

Modelling Heat Exchangers with Embedded Phase Change Materials for Aircraft

Julie Yvonne Frank

University of Leeds, School of Computing
Centre for Doctoral Training in Fluid Dynamics

Submitted in accordance with the requirements for the degree of
Doctor of Philosophy

June 8, 2025

Intellectual Property

The candidate confirms that the work is their own, except where work which has formed part of jointly-authored publications has been included. The contribution of the candidate and the other authors work has been explicitly indicated herein. The candidate confirms that appropriate credit has been given within the thesis where reference has been made to the work of others.

Chapter 4:

The validation results on the shell and tube heat exchanger geometry were presented at the international conference on heat transfer and fluid flow (HTTF23) in August 2023.

Chapter 5:

The comparison of the different fluid side fin geometries were presented at the 14th IIR Conference on Phase-Change Materials and Slurries for Refrigeration and Air Conditioning (PCM 2024) conference. This resulted in a peer-reviewed article published within the proceedings of the conference:

Effect of different fin geometries on the thermal and hydraulic performance of PCM-Compact Heat Exchanger. Frank J. Y., Borman D. J., Greiciunas E., Khan A., Summers J. 14th IIR Conference on Phase-Change Materials and Slurries for Refrigeration and Air Conditioning. Proceedings: Paris France, May 29-31, 2024. (Available from: <http://dx.doi.org/10.18462/iir.pcm.2024.0007>)

Chapter 6:

The results on the PCM-Compact Heat Exchanger geometry were presented at the UK Fluids Conference 2024 in Swansea, UK.

Chapter 7:

The model development of the effectiveness-NTU model was presented at the Sum-

mer Heat Transfer Conference 2023 and this resulted in a peer-reviewed article:

Phase Change Materials for Absorbing Peak Heat Loads in Aircraft: An Effectiveness-NTU Model for Predicting the Performance of Compact Heat Exchanger. Frank J. Y., Borman D. J., Greiciunas E., Khan A., Summers J. Proceedings of the ASME 2023 Heat Transfer Summer Conference collocated with the ASME 2023 17th International Conference on Energy Sustainability. ASME 2023 Heat Transfer Summer Conference. Washington, DC, USA. July 10-12, 2023. V001T06A002. ASME. (Available from: <https://doi.org/10.1115/HT2023-106867>)

The results on the application and validation of the model without fins were presented at HiSST 2024. A peer-reviewed article was written for the conference, but the proceedings have not been made available to the public yet.

Effectiveness-NTU Model for PCM-Compact Heat Exchanger Performance Prediction, Frank J. Y., Borman D. J., Greiciunas E., Khan A., Summers J.

I was responsible for the research work undertaken in this thesis and publications while Duncan Borman, Amirul Khan, Evaldas Greiciunas and Jon Summers are my thesis supervisors and contributed in editing of the manuscripts. This project was conducted in partnership with BAE Systems who provided the example aerospace loading profile for a PCM heat exchanger.

This copy has been supplied on the understanding that it is copyright material and that no quotation from the thesis may be published without proper acknowledgement.

The right of Julie Yvonne Frank to be identified as Author of this work has been asserted by Julie Yvonne Frank in accordance with the Copyright, Designs and Patents Act 1988.

Acknowledgments

I would like to thank my academic supervisors Dr Duncan Borman, Dr Amirul Khan, Dr Evaldas Greiciunas and Dr Jon Summers who guided me and provided me with their valuable support throughout this project. I am also thankful to the CDT for Fluid Dynamics at the University of Leeds for funding this research work (Grant number EP/S022732/1) and providing me with the tools for completing a Masters and a PhD. Many thanks to BAE Systems, that supplied key information on PCM heat exchangers for aircraft and for funding a part of my supervision team.

This thesis is dedicated to my parents and my friends for their support and patience in the last three years, and to all the people from the CDT office, who have had to bear with my antics on a daily basis. In particular I would like to extend my gratitude to Yatin, my honorary supervisor, who took so much of his time to patiently explain basic concepts to me. I would like to mention Danielle for her outstanding support in providing me with long tea breaks and consistently bringing in milk. To Elisa and Dounia, who set a very high standard on friendship and brighten my day when I get to spend it with them. Finally, thanks to Leo for encouraging me on a daily basis and reminding me that there is some enjoyment in completing a long project.

Abstract

The growing thermal load in aircraft design poses challenges for the development of thermal management systems (TMS) for future generation aircraft. Current systems are often oversized to manage highly transient thermal loads encountered during flight. Phase change material (PCM)-based heat exchangers offer a promising alternative by mitigating temperature fluctuations, which could lead to an overall system size reduction. PCM are materials that absorb and release thermal energy during phase transitions, typically between solid and liquid states, at nearly constant temperatures. However, existing PCM heat exchanger designs, primarily developed for building applications, require significant geometric modifications to meet the aerospace industry's stringent requirements for high thermal loads and rapid melting cycles. In this work, the integration of PCM in compact heat exchanger is studied on the basis of a plate-fