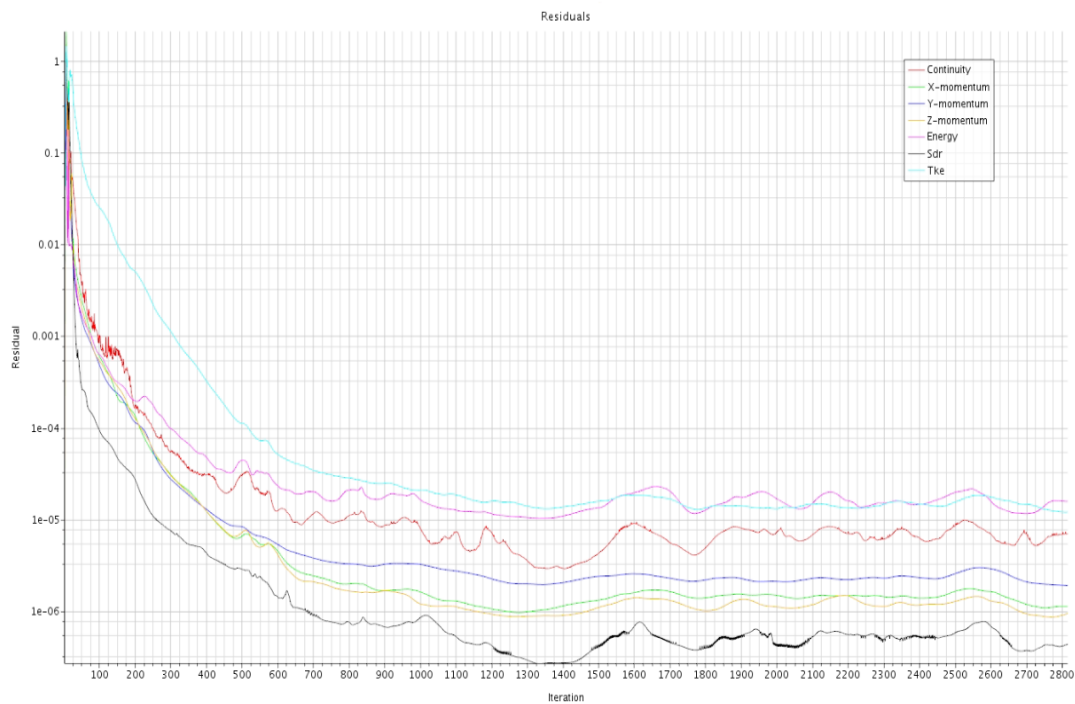


Figure C-17. Residuals for the Star-CCM+ simulation for Mesh 3.

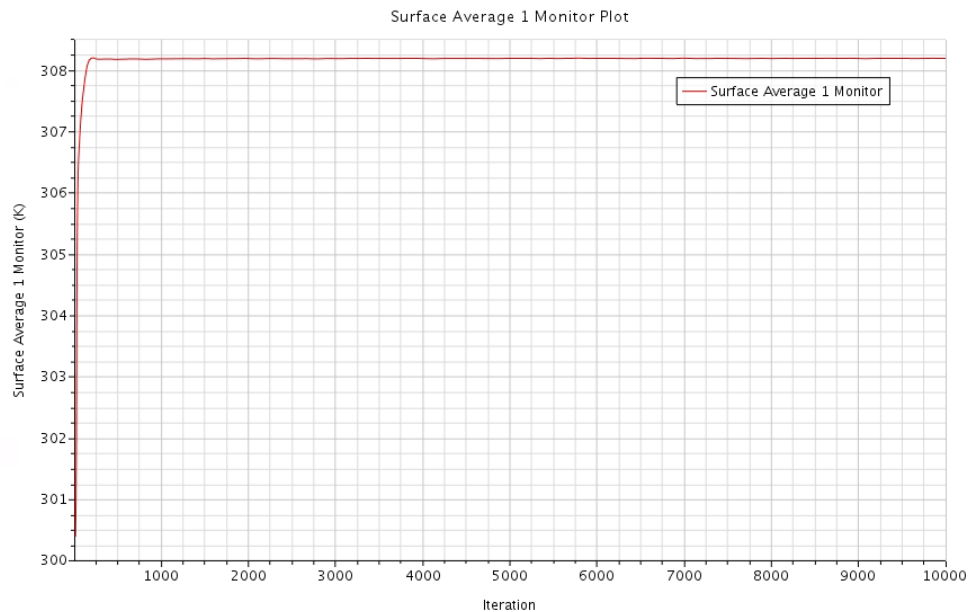


Figure C-18. Surface average temperature monitor for Mesh 3.

C.7 k- ϵ turbulence model

Figure C-19 shows the residual criteria obtained for the circular tubes case with fins, with the k- ϵ turbulence model by using Star-CCM+. Convergence was assumed since the residual criteria were below 10^{-4} by the time the simulation was stopped after approximately 8000 iterations. Figure C-20 shows the surface integral of the heat flux on Riser 2's outer surface as a function of iteration. The first 3400 iterations were truncated from Figure C-20 for optimal resolution for the final iterations. The integral had also stabilized by 8000 iterations strengthening the assumption that convergence had been achieved.

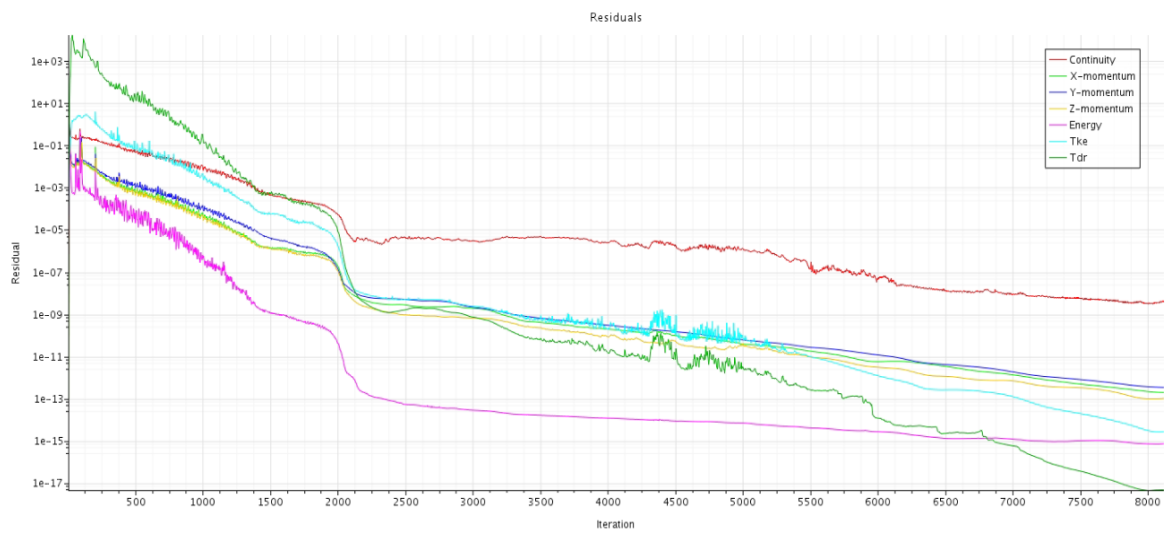


Figure C-19. Residual criteria of the k- ϵ model in the verification chapter.

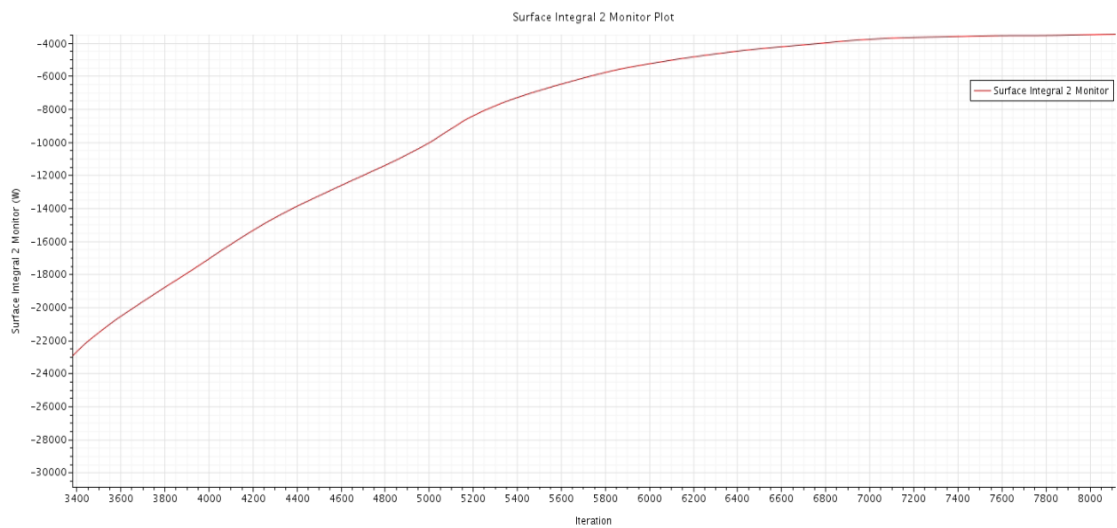


Figure C-20. Surface integral of the heat flux on the outside surface of Riser 2.

C.8 Fluent 3D model

Figure C-21 shows the residual criteria of the Fluent model that was directly compared with the corresponding Star-CCM+ model discussed in Chapter 6. The criteria were below 10^{-4} when the simulation was stopped after approximately 6500 iterations.

Figure C-22 shows an area weighted average of the heat flux on Riser 2's outer surface. The flux had stabilised by the time the simulation was stopped. Considering the residual criteria and the monitored heat flux it was assumed that convergence had been achieved.

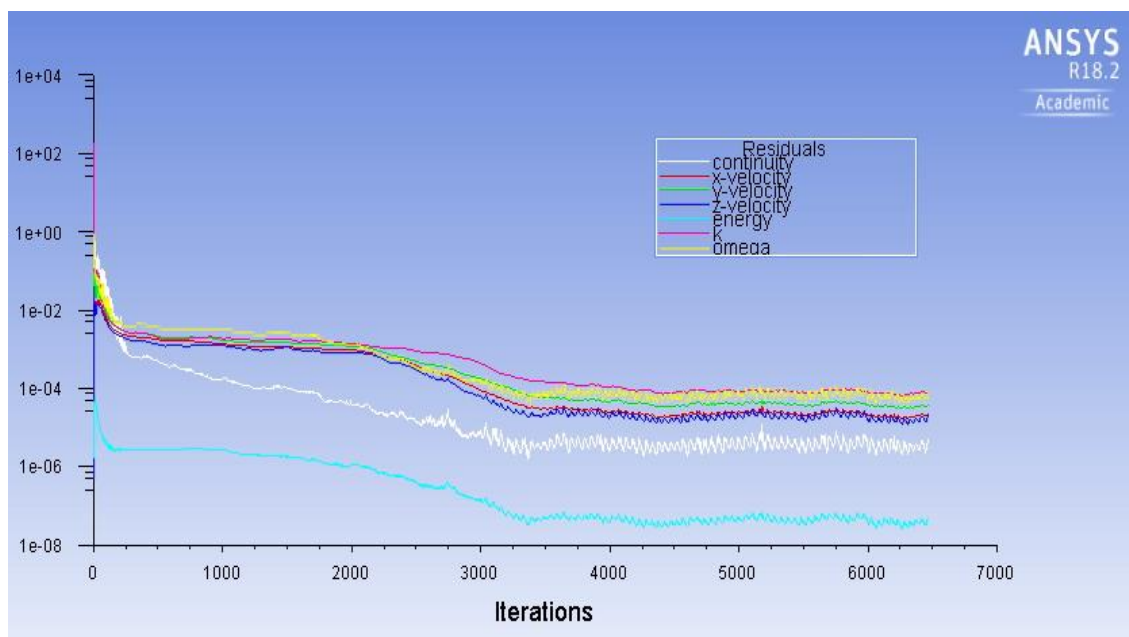


Figure C-21. Residual criteria for the Fluent 3D model case.

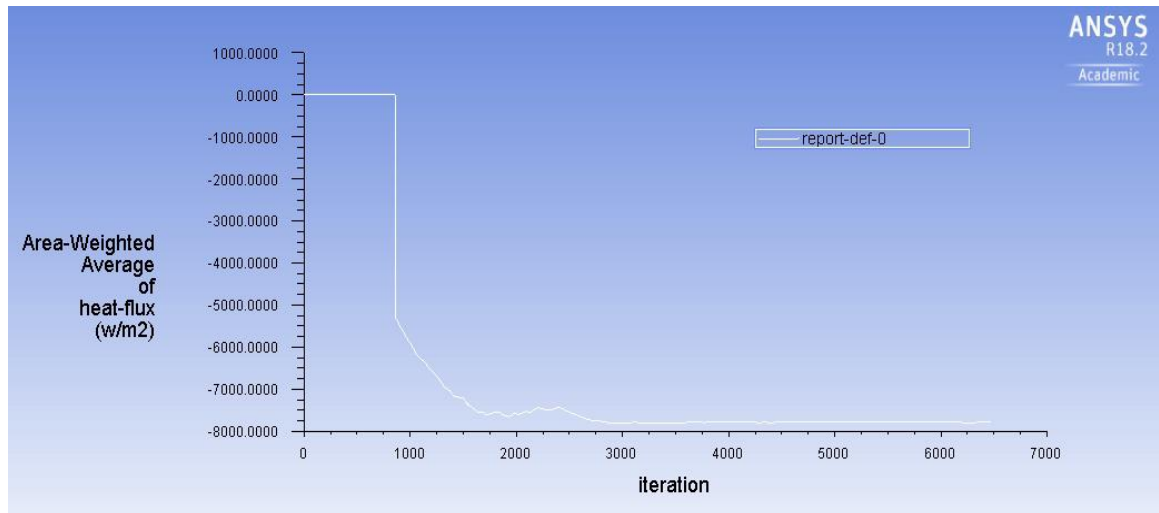


Figure C-22. Area weighted average of the heat flux on Riser 2's outer surface.

C.9 Round riser tubes with fins

The criteria used to judge whether the simulation (of which the results are discussed in section 8.2.2) had converged were the residuals and the integral of the heat flux on the middle riser tube. The residuals obtained by using Star-CCM+ are shown in Figure C-23, the integral of the heat flux on the middle riser's outer surface is shown in Figure C-25.. The residuals were below 10^{-7} for all criteria after the simulation was stopped after 10400 iterations. The integral of the heat flux was also indicative of a converged solution since there was relatively small changes in this parameter after 8000 iterations. The solution was therefore considered as being converged.

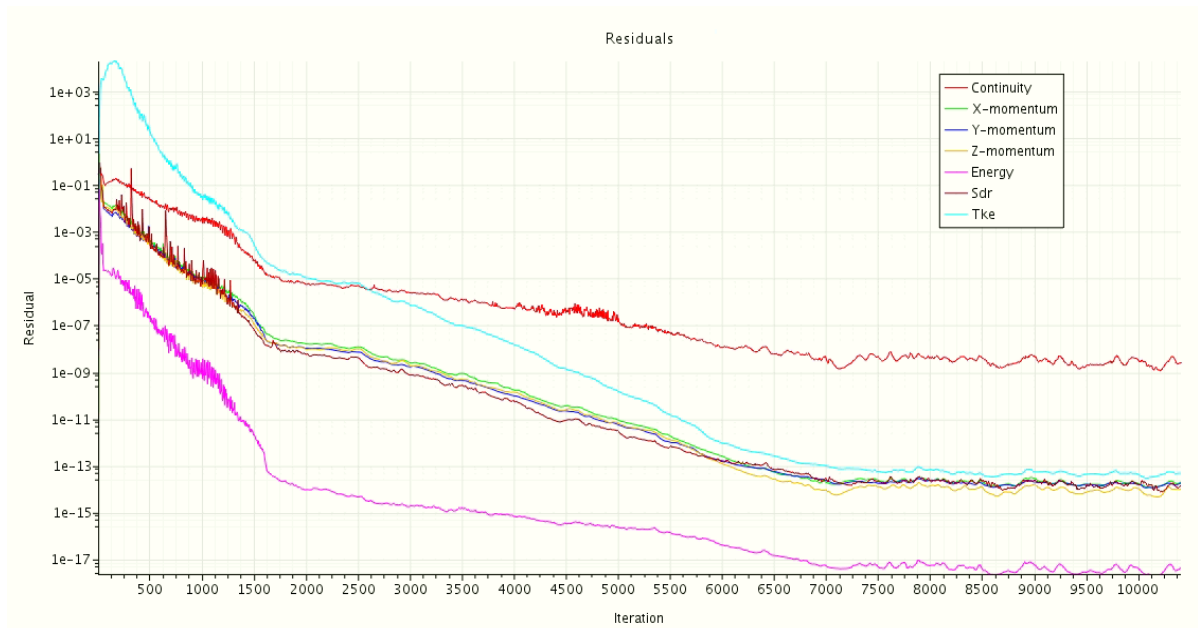


Figure C-23. Residuals for the case with round riser tubes with fins.

C.10 Round tubes without fins

The residual criteria for the case (of which the results are discussed in section 8.2.3) are shown in Figure C-24 and all of the criteria are below 10^{-8} . The residuals had reached a semi stable condition at approximately 2500 iterations, with oscillations that are negligibly small.

The surface average of the middle riser's outlet temperature was monitored per iteration and is shown in Figure C-26. The temperature varied considerably in the beginning stages of the simulation (iteration 1 - 1500) but had reached a stable temperature around 307.5 K at approximately 2000 iterations.



Figure C-24. Residuals for the case with round tubes without fins.

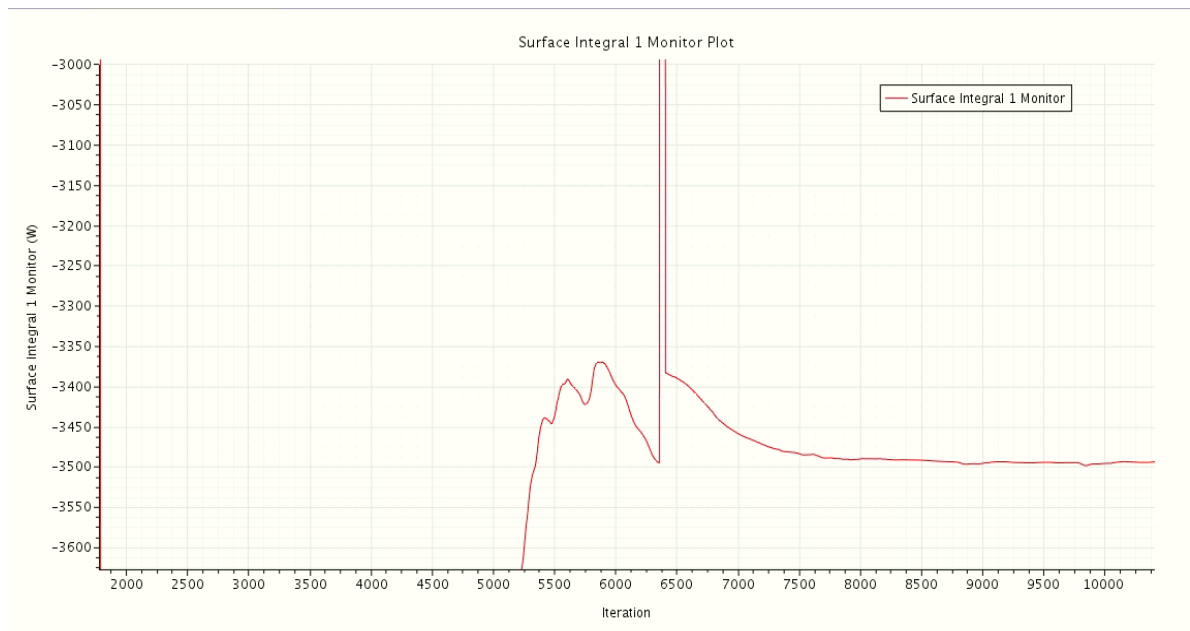


Figure C-25. Integral of heat flux monitor used for the case with round riser tubes with fins.

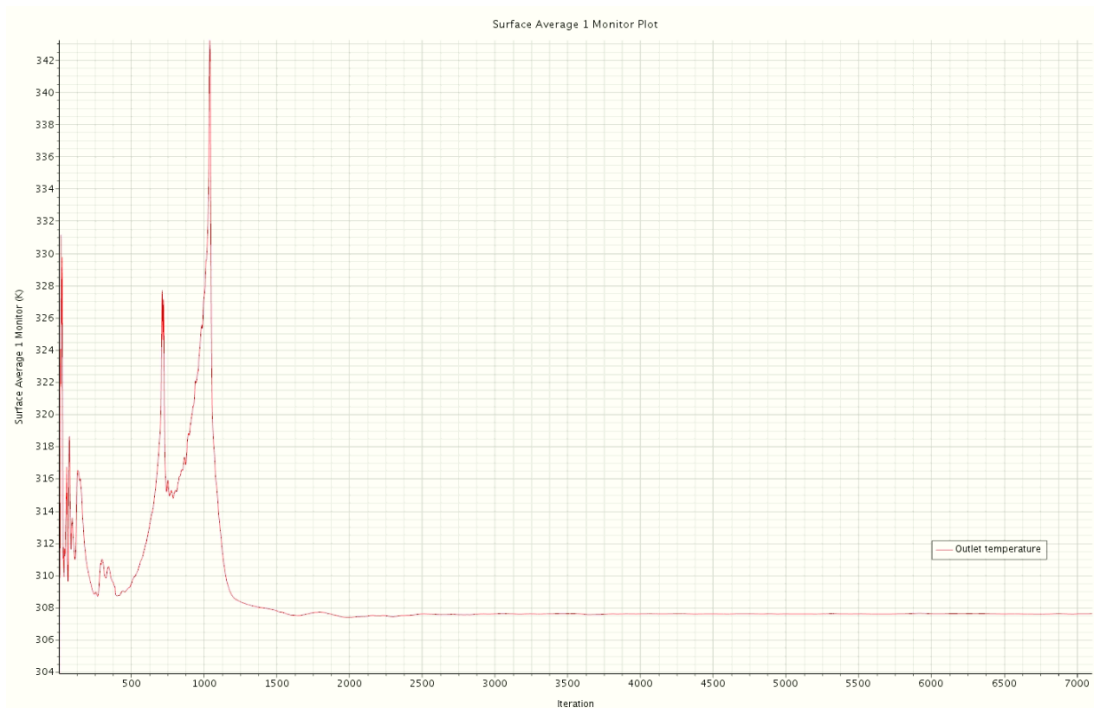


Figure C-26. Outlet temperature monitor for the case with round tubes without fins.

C.11 Square tubes with fins

The residuals for the case (of which the results are discussed in section 8.2.4) with square tubes with fins are shown in Figure C-27. The criteria are below 10^{-7} after 5600 iterations. The integral of the heat flux on the middle riser's front surface was also monitored and is shown in Figure C-28. The integral had reached a stable solution after approximately 5000 iterations. The zero value from 0 to 980 iterations was due to the wrong surface being selected to compute the integral. The scale was chosen to get maximum resolution from the later iterations, hence the maximum values between 1000 and 1600 iterations are not shown. Lastly, the outlet temperature of the middle riser was also monitored and is shown in Figure C-29. The change in

outlet temperature is relatively small above 3000 iterations. The solutions were considered as being converged since all of the monitored parameters reached stable values.

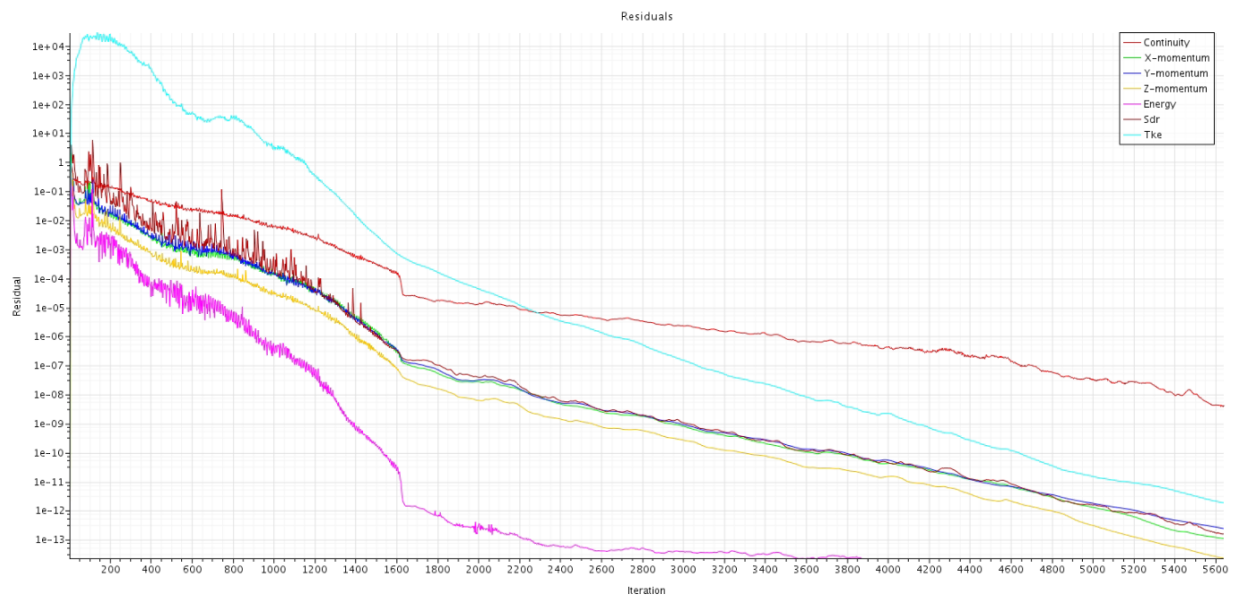


Figure C-27. Residuals for the case with square tubes with fins.

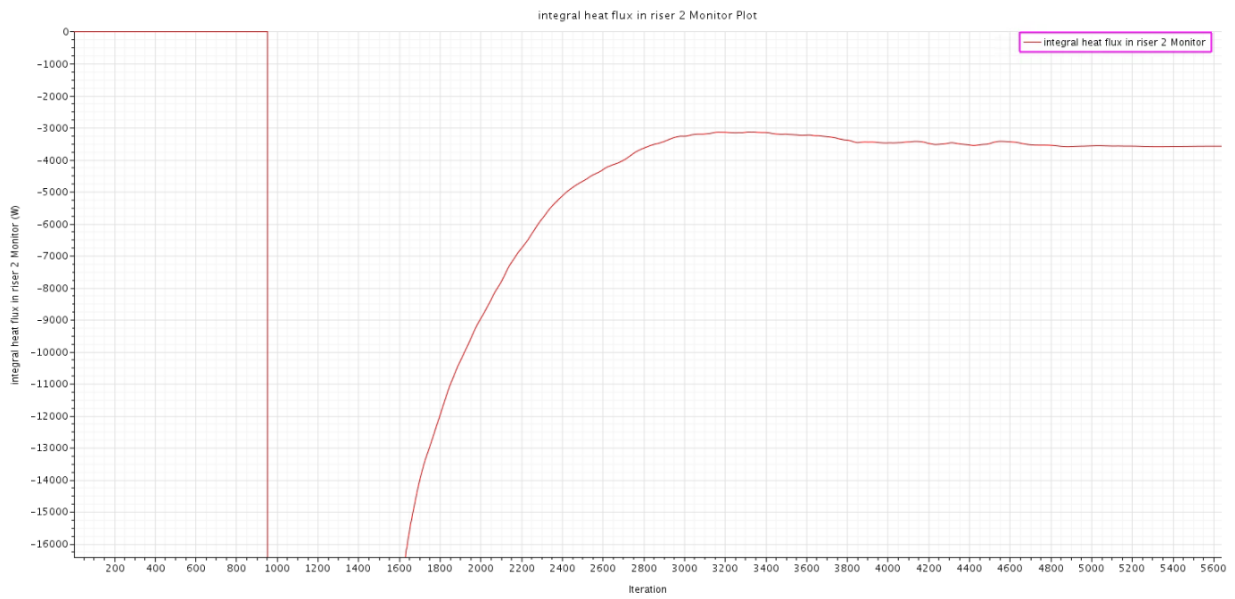


Figure C-28. Integral of the heat flux monitor for the case with square tubes with fins.

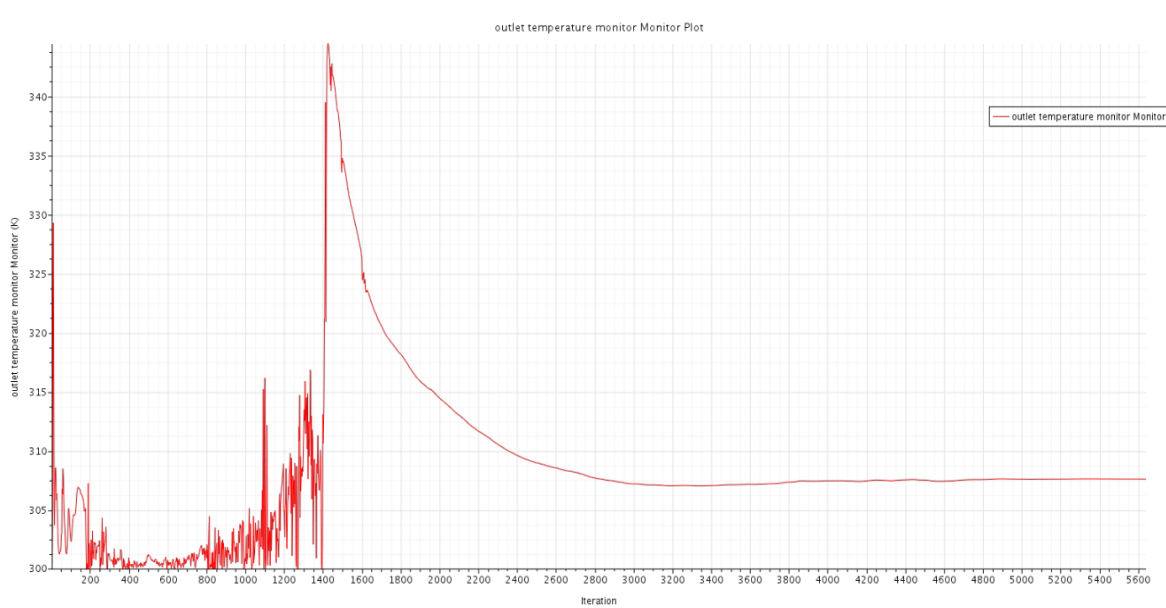


Figure C-29. Outlet temperature monitor for the case with square tubes with fins.

C.12 Square fins without fins

The residuals for the case (of which the results are discussed in section 8.2.5) with square tubes with fins are shown in Figure C-30. The criteria are all below 10^{-9} after 7500 iterations.

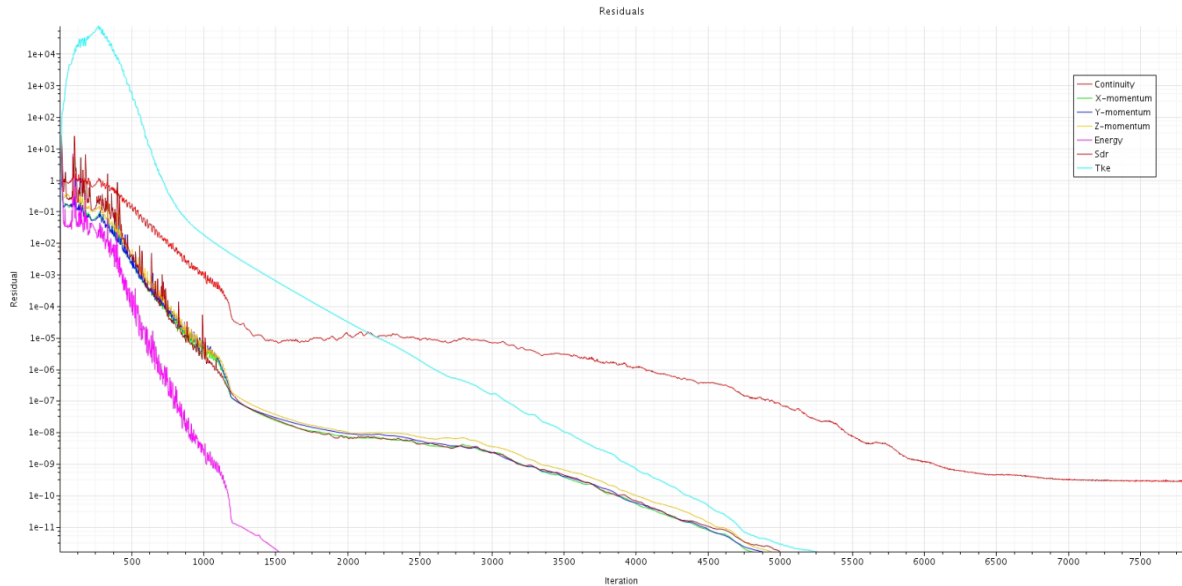


Figure C-30. Residuals for the case with square tubes without fins.

The surface average of the heat flux on the boundary of the middle riser is shown in Figure C-31. The scale was chosen for a good resolution at the higher iterations, therefore the minimum values between 0 and 1700 iterations are truncated.

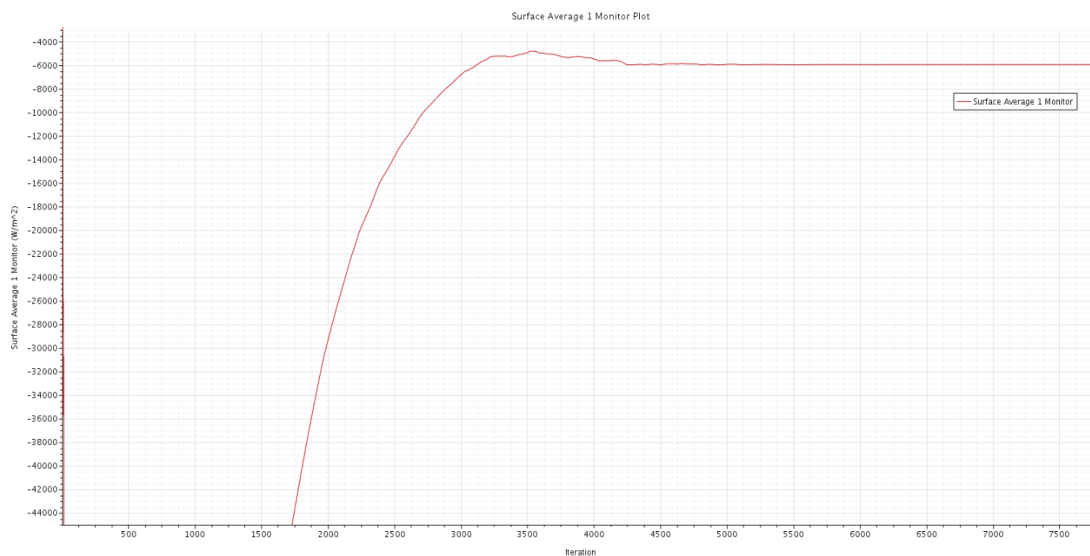


Figure C-31. Surface average heat flux monitor for the case with square tubes with fins.

Since the residuals were well below the criteria (residuals $< 10^{-4}$) and the average heat flux had stabilized the solution was considered as being converged.

C.13 Rectangular with fins

The residuals for the case (of which the results were discussed in section 8.2.6) with rectangular tubes with fins is shown in Figure C-32. The criteria are all below 10^{-6} at 10000 iterations when the simulation was stopped. The average heat flux on one of the middle risers was also monitored and is shown in Figure C-33. The flux had stabilized at 4400 iterations, after which the change in heat flux was very small. The scale of the graph was chosen to provide maximum resolution for the values at the higher iterations and therefore the maximum values of the initial stages are truncated.

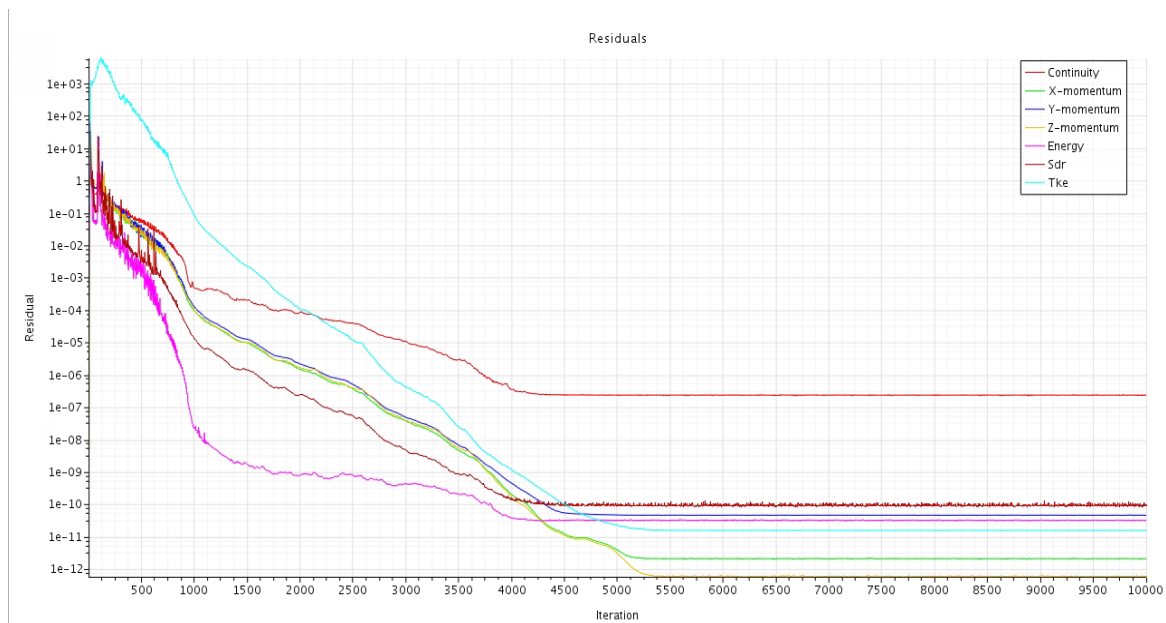


Figure C-32. Residuals for the case with rectangular tubes with fins.

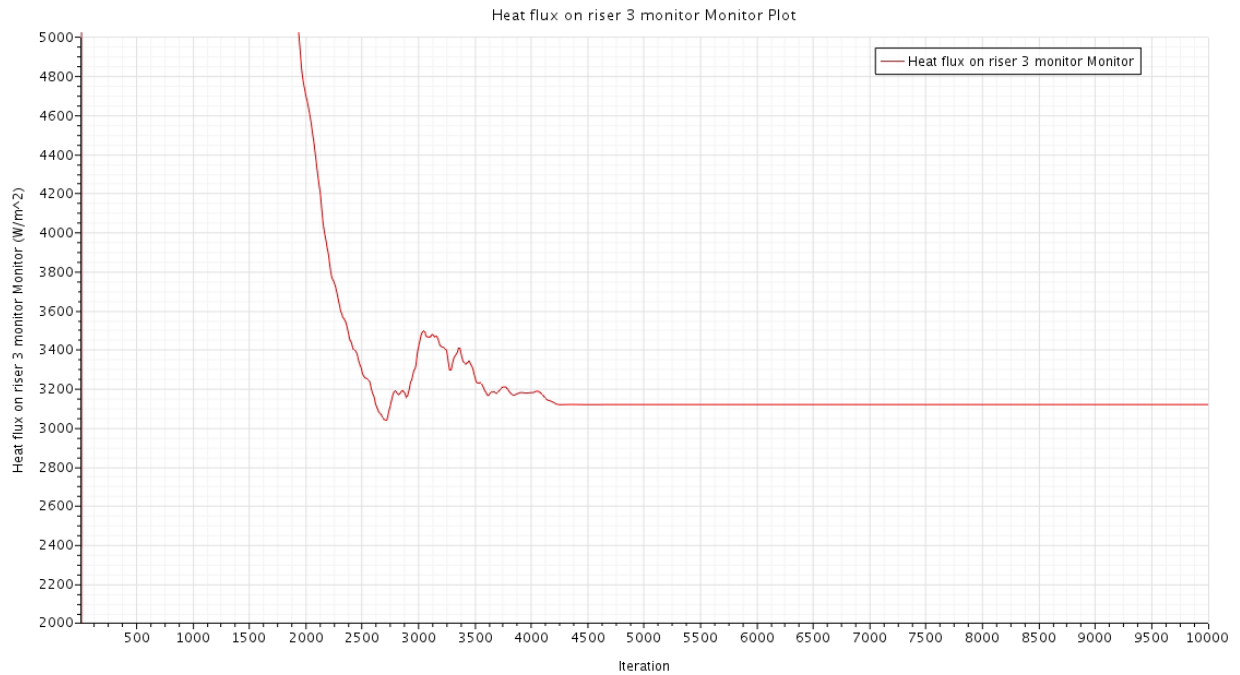


Figure C-33. Average heat flux monitor for the case with rectangular tubes with fins.

C.14 Rectangular tubes without fins

The residual criteria (of the which the results were discussed in section 8.2.7) and surface average monitor that was used to determine convergence are shown in Figure C-34 and Figure C-35 respectively. The residual criteria were all below 10^{-6} when the simulation was stopped at 10000 iterations. The heat flux on the second riser tube's surface stabilized at around 5000 iterations. Note that the values between 1 and 1700 iterations are truncated to provide the best resolution for the higher iterations in the graph.

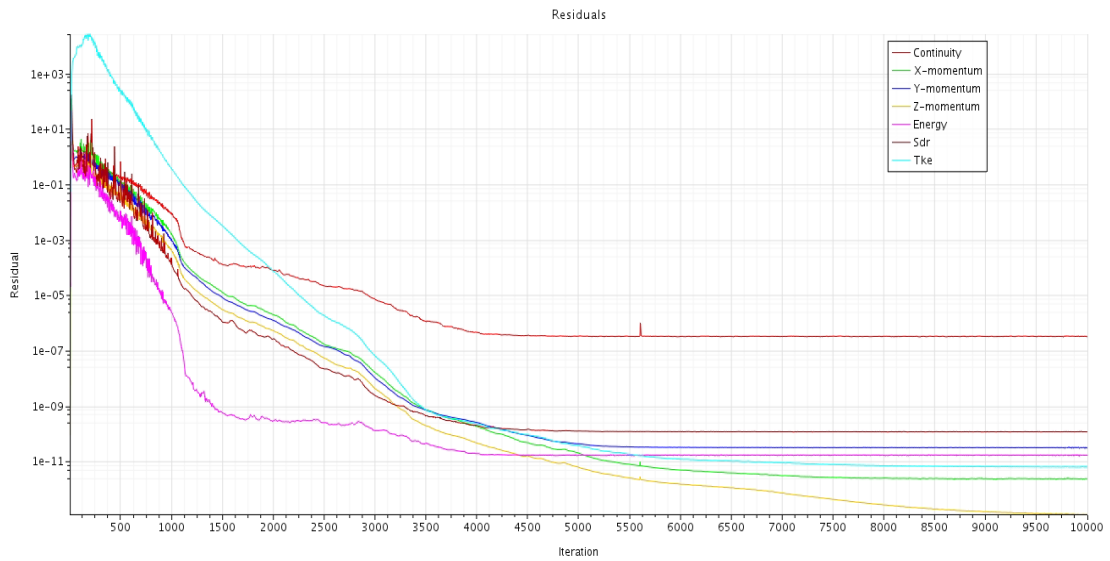


Figure C-34. Residuals for the case with rectangular tubes without fins.

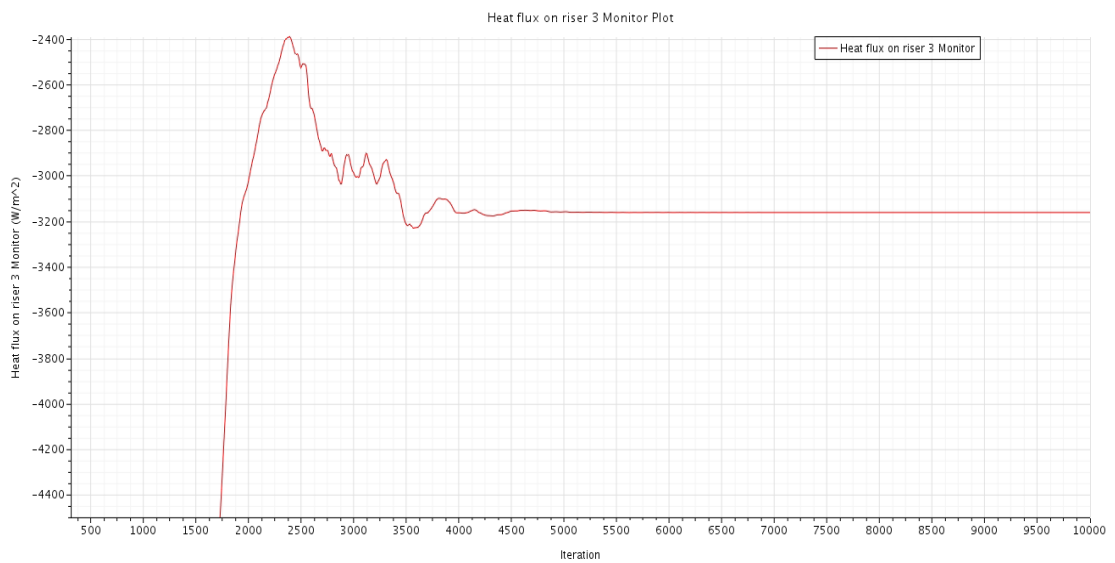


Figure C-35. Heat flux monitor for the case with rectangular tubes without fins.

C.15 Coupled simulations

C.16 Manifolds modelled by using Flownex (Variation B)

The residuals for the Variation B model (discussed in 7.3 and 8.3.1) are shown in Figure C-36 while the outlet temperature monitors are shown in Figure C-37, as obtained by using Ansys Fluent. The criteria are below 10^{-4} . The outlet temperatures for each individual riser tube were monitored and had stabilized at approximately 4000 iterations.

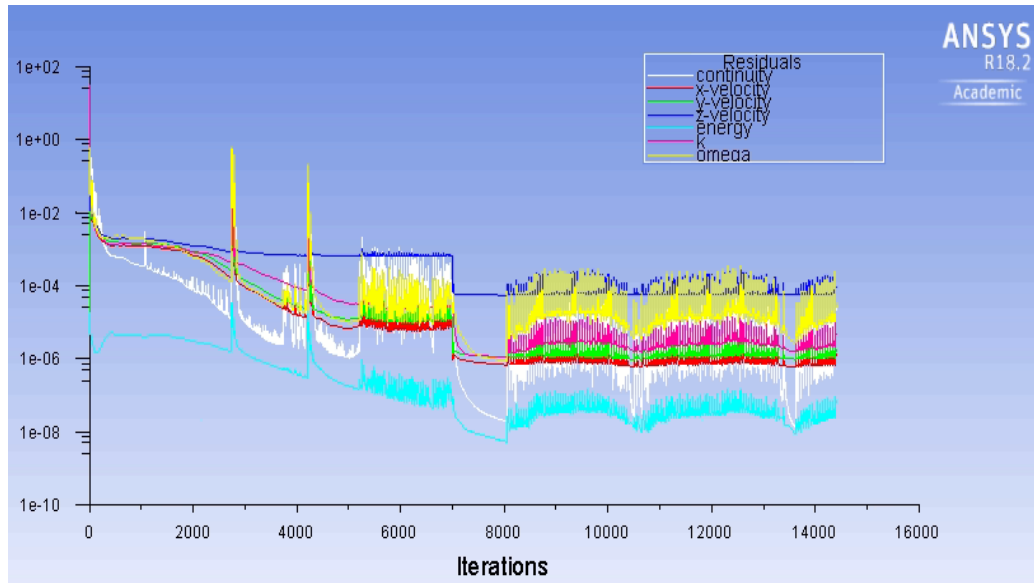


Figure C-36. Residuals for Variation B.

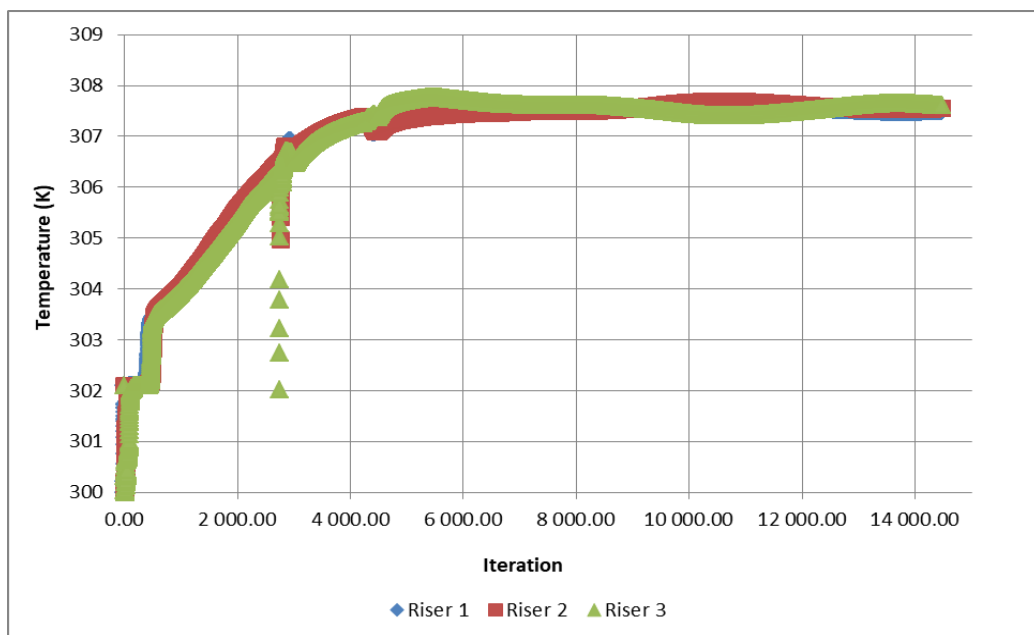


Figure C-37. Outlet temperature monitors for Variation B.

C.17 Natural circulation Variation C

The residual criteria obtained for the Variation C model by Fluent is shown in Figure C-38. The run was stopped after approximately 11000 iterations when the residual criteria were below 10^{-4} . The noticeable decrease in the residuals at 3800 iterations was due to the value of the heat energy on the riser tubes' inner surface approaching the expected final value. The limiter within Flownex had therefore been used up until that point. The momentum under-relaxation factor was decreased at approximately 5000, 10400 iterations, which resulted in the decreases that are most visible in the z-velocity residual.

The integral of the heat flux was calculated on the inside surface of the riser tubes to obtain the heat energy added per riser tube. The heat energy added per riser tube was one of the output parameters that were transferred to Flownex. It was therefore also monitored as an indication of convergence. The heat energy added for the middle riser per iteration is shown in Figure C-39. The graph indicated that the heat added per riser had stabilized by approximately 4000 iterations. The graph also shows the gradual increase in the energy due to the decreased under-relaxation factor imposed on energy during the run.

Based on the residual criteria and the values of the integral of the heat flux the simulation was considered as being sufficiently converged.

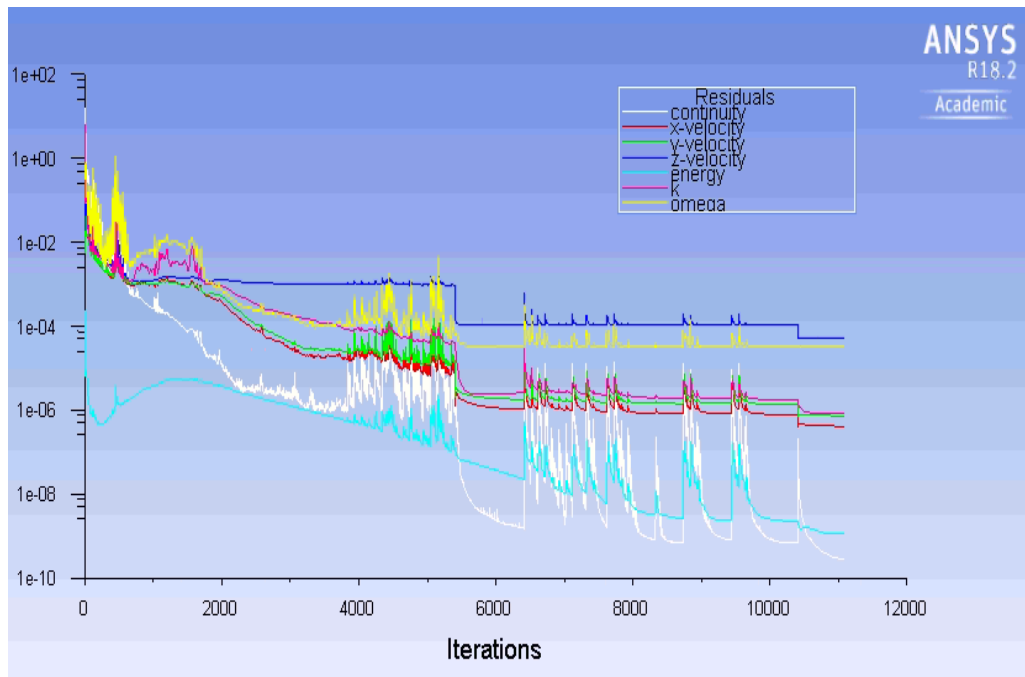


Figure C-38. Residuals of the Fluent simulation for Variation C.

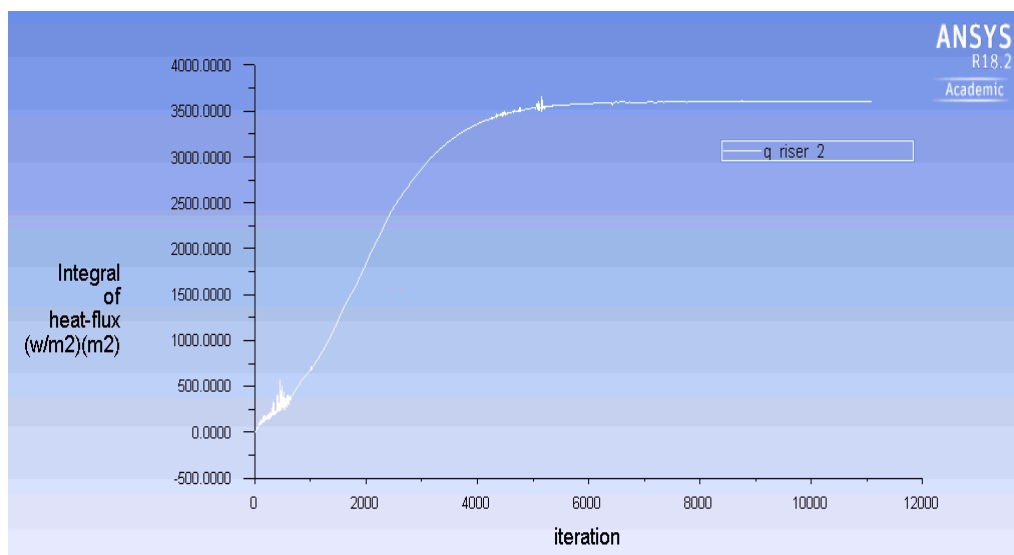


Figure C-39. Heat energy added per iteration per riser for Variation C.

APPENDIX D ADDITIONAL PARAMETERS AND PROPERTIES CALCULATED

D.1 Additional parameters and properties

The calculation of the convection heat transfer coefficients required that a number of other parameters to be calculated. The parameters were omitted from Chapter 8.2 for the sake of brevity. The parameters are presented in this chapter. D.2 shows the table of additional results for the circular case with fins. D.3 shows the results for the circular tubes case without fins. D.4 shows the results for the square tubes case with fins and D.5 the case without fins. D.6 shows the results for the case with rectangular tubes with fins and D.7 for the rectangular case without fins. The parameters are shown for the middle riser since the distributions were similar in the other tubes.

In each case the same parameter table is presented. The results were calculated on four surfaces, at the eight elevations (denoted by z) indicated, all within the heated section. The parameters that were calculated included Grashof number (Gr), Reynolds number (Re), Prandtl number (Pr), Nusselt number (Nu) and Rayleigh number (Ra). The parameters were evaluated for the film temperature. The film temperature was calculated as the average temperature between the wall temperature and bulk fluid temperature. The fluid properties of dynamic viscosity (μ), density (ρ), thermal conductivity (k) and specific heat capacity (C_p) were also calculated for the film temperature. The column dT represents the difference between the wall and bulk fluid temperature at the given elevation.

D.2 Circular tubes with fins

Table D.2-1 shows additional parameters that were calculated for the front surface in the middle riser for the case with circular tubes with fins.

Table D.2-1. Additional parameters (circular tubes with fins) associated with the front surface inside Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	C _p (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	308.60	5.47	305.87	994.79	6383.27	7.54	4179.52	0.62	5.10	3.67	1.87	45.27	44.91	13.02
0.4	313.69	10.54	308.41	993.93	6712.31	7.17	4179.43	0.62	4.82	7.81	3.76	46.71	54.98	40.05
0.8	314.86	11.64	309.05	993.71	6795.75	7.08	4179.42	0.62	4.75	8.83	4.20	47.05	56.74	42.81
1.2	315.44	11.87	309.51	993.55	6857.01	7.02	4179.43	0.62	4.71	9.17	4.32	47.23	57.17	43.35
1.6	316.22	12.14	310.15	993.32	6943.24	6.93	4179.44	0.62	4.64	9.61	4.46	47.47	57.67	43.95
2	316.84	12.20	310.74	993.10	7023.26	6.85	4179.45	0.63	4.58	9.88	4.53	47.66	57.87	44.07
2.4	317.15	11.92	311.19	992.94	7084.30	6.80	4179.47	0.63	4.54	9.82	4.46	47.77	57.57	43.41
2.8	318.55	12.68	312.21	992.56	7223.26	6.67	4179.52	0.63	4.44	10.8	4.82	48.18	58.84	45.12

Table D.2-2 shows additional parameters that were calculated for the back surface in the middle riser for the case with circular tubes with fins.

Table D.2-2. Additional parameters (circular tubes with fins) associated with the back surface inside Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	C _p (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	304.14	1.01	303.64	995.50	6104.64	7.89	4179.70	0.62	5.36	0.620	0.332	44.01	27.44	40.13
0.4	305.41	2.27	304.28	995.30	6183.45	7.79	4179.64	0.62	5.28	1.43	0.756	44.37	34.63	35.79
0.8	306.12	2.89	304.67	995.18	6232.69	7.72	4179.60	0.62	5.24	1.85	0.970	44.57	37.17	33.38
1.2	306.66	3.10	305.12	995.03	6288.35	7.66	4179.57	0.62	5.18	2.02	1.05	44.74	37.97	32.65
1.6	307.29	3.21	305.69	994.85	6360.23	7.57	4179.53	0.62	5.12	2.14	1.10	44.93	38.45	32.34
2	308.09	3.44	306.37	994.63	6446.79	7.47	4179.49	0.62	5.04	2.36	1.19	45.17	39.33	31.52
2.4	308.98	3.75	307.10	994.38	6541.66	7.36	4179.46	0.62	4.96	2.64	1.31	45.44	40.40	30.33
2.8	309.72	3.86	307.79	994.14	6631.23	7.26	4179.44	0.62	4.88	2.79	1.36	45.67	40.84	30.05

Table D.2-3 shows additional parameters that were calculated for the left surface in the middle riser for the case with circular tubes with fins.

Table D.2-3. Table Additional parameters (circular tubes with fins) associated with the left surface inside Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁷ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	317.02	13.89	310.08	993.34	6933.71	6.94	4179.43	0.62	4.65	1.10	5.10	47.67	60.01	47.60
0.4	323.35	20.21	313.24	992.17	7367.16	6.53	4179.59	0.63	4.35	1.80	7.81	49.48	67.87	57.64
0.8	325.15	21.92	314.19	991.81	7499.12	6.42	4179.68	0.63	4.26	2.02	8.60	49.99	69.77	59.88
1.2	326.12	22.55	314.84	991.55	7592.21	6.34	4179.75	0.63	4.20	2.13	8.94	50.28	70.52	60.70
1.6	327.40	23.32	315.74	991.19	7719.42	6.24	4179.85	0.63	4.13	2.27	9.38	50.65	71.44	71.44
2	328.75	24.10	316.70	990.81	7856.95	6.13	4179.99	0.63	4.05	2.43	9.84	51.05	72.39	72.39
2.4	329.81	24.58	317.52	990.47	7976.77	6.04	4180.11	0.63	3.98	2.55	10.2	51.37	73.02	73.02
2.8	331.33	25.46	318.60	990.02	8133.73	5.92	4180.29	0.64	3.89	2.75	10.7	51.81	74.05	74.05

Table D.2-4 shows additional parameters that were calculated for the right surface in the middle riser for the case with circular tubes with fins.

Table D.2-4. Additional parameters (circular tubes with fins) associated with the right surface inside Riser 2

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁷ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	317.91	14.77	310.52	993.18	6993.23	6.88	4179.44	0.62	4.60	1.19	5.46	47.92	61.22	49.24
0.4	324.41	21.26	313.77	991.96	7441.24	6.47	4179.64	0.63	4.30	1.93	8.29	49.78	69.04	59.03
0.8	326.09	22.86	314.66	991.62	7565.83	6.36	4179.73	0.63	4.22	2.14	9.04	50.26	70.77	61.05
1.2	327.12	23.56	315.35	991.35	7663.61	6.28	4179.81	0.63	4.16	2.26	9.41	50.56	71.57	71.57
1.6	328.64	24.56	316.36	990.94	7808.57	6.17	4179.94	0.63	4.07	2.45	9.97	51.01	72.72	72.72
2	330.23	25.59	317.44	990.50	7964.50	6.04	4180.10	0.63	3.99	2.65	1.06	51.47	73.88	73.88
2.4	331.48	26.24	318.35	990.12	8097.96	5.95	4180.25	0.64	3.91	2.81	11.0	51.83	74.67	74.67
2.8	333.41	27.55	319.64	989.58	8287.27	5.81	4180.49	0.64	3.82	3.08	11.8	52.39	76.08	76.08

D.3 Circular tubes without fins

Table D.3-1 shows additional parameters that were calculated for the front surface in the middle riser for the case with circular tubes without fins.

Table D.3-1. Additional parameters (circular tubes without fins) associated with the front surface on the inside of Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	307.58	4.43	305.37	994.95	6319.67	7.62	4179.55	0.62	5.15	2.92	1.50	44.98	42.17	25.21
0.4	313.13	10.01	308.13	994.03	6674.50	7.21	4179.43	0.62	4.85	7.33	3.56	46.55	54.08	38.55
0.8	314.41	11.18	308.82	993.79	6765.56	7.12	4179.42	0.62	4.78	8.42	4.02	46.92	56.02	41.71
1.2	315.62	12.12	309.56	993.53	6864.76	7.01	4179.43	0.62	4.70	9.38	4.41	47.28	57.52	43.91
1.6	317.25	13.31	310.59	993.16	7002.44	6.88	4179.45	0.63	4.60	10.7	4.93	47.75	59.37	46.47
2	318.29	13.84	311.37	992.87	7108.82	6.77	4179.48	0.63	4.52	11.5	5.19	48.07	60.23	47.53
2.4	318.66	13.65	311.83	992.70	7171.76	6.71	4179.50	0.63	4.48	11.5	5.16	48.19	60.07	47.15
2.8	321.49	15.92	313.53	992.06	7407.00	6.50	4179.62	0.63	4.32	14.3	6.18	49.01	63.26	51.34

Table D.3-2 shows additional parameters that were calculated for the back surface in the middle riser for the case with circular tubes without fins

Table D.3-2. Additional parameters (circular tubes without fins) associated with the back surface on the inside of Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	306.11	2.96	304.63	995.19	6227.15	7.73	4179.61	0.62	5.24	1.89	0.991	44.57	37.41	33.07
0.4	311.42	8.29	307.27	994.32	6562.80	7.34	4179.45	0.62	4.94	5.88	2.91	46.07	51.01	32.69
0.8	312.70	9.48	307.96	994.09	6653.40	7.24	4179.43	0.62	4.87	6.90	3.36	46.44	53.18	36.92
1.2	314.03	10.52	308.77	993.81	6758.94	7.12	4179.42	0.62	4.78	7.90	3.78	46.82	55.01	39.97
1.6	315.12	11.19	309.53	993.54	6859.98	7.02	4179.43	0.62	4.70	8.65	4.07	47.15	56.18	41.69
2	315.86	11.41	310.16	993.32	6943.95	6.93	4179.44	0.62	4.64	9.04	4.19	47.37	56.62	42.21
2.4	317.55	12.54	311.28	992.91	7095.90	6.78	4179.47	0.63	4.53	10.4	4.69	47.87	58.46	44.83
2.8	321.07	15.50	313.32	992.14	7377.66	6.53	4179.60	0.63	4.34	13.8	6.00	48.89	62.71	50.62

Table D.3-3 shows additional parameters that were calculated for the left surface in the middle riser for the case with circular tubes without fins.

Table D.3-3. Additional parameters (circular tubes without fins) associated with the left surface on the inside of Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	3.88	307.03	305.09	995.04	6285.05	7.66	4179.57	0.62	5.19	2.53	1.31	44.83	40.54	28.63
0.4	8.69	311.81	307.47	994.26	6588.67	7.31	4179.45	0.62	4.92	6.21	3.06	46.18	51.75	34.24
0.8	10.15	313.38	308.30	993.97	6697.72	7.19	4179.43	0.62	4.83	7.49	3.62	46.63	54.35	38.96
1.2	10.66	314.16	308.83	993.78	6767.76	7.11	4179.42	0.62	4.77	8.02	3.83	46.86	55.23	40.33
1.6	11.35	315.28	309.61	993.51	6870.37	7.01	4179.43	0.62	4.70	8.80	4.13	47.19	56.42	42.08
2	12.37	316.82	310.63	993.14	7008.43	6.87	4179.45	0.63	4.59	9.97	4.58	47.65	58.09	44.45
2.4	13.24	318.24	311.63	992.78	7143.47	6.74	4179.49	0.63	4.50	11.1	4.98	48.07	59.48	46.32
2.8	13.41	318.98	312.27	992.54	7232.50	6.66	4179.52	0.63	4.43	11.5	5.10	48.30	59.84	46.67

Table D.3-4 shows additional parameters that were calculated for the right surface in the middle riser for the case with circular tubes without fins.

Table D.3-4. Additional parameters (circular tubes without fins) associated with the right surface on the inside of Riser 2.

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	C _p (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	3.72	306.87	305.01	995.07	6274.99	7.67	4179.58	0.62	5.20	2.42	1.26	44.78	40.03	29.49
0.4	8.66	311.78	307.45	994.26	6586.35	7.31	4179.45	0.62	4.92	6.18	3.04	46.17	51.69	34.11
0.8	10.06	313.29	308.26	993.99	6691.83	7.19	4179.43	0.62	4.84	7.41	3.58	46.60	54.19	38.70
1.2	10.87	314.38	308.94	993.75	6782.18	7.10	4179.42	0.62	4.76	8.22	3.92	46.92	55.59	40.90
1.6	11.47	315.40	309.67	993.49	6878.39	7.00	4179.43	0.62	4.69	8.91	4.18	47.23	56.61	42.38
2	11.94	316.39	310.42	993.22	6979.92	6.90	4179.44	0.62	4.61	9.56	4.41	47.53	57.45	43.49
2.4	12.79	317.80	311.40	992.86	7113.14	6.77	4179.48	0.63	4.52	10.6	4.80	47.94	58.83	45.38
2.8	13.95	319.52	312.55	992.43	7270.20	6.62	4179.54	0.63	4.41	12.1	5.33	48.45	60.61	47.76

D.4 Square tubes with fins

Table D.4-1 shows additional parameters that were calculated for the front surface in the middle riser for the case with square tubes with fins.

Table D.4-1. Additional parameters (square tubes with fins) associated with the front surface on the inside of Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	C _p (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁷ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	310.03	6.89	306.59	994.55	6516.67	7.44	4179.48	0.62	5.02	0.476	2.39	45.91	48.19	24.72
0.4	316.09	12.95	309.62	993.51	6916.20	7.01	4179.43	0.62	4.69	1.01	4.72	47.65	58.69	45.46
0.8	317.69	14.48	310.45	993.21	7029.38	6.89	4179.44	0.62	4.61	1.16	5.35	48.11	60.85	48.48
1.2	318.34	14.80	310.94	993.03	7095.67	6.83	4179.46	0.63	4.56	1.21	5.51	48.30	61.35	49.08
1.6	319.23	15.21	311.63	992.78	7190.58	6.74	4179.49	0.63	4.50	1.27	5.72	48.58	61.99	49.81
2	320.36	15.79	312.47	992.46	7306.66	6.63	4179.54	0.63	4.42	1.36	6.03	48.92	62.87	50.85
2.4	321.48	16.32	313.32	992.14	7425.27	6.53	4179.60	0.63	4.34	1.46	6.32	49.25	63.68	51.78
2.8	322.92	17.13	314.36	991.74	7572.53	6.40	4179.70	0.63	4.24	1.59	6.74	49.68	64.83	53.12

Table D.4-2 shows additional parameters that were calculated for the back surface in the middle riser for the case with square tubes with fins.

Table D.4-2. Additional parameters (square tubes with fins) associated with the back surface on the inside of Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^6$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	303.61	0.48	303.38	995.58	6111.79	7.93	4179.73	0.61	5.39	0.289	1.56	44.09	22.25	42.12
0.4	304.81	1.67	303.97	995.40	6185.68	7.83	4179.67	0.62	5.32	1.04	5.53	44.43	31.68	38.24
0.8	305.45	2.23	304.33	995.29	6229.92	7.78	4179.63	0.62	5.28	1.41	7.44	44.61	34.47	36.30
1.2	306.00	2.46	304.77	995.15	6284.85	7.71	4179.60	0.62	5.22	1.58	8.25	44.78	35.48	35.60
1.6	306.66	2.63	305.34	994.96	6357.75	7.62	4179.55	0.62	5.16	1.73	8.92	44.98	36.25	35.13
2	307.39	2.82	305.98	994.75	6439.31	7.52	4179.51	0.62	5.08	1.90	9.65	45.20	37.06	34.62
2.4	308.13	2.97	306.64	994.53	6524.10	7.43	4179.48	0.62	5.01	2.06	10.3	45.43	37.72	34.23
2.8	309.20	3.41	307.50	994.24	6635.71	7.30	4179.44	0.62	4.92	2.44	12.0	45.76	39.36	32.66

Table D.4-3 shows additional parameters that were calculated for the left surface in the middle riser for the case with square tubes with fins.

Table D.4-3. Additional parameters (square tubes with fins) associated with the left surface on the inside of Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^7$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	317.34	14.20	310.24	993.28	7000.49	6.92	4179.44	0.62	4.63	1.13	5.23	48.01	60.45	47.95
0.4	325.45	22.32	314.30	991.76	7563.37	6.41	4179.69	0.63	4.25	2.06	8.77	50.34	70.17	60.18
0.8	327.61	24.40	315.41	991.32	7722.49	6.27	4179.81	0.63	4.15	2.35	9.76	50.96	72.35	62.69
1.2	329.32	25.78	316.43	990.92	7868.81	6.16	4179.95	0.63	4.07	2.58	10.5	51.45	73.80	64.30
1.6	330.80	26.77	317.41	990.51	8012.55	6.05	4180.09	0.63	3.99	2.77	11.1	51.88	74.89	65.45
2	332.07	27.49	318.32	990.13	8145.78	5.95	4180.25	0.63	3.92	2.94	11.5	52.25	75.72	66.30
2.4	334.07	28.91	319.61	989.59	8336.79	5.81	4180.49	0.64	3.82	3.23	12.3	52.83	77.19	67.86
2.8	336.00	30.20	320.90	989.03	8529.01	5.68	4180.76	0.64	3.72	3.53	13.1	53.38	78.53	69.26

Table D.4-4 shows additional parameters that were calculated for the right surface in the middle riser for the case with square tubes with fins.

Table D.4-4. Additional parameters (square tubes with fins) associated with the right surface on the inside of Riser 2.

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁷ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	317.34	14.20	310.24	993.28	7000.49	6.92	4179.44	0.62	4.63	1.13	5.23	48.01	60.45	47.95
0.4	325.45	22.32	314.30	991.76	7563.37	6.41	4179.69	0.63	4.25	2.06	8.77	50.34	70.17	60.18
0.8	327.61	24.40	315.41	991.32	7722.49	6.27	4179.81	0.63	4.15	2.35	9.76	50.96	72.35	62.69
1.2	329.32	25.78	316.43	990.92	7868.81	6.16	4179.95	0.63	4.07	2.58	10.5	51.45	73.80	64.30
1.6	330.80	26.77	317.41	990.51	8012.55	6.05	4180.09	0.63	3.99	2.77	11.1	51.88	74.89	65.45
2	332.07	27.49	318.32	990.13	8145.78	5.95	4180.25	0.63	3.92	2.94	11.5	52.25	75.72	66.30
2.4	334.07	28.91	319.61	989.59	8336.79	5.81	4180.49	0.64	3.82	3.23	12.3	52.83	77.19	67.86
2.8	336.00	30.20	320.90	989.03	8529.01	5.68	4180.76	0.64	3.72	3.53	13.1	53.38	78.53	69.26

D.5 Square tubes without fins

Table D.5-1 shows additional parameters that were calculated for the front surface in the middle riser for the case with square tubes without fins.

Table D.5-1. Additional parameters (square tubes without fins) associated with the front surface on the inside of Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	307.62	4.48	305.38	994.95	6362.19	7.62	4179.55	0.62	5.15	2.95	1.52	45.22	42.30	25.61
0.4	311.75	8.63	307.44	994.27	6627.47	7.31	4179.45	0.62	4.92	6.15	3.03	46.40	51.63	33.54
0.8	312.96	9.71	308.10	994.04	6714.64	7.22	4179.43	0.62	4.85	7.11	3.45	46.75	53.59	37.26
1.2	313.84	10.31	308.68	993.84	6791.50	7.13	4179.42	0.62	4.79	7.72	3.70	47.01	54.67	39.04
1.6	314.86	10.93	309.40	993.59	6886.73	7.04	4179.42	0.62	4.72	8.41	3.97	47.32	55.76	40.70
2	315.79	11.37	310.11	993.33	6982.98	6.94	4179.43	0.62	4.64	8.99	4.17	47.60	56.55	41.80
2.4	316.64	11.67	310.81	993.08	7077.46	6.85	4179.45	0.63	4.58	9.47	4.33	47.86	57.13	42.52
2.8	317.95	12.41	311.74	992.74	7205.79	6.72	4179.49	0.63	4.49	10.4	4.68	48.25	58.38	44.24

Table D.5-2 shows additional parameters that were calculated for the back surface in the middle riser for the case with square tubes without fins.

Table D.5-2. Additional parameters (square tubes without fins) associated with the back surface on the inside of Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	306.30	3.17	304.72	995.16	6279.15	7.72	4179.60	0.62	5.23	2.03	1.06	44.85	38.16	32.60
0.4	309.83	6.70	306.48	994.59	6502.51	7.45	4179.48	0.62	5.03	4.61	2.32	45.85	47.77	23.27
0.8	311.12	7.87	307.18	994.35	6594.29	7.35	4179.45	0.62	4.95	5.56	2.75	46.22	50.21	30.30
1.2	312.02	8.50	307.77	994.15	6671.49	7.26	4179.44	0.62	4.89	6.14	3.00	46.49	51.46	32.96
1.6	312.94	9.01	308.44	993.92	6758.90	7.17	4179.43	0.62	4.82	6.68	3.22	46.77	52.48	34.84
2	313.84	9.41	309.14	993.68	6851.72	7.07	4179.42	0.62	4.74	7.17	3.40	47.04	53.30	36.17
2.4	314.81	9.83	309.89	993.41	6952.94	6.97	4179.43	0.62	4.67	7.71	3.60	47.33	54.13	37.46
2.8	316.15	10.61	310.84	993.07	7082.37	6.84	4179.46	0.63	4.57	8.63	3.94	47.73	55.55	39.72

Table D.5-3 shows additional parameters that were calculated for the left surface in the middle riser for the case with square tubes without fins.

Table D.5-3. Additional parameters (square tubes without fins) associated with the left surface on the inside of Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	306.76	3.62	304.95	995.09	6307.98	7.68	4179.58	0.62	5.20	2.35	1.22	44.98	39.71	30.50
0.4	310.19	7.07	306.66	994.53	6526.25	7.42	4179.48	0.62	5.01	4.89	2.45	45.96	48.55	25.92
0.8	311.43	8.18	307.34	994.30	6614.58	7.33	4179.45	0.62	4.93	5.82	2.87	46.31	50.81	31.71
1.2	312.29	8.76	307.91	994.11	6688.96	7.24	4179.43	0.62	4.87	6.37	3.10	46.57	51.96	33.99
1.6	313.22	9.28	308.57	993.88	6777.08	7.15	4179.42	0.62	4.80	6.92	3.32	46.84	52.98	35.80
2	314.12	9.69	309.27	993.63	6870.17	7.05	4179.42	0.62	4.73	7.42	3.51	47.12	53.78	37.07
2.4	314.96	9.99	309.97	993.38	6963.66	6.96	4179.43	0.62	4.66	7.86	3.66	47.38	54.40	37.96
2.8	316.18	10.64	310.86	993.06	7084.50	6.84	4179.46	0.63	4.57	8.66	3.96	47.74	55.60	39.81

Table D.5-4 shows additional parameters that were calculated for the right surface in the middle riser for the case with square tubes without fins.

Table D.5-4. Additional parameters (square tubes without fins) associated with the right surface on the inside of Riser 2.

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	306.76	3.63	304.95	995.09	6308.15	7.68	4179.58	0.62	5.20	2.35	1.22	44.98	39.72	30.49
0.4	310.20	7.07	306.66	994.53	6526.51	7.42	4179.48	0.62	5.01	4.90	2.45	45.96	48.56	25.95
0.8	311.43	8.19	307.34	994.30	6614.80	7.33	4179.45	0.62	4.93	5.82	2.87	46.31	50.82	31.72
1.2	312.29	8.77	307.91	994.11	6689.09	7.24	4179.43	0.62	4.87	6.37	3.10	46.57	51.96	34.00
1.6	313.22	9.29	308.58	993.88	6777.15	7.15	4179.42	0.62	4.80	6.92	3.32	46.85	52.98	35.80
2	314.12	9.69	309.27	993.63	6870.25	7.05	4179.42	0.62	4.73	7.42	3.51	47.12	53.78	37.08
2.4	314.97	9.99	309.97	993.38	6963.75	6.96	4179.43	0.62	4.66	7.86	3.66	47.38	54.41	37.96
2.8	316.18	10.65	310.86	993.06	7084.53	6.84	4179.46	0.63	4.57	8.66	3.96	47.74	55.60	39.81

D.6 Rectangular tubes with fins

Table D.6-1 shows additional parameters that were calculated for the front surface in Riser 2 for the case with rectangular tubes with fins.

Table D.6-1. Additional parameters (rectangular tubes with fins) associated with the front surface on the inside of Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	307.38	4.33	305.22	995.00	2962.83	7.64	4179.56	0.62	5.17	2.83	1.47	10.50	41.86	41.64
0.4	314.69	11.58	308.90	993.76	3186.56	7.10	4179.42	0.62	4.77	8.74	4.17	12.41	56.63	56.43
0.8	314.06	10.61	308.77	993.81	3177.89	7.12	4179.42	0.62	4.78	7.96	3.81	12.32	55.14	54.93
1.2	314.35	10.34	309.19	993.66	3204.07	7.06	4179.42	0.62	4.74	7.89	3.74	12.53	54.81	54.59
1.6	315.10	10.42	309.90	993.41	3248.70	6.97	4179.43	0.62	4.67	8.17	3.81	12.90	55.06	54.83
2	316.08	10.68	310.74	993.10	3302.02	6.86	4179.45	0.63	4.58	8.65	3.96	13.35	55.64	55.38
2.4	317.10	11.04	311.58	992.80	3355.93	6.75	4179.49	0.63	4.50	9.23	4.15	13.82	56.35	56.07
2.8	318.48	11.88	312.54	992.44	3418.16	6.62	4179.54	0.63	4.41	10.3	4.54	14.38	57.77	57.47

Table D.6-2 shows additional parameters that were calculated for the back surface in Riser 2 for the case with rectangular tubes with fins.

Table D.6-2. Additional parameters (rectangular tubes with fins) associated with the back surface on the inside of Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	C _p (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^5$ (-)	Ra $\times 10^5$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	303.34	0.29	303.20	995.64	2844.91	7.96	4179.75	0.61	5.41	1.73	9.34	9.56	19.35	18.53
0.4	304.65	1.53	303.88	995.43	2884.29	7.85	4179.68	0.62	5.33	9.51	50.7	9.86	30.90	30.56
0.8	304.75	1.29	304.11	995.36	2897.41	7.81	4179.66	0.62	5.30	8.05	42.7	9.96	29.43	29.05
1.2	304.87	0.86	304.44	995.25	2917.22	7.76	4179.62	0.62	5.26	5.45	28.7	10.10	26.30	25.80
1.6	305.17	0.48	304.93	995.09	2945.69	7.68	4179.58	0.62	5.21	3.12	16.2	10.30	22.44	21.69
2	305.58	0.18	305.49	994.91	2978.81	7.60	4179.54	0.62	5.14	1.22	6.25	10.55	17.32	15.90
2.4	306.09	0.03	306.08	994.72	3013.85	7.51	4179.51	0.62	5.07	0.225	1.14	10.81	11.14	4.91
2.8	306.81	0.21	306.71	994.51	3051.84	7.42	4179.47	0.62	5.00	1.44	7.20	11.10	17.96	16.41

Table D.6-3 shows additional parameters that were calculated for the left surface in Riser 2 for the case with rectangular tubes with fins.

Table D.6-3. Additional parameters (rectangular tubes with fins) associated with the left surface on the inside of Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	308.44	5.39	305.75	994.83	2994.12	7.56	4179.53	0.62	5.11	3.60	1.84	10.75	44.69	44.48
0.4	314.36	11.25	308.74	993.82	3176.14	7.13	4179.42	0.62	4.79	8.44	4.04	12.32	56.10	55.90
0.8	316.20	12.74	309.83	993.43	3244.70	6.98	4179.43	0.62	4.67	9.97	4.66	12.93	58.44	58.23
1.2	313.62	9.61	308.82	993.79	3181.28	7.12	4179.42	0.62	4.78	7.23	3.45	12.33	53.56	53.34
1.6	314.46	9.77	309.57	993.52	3228.34	7.01	4179.43	0.62	4.70	7.57	3.56	12.72	53.97	53.73
2	315.53	10.13	310.46	993.20	3284.46	6.89	4179.44	0.62	4.61	8.12	3.74	13.19	54.72	54.46
2.4	316.68	10.63	311.37	992.87	3342.62	6.77	4179.48	0.63	4.52	8.81	3.98	13.70	55.67	55.39
2.8	318.52	11.92	312.56	992.43	3419.54	6.62	4179.54	0.63	4.41	10.3	4.56	14.39	57.84	57.54

Table D.6-4 shows additional parameters that were calculated for the right surface in Riser 2 for the case with rectangular tubes with fins.

Table D.6-4. Additional parameters (rectangular tubes with fins) associated with the right surface on the inside of Riser 2.

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	308.88	5.83	305.97	994.76	3007.34	7.53	4179.51	0.62	5.09	3.93	2.00	10.86	45.77	45.57
0.4	315.37	12.25	309.24	993.64	3207.50	7.06	4179.42	0.62	4.73	9.37	4.44	12.60	57.65	57.45
0.8	316.32	12.86	309.89	993.41	3248.30	6.97	4179.43	0.62	4.67	10.1	4.70	12.96	58.61	58.40
1.2	314.09	10.07	309.05	993.71	3195.80	7.08	4179.42	0.62	4.75	7.65	3.64	12.46	54.36	54.14
1.6	315.18	10.49	309.93	993.40	3251.06	6.96	4179.43	0.62	4.66	8.24	3.84	12.92	55.19	54.95
2	316.25	10.85	310.82	993.07	3307.50	6.84	4179.45	0.63	4.57	8.82	4.03	13.41	55.92	55.66
2.4	317.38	11.32	311.72	992.74	3365.04	6.73	4179.49	0.63	4.49	9.51	4.27	13.91	56.80	56.52
2.8	319.11	12.50	312.86	992.32	3438.58	6.58	4179.56	0.63	4.38	11.0	4.80	14.58	58.72	58.42

D.7 Rectangular tubes without fins

Table D.7-1 shows additional parameters that were calculated for the front surface in Riser 2 for the case with rectangular tubes without fins.

Table D.7-1. Additional parameters (rectangular tubes without fins) associated with the front surface on the inside of Riser 2.

z (m)	T _{wall} (front) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	308.03	4.88	305.59	994.88	2984.77	7.58	4179.54	0.62	5.13	3.24	1.66	10.67	43.40	43.18
0.4	314.71	11.56	308.93	993.75	3188.25	7.10	4179.42	0.62	4.76	8.74	4.16	12.43	56.61	56.41
0.8	315.27	12.10	309.22	993.65	3206.29	7.06	4179.42	0.62	4.73	9.25	4.38	12.59	57.43	57.23
1.2	314.29	10.39	309.10	993.69	3198.65	7.08	4179.42	0.62	4.75	7.90	3.75	12.49	54.87	54.65
1.6	315.26	10.95	309.79	993.45	3241.99	6.98	4179.43	0.62	4.68	8.55	4.00	12.86	55.86	55.63
2	316.31	11.57	310.53	993.18	3288.83	6.88	4179.45	0.62	4.60	9.29	4.28	13.27	56.93	56.68
2.4	317.83	12.59	311.53	992.81	3352.65	6.75	4179.48	0.63	4.51	10.5	4.73	13.84	58.59	58.33
2.8	318.81	13.03	312.29	992.53	3401.84	6.65	4179.53	0.63	4.43	11.2	4.96	14.27	59.34	59.07

Table D.7-2 shows additional parameters that were calculated for the back surface in Riser 2 for the case with rectangular tubes without fins.

Table D.7-2. Additional parameters (rectangular tubes without fins) associated with the back surface on the inside of Riser 2.

z (m)	T _{wall} (back) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	306.27	3.12	304.71	995.16	2932.86	7.72	4179.60	0.62	5.23	2.00	1.05	10.25	38.01	37.76
0.4	311.41	8.26	307.05	994.40	3072.36	7.37	4179.46	0.62	4.97	5.80	2.88	11.42	50.90	50.71
0.8	311.03	7.86	307.10	994.38	3075.66	7.36	4179.46	0.62	4.96	5.54	2.75	11.44	50.18	49.98
1.2	311.67	7.76	307.79	994.15	3117.50	7.26	4179.44	0.62	4.89	5.62	2.74	11.77	50.11	49.90
1.6	312.44	8.12	308.38	993.94	3153.90	7.18	4179.43	0.62	4.82	6.01	2.90	12.07	50.89	50.66
2	313.37	8.62	309.06	993.71	3196.19	7.08	4179.42	0.62	4.75	6.55	3.11	12.43	51.92	51.68
2.4	314.32	9.09	309.78	993.45	3241.13	6.98	4179.43	0.62	4.68	7.09	3.32	12.81	52.86	52.61
2.8	315.75	9.98	310.76	993.09	3303.70	6.85	4179.45	0.63	4.58	8.09	3.70	13.35	54.54	54.27

Table D.7-3 shows additional parameters that were calculated for the left surface in Riser 2 for the case with rectangular tubes without fins.

Table D.7-3. Additional parameters (rectangular tubes without fins) associated with the left surface on the inside of Riser 2.

z (m)	T _{wall} (left) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	$\mu \times 10^{-4}$ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr $\times 10^6$ (-)	Ra $\times 10^7$ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	305.25	2.10	304.20	995.33	2902.91	7.80	4179.65	0.62	5.29	1.32	0.698	10.01	33.85	33.55
0.4	309.07	5.92	306.11	994.71	3015.73	7.51	4179.50	0.62	5.07	4.01	2.03	10.93	46.00	45.79
0.8	310.74	7.57	306.96	994.43	3066.86	7.38	4179.46	0.62	4.98	5.30	2.64	11.36	49.60	49.40
1.2	308.18	4.28	306.04	994.73	3011.82	7.52	4179.51	0.62	5.08	2.89	1.47	10.87	41.83	41.58
1.6	308.83	4.51	306.57	994.56	3043.65	7.44	4179.48	0.62	5.02	3.11	1.56	11.12	42.56	42.31
2	309.58	4.84	307.17	994.36	3079.60	7.35	4179.46	0.62	4.95	3.42	1.69	11.41	43.53	43.27
2.4	310.48	5.24	307.85	994.12	3121.61	7.25	4179.44	0.62	4.88	3.80	1.85	11.75	44.67	44.40
2.8	311.62	5.84	308.69	993.83	3173.52	7.13	4179.42	0.62	4.79	4.38	2.10	12.18	46.24	45.96

Table D.7-4 shows additional parameters that were calculated for the right surface in Riser 2 for the case with rectangular tubes without fins.

Table D.7-4. Additional parameters (rectangular tubes without fins) associated with the right surface on the inside of Riser 2.

z (m)	T _{wall} (right) (K)	dT (K)	T _{film} (K)	ρ (kg/m ³)	Re (-)	μ x10 ⁻⁴ (kg/m-s)	Cp (J/kg-K)	k (w/m-K)	Pr (-)	Gr x10 ⁶ (-)	Ra x10 ⁷ (-)	Nu (forced) (-)	Nu (free) (-)	Nu (mixed) (-)
0	305.57	2.42	304.36	995.28	2912.39	7.77	4179.63	0.62	5.27	1.53	0.809	10.09	35.29	35.02
0.4	309.73	6.58	306.44	994.60	3035.60	7.46	4179.49	0.62	5.03	4.52	2.27	11.10	47.50	47.30
0.8	310.54	7.37	306.85	994.46	3060.63	7.40	4179.47	0.62	4.99	5.14	2.56	11.31	49.18	48.98
1.2	308.51	4.60	306.21	994.68	3021.61	7.49	4179.50	0.62	5.06	3.13	1.58	10.95	42.76	42.52
1.6	309.30	4.98	306.81	994.48	3057.80	7.40	4179.47	0.62	4.99	3.47	1.73	11.24	43.84	43.60
2	310.08	5.34	307.42	994.27	3094.83	7.31	4179.45	0.62	4.93	3.80	1.87	11.54	44.84	44.58
2.4	311.02	5.78	308.12	994.03	3138.23	7.21	4179.43	0.62	4.85	4.24	2.06	11.89	46.02	45.75
2.8	312.23	6.46	309.00	993.73	3192.68	7.09	4179.42	0.62	4.76	4.89	2.33	12.35	47.67	47.40

APPENDIX E PHILOSOPHIES IN COUPLING

E.1 Philosophies in coupling

The coupling of 1D and 3D codes can be done in a number of ways. The way in which the domain to be simulated is deconstructed into a 3D or 1D domain is to the user's discretion. Since the different domains have unilateral interaction the optimal approach should be obtained to gain full advantage of the strengths of each of the coupled codes.

There are several different philosophies in how the coupling is implemented:

1. Directly coupled -parallel

In this configuration the 3D code would be used to model the convection, radiation and conduction in the heater box, but not the convection within the riser tubes. The 1D code would be used to simulate the loop including the parts that pass through the heated section. The 3D code would send the heat flux on the inside of the riser tube to the 1D code, which would in turn send the resulting temperature to the 3D code.

The advantage of this method is that computational resources are spared in terms of the modelling of the flow in the riser tubes. One disadvantage is the model could be time consuming to set up, depending on the discretization that the user chooses.

2. Indirectly coupled -parallel

In this configuration the 3D code would be used to simulate the convection, conduction and radiation in the heater box as well as the convection and fluid flow within the risers in the heater box. The 1D code would simulate the entire loop, including the part that runs through the heated section. The 3D code would send an outlet temperature or flux to the 1D code. The 1D code would send an inlet mass flow rate, inlet temperature and outlet pressure. An alternative would also be that the outputs from the 3D code could be used to calibrate the pressure drop in the 1D code.

The advantage of this method is that the setup would be less time consuming than the previously mentioned method. Discrepancies are easier to identify since both codes would generate boundary conditions on a feedback loop.

3. Directly coupled-series

In this configuration, the 3D code would be used to model the heater box in its entirety and the 1D code to be connected at the inlet and outlet of the 3D code.

The disadvantage of this method would be the loss in stability due to the series coupling. The initial guess values would be important and large changes in the exchanged parameters could lead to instabilities. This method could also be the most explicit and possibly the shortest solving times.

APPENDIX F ARTICLE TO BE SUBMITTED

Evaluation of the frictional characteristics of the test section of an experimental reactor cavity cooling system using 3D and 1D CFD approaches

C.G. du Toit, P.F. Niemand¹
North-West University
Private Bag X6001, Potchefstroom 2520, South Africa
phone: +27-18-299-1322, jat.dutoit@nwu.ac.za

¹North-West University, Private Bag X6001, Potchefstroom 2520, South Africa

Abstract – RCCS experimental facilities are used to gain a fundamental understanding of the governing physical phenomena and the data from the experiments can be used to validate numerical modeling approaches. In this study the frictional characteristics of an experimental RCCS is evaluated using 1D and 3D CFD approaches. It is shown that more information must be provided for the frictional losses and the form factors to be fully evaluated. During the evaluation of the results of the steady-state natural circulation tests it was concluded that the form factors of the test section are dependent on the mass flow rate. 1D and 3D CFD models of the test section were used to determine the primary frictional loss factors and the total loss factors. The difference between the two sets of loss factors provided the form loss factors. The results confirmed the conclusion that the form factors are dependent on the mass flow rate.

1. INTRODUCTION

The safety of high temperature reactors (HTRs) under normal and upset conditions is of utmost importance. In this regard the reactor cavity cooling system (RCCS) plays an important role to remove the heat released from the surface of the reactor pressure vessel (RPV) passively under all conditions [1]. A crucial function of the RCCS is to maintain the temperature of the containment below the maximum allowable temperature.

Due to the complexity of the phenomena and the geometry the RCCS cannot be designed analytically and recourse must be taken to experimental and numerical methods.

Experimental facilities typically are scale models and require a fundamental understanding of the governing physical phenomena and associated scaling laws for the design of the facility and the evaluation of the results [2]. Experimental facilities are therefore built to identify and evaluate governing physical phenomena, derive where possible correlations and study the behaviour of the system [3-6].

An RCCS can also be simulated numerically to evaluate the design and the behaviour of the system. However, detailed 3D computational fluid dynamics models can only be used to simulate selected parts of the system [7-8]. The full system can typically only be modelled employing a 1D system computational fluid dynamic (CFD) approach [9-12].

The data obtained from experiments can also be used to validate the numerical methodologies [13]. Similarly numerical simulations can be used to evaluate experimental results.

This paper describes 3D and 1D numerical simulations that were performed to evaluate selected results from the single phase steady-state tests that were performed by Lisowski [14]. In particular the open loop frictional losses obtained from an isothermal separate effects test and the steady-state natural circulation results for various heat loads were evaluated using a 1D approach. Based on the results obtained for the latter tests, 3D and 1D simulations were employed to evaluate the frictional characteristics of the test section for isothermal conditions in more detail.

II. LAYOUT OF TEST FACILITY

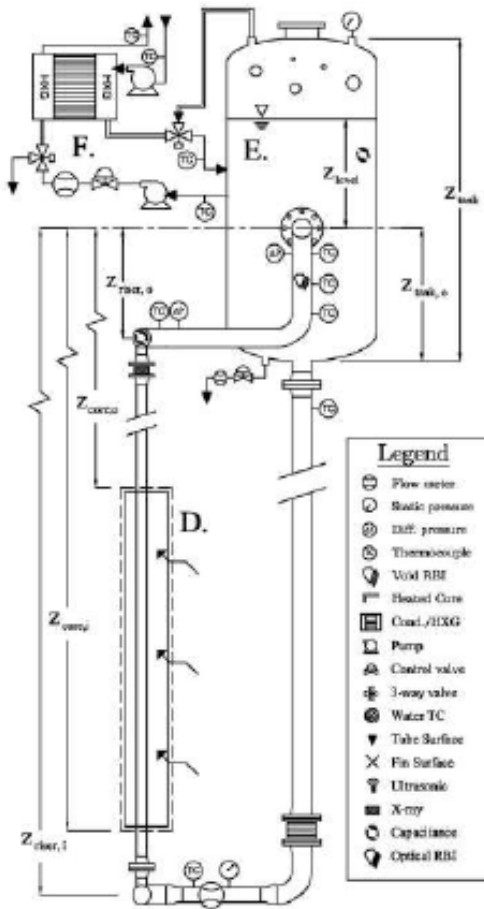


Fig. 1: Layout of experimental test facility.

The layout of the experimental RCCS test facility constructed by Lisowski [14] is shown in Fig. 1. The facility consists of a storage tank that feeds water under gravity into a down-comer which also contains a flow meter. The test section consists of the lower manifold, three risers enclosed in a heater box and the upper manifold. The lower manifold splits the flow into the three riser tubes and the upper manifold collects the flow from the risers and feeds the water into the chimney which takes the flow back to the storage tank. In the test section 34 radiant heaters exposed the riser tubes to a heat flux which heated the water and resulted in buoyancy driven natural circulation occurring in the system. An auxiliary cooling loop connected to the storage tank rejected the heat added to the water and ensured a constant water temperature at the inlet of the down-comer. Temperature and pressure sensors, as well as additional flow meters were placed at strategic positions to measure the temperatures, pressures and flow rate. The test section represented a 5° radial sector of an actual RCCS design. A 3D

CFD model employing Fluent was constructed of the manifolds and risers and a 1D system CFD model using RELAP5 was constructed of the full facility [15].

In this study the 1D system CFD model shown in Fig. 2 was constructed employing Flownex SE [16]. The dimensions of the components were taken from the engineering drawings given by Lisowski [14]. The storage tank was divided into three sections, i.e. an upper reservoir (RU) from the inlet of the chimney to the water level, a middle reservoir (RM), and a lower reservoir (RL) from the bottom of the RM to the outlet at the bottom of the tank as shown in Fig. 2. In the absence of the air-cooled heat exchanger loop the value of the outlet temperature of the RM is fixed to the cold flow temperature. As no values were given by Lisowski [2013] and Lisowski et al. [2013] for the form factors (K factors) of the components appropriate values were obtained from Streeter and Wylie [17]. All bends were assigned a value of 0.25; the expansion coupling in the down-comer a value of 0.3; the T-junctions in the manifolds a value of 2.0; the connections between the T-junctions a value of 0.2; the diffuser upstream of the flow meter a value of 0.18 and the diffuser downstream of the flow meter a value of 0.13; and the inlet and outlet form losses associated with storage tank were assigned values of 1.0 and 0.5 respectively. The roughness of all the components was taken to be 15 μm . The upper node of the upper reservoir (RU) represents the water level in the reservoir and a total pressure of 100 kPa was specified at the point. Due to the layout (and the incompressible nature of the water) the mass flow rate in the RU will be zero. A third of the total heat input is directly applied to each of the three risers as shown in Fig. 2. Lumped values for the form losses could also be applied to the horizontal section (DH1) of the down-comer, the sections of the risers above the heater box (TRO1, TRO2, TRO3), and the vertical section (CV) of the chimney as shown in Fig. 2.

III. EVALUATION OF FRICTIONAL CHARACTERISTICS

Three aspects were evaluated. In the first place the open loop frictional losses obtained from an isothermal separate effects test were evaluated using the 1D Flownex model. Secondly the steady-state natural circulation results for various heat loads were evaluated also using the 1D Flownex model. Based on the results obtained for the latter tests, 3D simulations using Fluent and 1D simulations using Flownex were employed to evaluate the frictional characteristics of the test section for isothermal conditions in more detail.

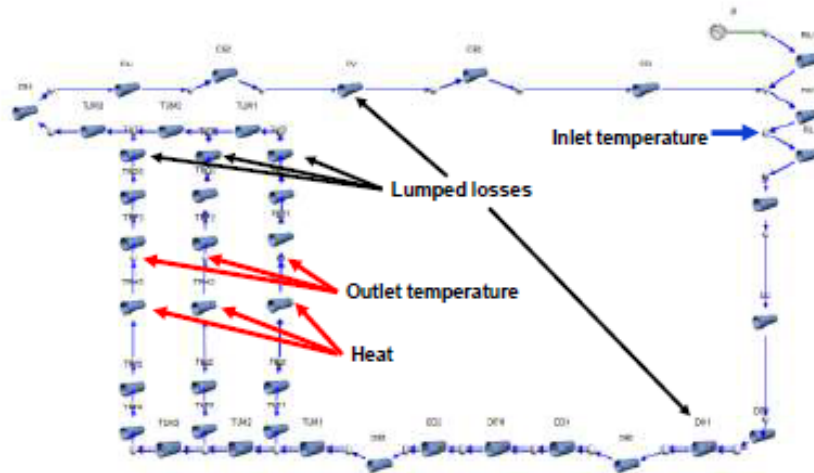


Fig. 2: Layout of Flownex SE 1D system CFD model of experimental test facility/

III.A. Isothermal frictional losses

Lisowski [14] performed iso-thermal forced flow separate effects tests and measured the pressure drop as a function of the flow rate between the outlet and the inlet of the tank. To determine the total and the segment losses multiple differential pressure taps were placed at selected locations along the loop. The frictional losses and lumped form loss factors for the down-comer / lower network (DC), the test section (TS) and the chimney (CH) for a flow rate of 5.8 GPM (0.367 l/s) found by Lisowski (2013) are tabulated in Table 1. No value is given for the temperature of the water during the test. In the evaluation of the results the water temperature was assumed to be 30 °C which gives a density of $\rho = 995.6 \text{ kg/s}$.

Table 1: Frictional losses and K factors [14].

Segment	Loss (Pa)	K factor
DC	2.75	2.78
TS	8.63	8.72
CH	3.07	3.10

Lisowski [14] gives no details of how the K factors were determined from the measurements and whether the frictional loss for each segment also includes the effect of the form loss.

To assess the results and the validity of the Flownex partially, four sets of calculations were performed. In the first case the primary and form losses for each segment were calculated using an Excel sheet. The pressure drop Δp over the segment was calculated using [17]:

$$\Delta p = \left(\frac{fL}{D} + K \right) \frac{\rho V^2}{2} \quad (\text{eq. 1})$$

Where f is the friction coefficient; L the length of the segment; D the diameter of the pipe; and V the average velocity in the pipe. The friction coefficient was obtained using the modified Colebrook-White correlation [17]:

$$f = \frac{0.25}{\left[\log \left(\frac{\epsilon}{3.7D} + \frac{5.74}{\text{Re}^{0.9}} \right) \right]^2} \quad (\text{eq. 2})$$

Where ϵ is the roughness, $\text{Re} = (\rho V D) / \mu$ the Reynolds number and μ the dynamic viscosity.

In the second case representative pipe models of the three segments were constructed using Flownex. The density, friction coefficient and velocity were extracted from the results of each segment and used in an Excel sheet embedded in Flownex to calculate the primary, form and total frictional losses for each segment. The total frictional loss obtained by Flownex for each segment was also extracted from the results for each segment and compared with the calculated total frictional loss.

In the third and fourth cases the full Flownex model was employed. In case 3 only primary losses were considered and all form factors were assumed to be zero. In case 4 lumped values for the form factors corresponding to the values given in Table 1 were specified at the positions mentioned previously. The loss, primary loss for case 3 and total loss for case 4, for each segment of the system was obtained by extracting and summing the losses obtained by Flownex for the pipe components comprising the segment. For the test section the flow path through riser 3 was chosen as the representative flow path.

The resulting primary and total losses for the segments obtained in the simulations are tabulated in Table 2. The form losses were obtained by subtracting the primary losses from the total losses.

Table 2: Primary, form and total losses.

Segment	Loss (Pa)		
	Primary	Form	Total
DC	3.043	2.743	5.786
TS	6.592	15.721	22.313
CH	0.713	3.059	3.771

The values for the losses obtained in the Excel sheet for case 1, as well as from the Excel sheet embedded in Flownex for case 2 were in excellent agreement with the losses obtained for cases 3 and 4. It can thus be concluded that the results verifies the capability to calculate the losses.

However, when the results in Table 2 are compared with the corresponding results in Table 1, it can be seen that the computed form losses for the DC and the CH are in agreement with the experimental frictional losses, whilst the form loss in Table 2 for the TS is considerably larger than the experimental value. The computed primary losses in Table 2 for the DC is also larger than the corresponding value in Table 1, whilst the primary loss in Table 2 for the TS is less than the corresponding value in Table 1.

It can therefore be concluded that more information must be provided in order for the results given in Table 1 to be evaluated fully.

III.B. Steady-state natural circulation

Lisowski [14] performed steady-state tests at various power levels to investigate the effects on parameters such as temperature rise, mass flow rate and heat loss. All the tests were kept at steady-state for a minimum of two hours. After a test was completed a full energy balance was performed to determine how much was transferred to the RCCS and how much was being lost to the environment. In the Flownex simulations no energy losses to the environment were considered and only the heat transferred to the risers was taken into account. The relevant data for the tests are summarized in Table 3.

Table 3: Data of single phase steady-state tests [14].

	Q_r (kW)	T_{in} (°C)	T_{out} (°C)	$\dot{m}$ (kg/s)
Run007	3.778	30.25	32.67	0.37
Run008	5.979	29.98	33.16	0.45
Run006	8.402	29.99	33.82	0.52
Run009	10.646	29.90	34.27	0.58
Run029	12.254	30.49	34.88	0.67

Where Q_r is the heat added to the risers, T_{in} inlet temperature of the water to the test section, T_{out} the outlet temperature of the water from the test section, and $\dot{m}$ is the steady state mass flow rate in the RCCS loop. In the Flownex simulations the outlet

temperature of the RM was assigned the relevant outlet temperature from Table 3. Either the relevant heat added to the risers given in Table 3 was equally divided and assigned to each of the heated riser pipe components, or the relevant outlet temperature given in Table 3 was assigned to outlet node of the each of the heated riser pipe components.

To characterize the system and develop an understanding of the behaviour and performance of the system various simulations were performed in which the specification of the form loss factors were varied. In all cases the primary losses were accounted for by Flownex based on the geometric details of the pipes and the relevant associated flow parameters.

In the first case no form losses were specified and the relevant heat inputs were specified. The results of the simulations are summarized in Table 4.

Table 4: Results for case 1.

	$\dot{m}$ (kg/s)	T_{out} (°C)
Run007	0.5048	32.04
Run008	0.5971	32.38
Run006	0.6846	32.93
Run009	0.7489	33.30
Run029	0.8086	34.12

A comparison of the relevant result in Tables 3 and 4 shows that the mass flow rates are larger the mass flow rates measured in the experiments and that the outlet temperatures are lower than the outlet temperatures measured in the experiments.

In the second case no form losses were again specified and the relevant outlet temperatures were specified. The results of the simulations are summarized in Table 5.

Table 5: Results for case 2.

	$\dot{m}$ (kg/s)	Q_{out} (kW)
Run007	0.6165	-6.234
Run008	0.7206	-9.576
Run006	0.8127	-13.008
Run009	0.8794	-16.059
Run029	0.9079	-16.654

Q_{out} is the heat extracted, therefore the negative sign, from the water in RM, to obtain the required outlet temperature. A comparison of the relevant result in Tables 3 and 5 shows that the mass flow rates are larger than the mass flow rates measured in the experiments and that the heat extracted are more than the heat loads applied in the experiments. From the fact that the mass flow rates in Tables 4 and 5 are considerably larger than the corresponding mass flow rates given in Table 3 it can be deduced that the resistance of the system is too low. As a result the

outlet temperatures for the various runs in case 1 are too low and the heat rejected for the various runs in case 2 is also considerably larger than the heat inputs given in Table 3.

In case 3 the form factors as discussed in section I were specified. A lumped form factor of 4.72 was also specified for the sections of the risers above the heater box (TRO1, TRO2, TRO3). The value was chosen such that the sum of the form factors for the test section was equivalent to the lumped value given in Table 1. It was also argued that the sum of the distributed form factors for the down-comer and the chimney are equivalent to the lumped values given in Table 1 and therefore no additional lumped values were specified. The relevant heat inputs were applied to the heated sections of the riser and results obtained are summarized in Table 6.

Table 6: Results for case 3.

	$\dot{m}$ (kg/s)	T_{out} (°C)
Run007	0.3632	32.74
Run008	0.4217	33.37
Run006	0.4756	34.22
Run009	0.5171	34.83
Run029	0.3632	32.74

It can be seen that the results for run007 are in good agreement with the experimental values given in Table 3, but as the heat input increases the simulated values deviate gradually more from the experimental values. It should be noted that the mass flow rate for Run007 is in agreement with the mass flow rate used to evaluate the frictional losses of the facility given in Table 1.

In case 4 the values of the lumped form factors specified for TRO1, TRO2 and TRO3 (risers) were varied or calibrated to obtain the mass flow rates measured in the experiments given in Table 3. The relevant heat inputs were applied to the heated sections of the riser and results obtained are summarized in Table 7.

Table 7: Results for case 4.

	$\dot{m}$ (kg/s)	T_{out} (°C)	K_{riser}
Run007	0.3700	32.69	3.74
Run008	0.4500	33.16	1.49
Run006	0.5201	33.86	0.71
Run009	0.5769	34.32	0.00
Run029	0.6170	35.24	0.00

It can be seen in Table 7 that for Run007, Run008 and Run006 the measured experimental mass flow rates and outlet temperatures could be obtained. As the mass flow increases – heat input increases – the value of the lumped form factor decreases. However, the required mass flow rates

and outlet temperatures for Run009 and Run029 could not be obtained when the minimum value of 0.0 was specified for the lumped form factor.

In order to address the fact that the required mass flow rate could not be obtained for Run029 in case 4, the form factors for all bends were reduced from 0.25 to 0.20, the form factor for the T-junctions were all reduced from 2.0 to 0.5 and form factors for the connections between the T-junctions were all reduced from 0.2 to 0.05 in case 5. The original values for the form factors were considered to be overly conservative and the improved values are considered to be more representative of the streamlined geometry of the manifolds. Again the values of the lumped form factors specified for TRO1, TRO2 and TRO3 (risers) were calibrated to obtain the mass flow rates measured in the experiments given in Table 3. The relevant heat inputs were applied to the heated sections of the riser and results obtained are summarized in Table 8.

Table 8: Results for case 5.

	$\dot{m}$ (kg/s)	T_{out} (°C)	K_{riser}
Run007	0.3701	32.69	7.120
Run008	0.4500	33.16	4.870
Run006	0.5200	33.86	4.092
Run009	0.5800	34.29	3.190
Run029	0.6700	34.87	0.835

It can be seen in Table 8 that for all the runs the required mass flow rate could be obtained.

In the last case, case 6, all the local form factors were set to zero and only lumped values for the form factors were applied to the horizontal section (DH1) of the down-comer, the sections of the risers above the heater box (TRO1, TRO2, TRO3), and the vertical section (CV) of the chimney. The relevant heat inputs were applied to the heated sections of the riser and results obtained are summarized in Table 9.

Table 9: Results for case 6.

	K_{down}	K_{riser}	K_{chim}
Run007	2.78	6.900	3.10
Run008	2.78	4.640	3.10
Run006	2.78	3.868	3.10
Run009	2.78	2.966	3.10
Run029	2.50	1.365	2.00

For Run007, Run008, Run006 and Run009 the values for K_{down} and K_{chim} were kept constant at the values given in Table 1 and only the value of K_{riser} was varied to obtain the required mass flow rates. For Run029 a value less than 1.0 had to be specified for K_{riser} , but no converged solution could be obtained. The values for K_{down} and K_{chim} were decreased and a converged solution could be

obtained for the values for the lumped form factors shown for Run029 in Table 9.

From the results obtained for the various cases discussed in this section it can be deduced that the suggested strategy to model the experimental facility in the absence of more detailed information is the approach followed in case 5. The appropriate form factors should be specified for the relevant components and augmented with lumped form factors specified for TRO1, TRO2 and TRO3. It will also appear that value of the lumped form factor associated with the test section is a function of the mass flow rate. It is recommended that a comprehensive study be performed to characterize the experimental facility. This is important if it is intended to employ the facility as a basis or vehicle to study the numerical modelling of natural circulation flows for various conditions.

III.C. Test section losses

The results shown in Tables 7, 8 and 9 suggest that the form factor(s) for the upper and lower manifolds might be dependent on the mass flow rate. To investigate the form factor(s) additional isothermal simulations were performed on only the test section (manifolds and the risers) shown in Fig. 3.

Two models were developed. In the first a 1D Flownex model of the test section shown in Fig. 4 was extracted from the full model shown in Fig. 2 and adapted by removing all the form factors, as well as the heat inputs. The model therefore only accounted for the primary frictional losses that occur in the manifolds and risers assuming the roughness to be 0 μm . A fixed mass flow rate and temperature was assigned at the inlet to the lower manifold according to the mass flow rates given in Table 3 and a pressure was assigned at the outlet of the top manifold.

The second model consisted of a 3D model of the test section shown in Fig. 3 developed in Fluent that would automatically account for the primary frictional losses and the form losses. The mesh consisted of 7 660 154 elements and 2 815 984 nodes. The maximum face size of 5 mm was used for the riser tubes with the size function set to proximity and curvature, with the growth rate set at 1.05. For the manifolds a maximum face size of 3 mm was used with the sizing function set to the curvature and a growth rate of 1.0. Eight prism layers were used on all of the boundaries. A first layer thickness of 0.3 mm was specified for the riser tubes and 0.2 mm for the manifolds. These ensured $y^+ < 5$ values on all surfaces. The orthogonal quality was found to have a value of 0.825, which was deemed acceptable [18]. The Fluent simulation used the standard $k-\epsilon$ turbulence model with scalable wall functions. The SIMPLE algorithm was used with second order spatial discretization for pressure,

momentum and energy while first order upwind discretization was applied for turbulent kinetic energy and turbulent dissipation rate. The simulation was run until the all residual criteria were below 10^{-4} , which typically occurred after 1200 iterations.

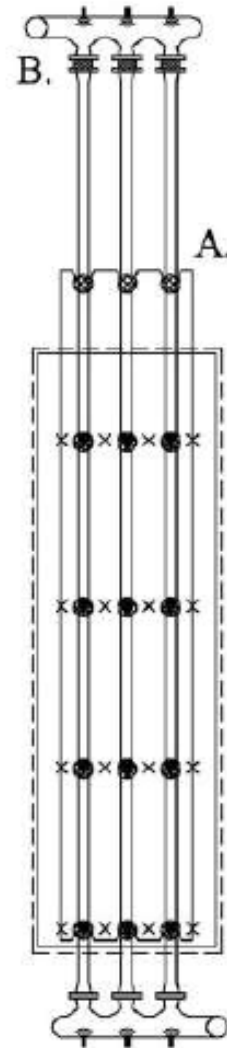


Fig. 3: Layout of test section [14].

Fixed values of $\rho = 998 \text{ kg/m}^3$ and $\mu = 1.03 \times 10^{-3} \text{ Pa.s}$ for the density and dynamic viscosity respectively were employed in both the Flownex and Fluent simulations. The pressure drops due to the frictional and form losses associated with the experimental mass flow rates listed in Table 3 were determined for both models. The pressure drop over the test section can be expressed as:

$$\Delta p = F' \frac{\dot{m}^2}{2\rho A_m^3} \quad (\text{eq. 3})$$

F' is a loss factor that is representative of all the relevant losses associated with all the components comprising the flow path considered and A_m is the cross sectional area of the inlet of the lower manifold which is the same as the cross sectional area of the outlet of the upper manifold.



Fig. 4: 1D model of test section.

We have that the total loss factor F'_t is defined as:

$$F'_t = \sum_i \left(\left[\frac{f_i L_i}{D_i} + K_i \right] \frac{A_m}{A_i} \right) \quad (\text{eq. 4})$$

Whilst the primary friction loss factor F'_f is defined as:

$$F'_f = \sum_i \left(\left[\frac{f_i L_i}{D_i} \right] \frac{A_m}{A_i} \right) \quad (\text{eq. 5})$$

The form loss factor F'_K is therefore defined as:

$$F'_K = F'_t - F'_f \quad (\text{eq. 6})$$

The Reynolds number for the flow at the inlet will be used as a reference and can be written as:

$$Re_m = \frac{4\dot{m}_m}{\pi \mu D_m} \quad (\text{eq. 7})$$

Because the dynamic viscosity is considered to be constant, the Reynolds number for the flow in component i can then be expressed as:

$$Re_i = \frac{\dot{m}_i}{\dot{m}_m} \cdot \frac{D_m}{D_i} Re_m \quad (\text{eq. 8})$$

The inlet diameter of the manifold is 0.1023 m and the diameter of each riser is 0.051 m. This implies that the Reynolds number for the flow in each riser will be approximately 67% of the inlet Reynolds number if the flow splits equally between the risers. The results of the Flownex simulations for the flow split between the risers and pressure drop due to the primary frictional losses as a function of the inlet mass flow rate are given Table 10.

Table 10: Flow split and pressure drop for 1D simulations.

$\dot{m}_m$ (kg/s)	$\dot{m}_1$ (kg/s)	$\dot{m}_2$ (kg/s)	$\dot{m}_3$ (kg/s)	Δp (Pa)
0.37	0.1235	0.1233	0.1233	6.33
0.45	0.1502	0.1499	0.1499	10.08
0.52	0.1735	0.1732	0.1733	14.26
0.58	0.1935	0.1932	0.1933	18.60
0.67	0.2237	0.2231	0.2232	24.96

From Table 10 it can be seen that flow splits almost equally between the risers with the maximum difference between risers being 0.1%. It can also be seen that mass flow rate through the first riser is slightly larger than the mass flow rates through the second and third risers. The mass flow rates through the second and third risers are the same.

Table 11: Flow split and pressure drop for 3D simulations.

$\dot{m}_m$ (kg/s)	$\dot{m}_1$ (kg/s)	$\dot{m}_2$ (kg/s)	$\dot{m}_3$ (kg/s)	Δp (Pa)
0.37	0.1227	0.1231	0.1242	8.00
0.45	0.1490	0.1499	0.1511	12.00
0.52	0.1723	0.1732	0.1745	16.00
0.58	0.1927	0.1935	0.1938	19.00
0.67	0.2222	0.2232	0.2246	25.00

The results of the Fluent simulations for the flow split between the risers and pressure drop due to the total losses as a function of the inlet mass flow rate are given Table 11. From Table 11 it can also be seen that flow splits almost equally between the risers with the maximum difference between risers being 0.47%. It can also be seen that mass flow rate through the third riser is the largest, followed by the second riser and lastly the first riser. The maximum difference the mass flow rates obtained by Fluent and Flownex for the first riser is -0.78%, for the second riser 0.17% and for the third riser 0.76%. The pressure drop obtained by Fluent for the mass flow rate of 0.37 kg/s is comparable with the pressures drop shown in Table 3 given by Lisowski [14] for the test section. The assumed mass flow rate measured by Lisowski [14] is approximately 1.3% less than the corresponding mass flow rate used in the Fluent simulation.

Table 12: Loss factors for test section.

Re_m	F'_t	F'_f	F'_K
4470.9	7.880	6.236	1.644
5437.6	7.991	6.712	1.279
6283.5	7.979	7.111	0.868
7008.5	7.616	7.455	0.161
8096.0	7.510	7.497	0.013

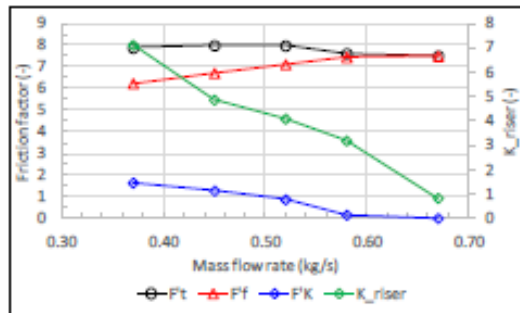


Fig. 5: Total, frictional and form loss factors as function of inlet Reynolds number.

The total, frictional and form loss factors as a function of the inlet Reynolds are summarized in Table 12. The loss factors are also shown graphically in Fig. 5. Also included in Fig. 5 are the lumped riser form factors given in Table 8.

It should be noted that only for the last Reynolds number the flow in the entire test section was in the turbulent regime. For all the other Reynolds numbers the flow in the risers was in the transitional regime. From Fig. 5 it can be seen that the primary friction loss factor increases with increasing mass flow rate as expected from the behaviour of the Darcy-Weisbach friction coefficients. The total loss factor increases with the increase in the mass flow rate from $Re_{in} = 4471$ to $Re_{in} = 6284$ and then decreases as the Reynolds number increases further. The form loss factor decreases as the Reynolds number increases. One would have expected the values of the of the form factor F_K not to be so small at the larger mass flow rates. This could possibly be attributed to the fact that the 3D model does not account for all the geometric details that could contribute to the form losses.

The trend for the form loss factor is in accordance with the trend for the lumped form factors K_{riser} observed in Tables 8, as well as Tables 7 and 9. The results shown in Fig. 5 therefore confirm the suspicion that the form factors for the lower and upper manifolds might be a function of the mass flow rate.

The 1D model and in particular the 3D model should be thoroughly revised and refined to ensure that all the relevant detail is properly accounted for in order that a comprehensive study can be performed to characterize the form factors for the test section.

IV. CONCLUSIONS

The paper first discussed the details of the experimental RCCS facility constructed by Lisowski [14]. The Flownex model that was constructed based the geometrical data was obtained from the

engineering drawing given by Lisowski [14] was described and shown in Fig. 2.

The losses comprising the primary frictional and the form losses were then evaluated for iso-thermal flow and compared with the corresponding experimental results given in Lisowski [14]. It can therefore be concluded that more information must be provided in order for the results given in Table 1 to be evaluated fully.

To characterize the system and to develop an understanding of the behaviour and performance of the system various simulations representing the steady-state natural circulation experimental runs were performed in which the specification of the form factors was varied. In all cases the primary losses were accounted for by Flownex based on the geometric details of the pipes and the relevant associated flow parameters. From the results obtained for the various cases it was deduced that the suggested strategy to model the experimental RCCS facility in the absence of more detailed information is the approach followed in case 5. The appropriate form factors should be specified for the relevant components and augmented with lumped form factors specified for TRO1, TRO2 and TRO3. It will also appear that the value of the lumped form factor is a function of the mass flow rate.

To investigate the behaviour of the form factors for the test section 1D Flownex and 3D Fluent models of the test section were constructed to determine the primary, total and form loss factors for iso-thermal flow conditions. The Flownex model was used to predict the primary frictional losses and the Fluent model was used to predict the total losses. The associated loss factors were then calculated and the form loss factors were obtained by subtracting the primary loss factors from the total loss factors. The result confirmed the suspicion that the form loss factors for the test section is a function of the mass flow rate.

It is recommended that a comprehensive detailed study be performed to characterize the experimental RCCS facility. This is important if it is intended to employ the facility as a basis or vehicle to study the 1D system CFD simulation of natural circulation flows for various conditions.

ACKNOWLEDGEMENT

This work is also based on research supported by the South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa (Grant No 61059). Any opinion, finding and conclusion or recommendation expressed in this material is that of

the authors and the NRF of South Africa do not accept any liability in this regard.

REFERENCES

- [1] L.J. Lommers, B.E. Mays, F. Shahrokhi, Passive heat removal impact on AERVA HTR design, *Nuclear Engineering and Design*, v271, p.569-577 (2014).
- [2] Y.Y. Bae, S.D. Hong, Y.W. Kim, Scaling analysis of reactor cavity cooling system for PMR200, Paper HTR2012-7-020, Proc. HTR 2012, Tokyo, Japan, 28 Oct. – 1 Nov. 2012.
- [3] L. Capone, Y. Hassan, R. Vaghetto, Reactor cavity cooling system (Rccs) experimental characterization, *Nuclear Engineering and Design*, v241, p.4775-4782 (2011).
- [4] M. Corradini, M. Anderson, M. Muci, Thermal-hydraulic analysis of an experimental reactor cavity cooling system with air, NUPE Final Report Part I: Experiments, NEUP 11-3079, University of Wisconsin-Madison (2014).
- [5] D.D. Lisowski, M.T. Farmer, Experimental observation of natural circulation flow in the NSTF at steady-state conditions, Paper HTR2014-6-143, Proc. HTR 2014, Weihai, China, 27 - 31 Oct. 2014.
- [6] J.H. Kim, Y.Y. Bae, C.S. Kim, S.D. Hong, M.H. Kim, The test results of the NACEF RCCS test facility, Trans. Korean Nucl. Soc. Spring Meeting, Jeju, Korea, 7 – 8 May 2015.
- [7] A. Frisani, Y. Hassan, Computation fluid dynamic analysis of the reactor cavity cooling system for very high temperature gas-cooled reactors, *Annals of Nuclear Energy*, v72, p.257-267 (2014).
- [8] H. Zhao, Y. Dong, Y. Zheng, T. Ma, X. Chen, Numerical simulation on heat transfer process in the reactor cavity of modular high temperature gas-cooled reactor, *Appl. Therm. Eng.*, v125, p.1015-1024 (2017).
- [9] M.H. Kim, N.I. Tak, H.S. Lim, Thermal-fluid assessment of the design options for reactor vessel cooling in a prismatic core VHTR, *Annals of Nuclear Engineering*, v37, p.1774-1782 (2010).
- [10] P.G. Rousseau, C.G. du Toit, J.S. Jun, J.M. Noh, Code-to-code comparison for analyzing the steady-state heat transfer and natural circulation in an air-cooled RCCS using GAMMA+ and Flownex, *Nuclear Engineering and Design*, v291, p.71-89 (2015).
- [11] P. Niemand, C.G. du Toit, N.I. Tak, M.H. Kim, Simulation of natural convective flow in an experimental reactor cavity cooling system facility, 10th South African Conf. Comp. Appl. Mech., Potchefstroom, South Africa, 3 – 5 Oct. 2016.
- [12] H. Zhao, Y. Zheng, X. Chen, T. Ma, Y. Dong, A one-dimensional code of the passive residual heat removal system for the modular high temperature gas-cooled reactor, *Progress in Nuclear Engineering*, v110, p.374-383 (2019).
- [13] R. Hu, D.D. Lisowski, M. Bucknor, A.R. Kraus, Q. Lv, Modeling the ambient condition effects of an air-cooled natural circulation system, Paper ICONE25-67418, Proc. ICONE25, Shanghai, China, 6 – 7 Jul. 2017.
- [14] D.D. Lisowski, Thermal hydraulic analysis of an experimental reactor cavity cooling system with water: Performance and stability, Doctoral dissertation, University of Wisconsin-Madison (2013).
- [15] D.D. Lisowski, T.C. Haskin, A. Tokuhito, M.H. Anderson, M.L. Corradini, Study on the behavior of an asymmetrically heated reactor cavity cooling system with water in single phase, *Nuclear Technology*, v183, p.75 – 87 (2013).
- [16] M-Tech Industrial, Flownex SE Version 8.10.0.3800 <http://www.flownex.com/> (2019).
- [17] V.L. Streeter, E.B. Wylie, *Fluid Mechanics*, 7th Edition, McGraw-Hill (1979).
- [18] ANSYS, ANSYS documentation: Meshing User's Guide, (2017).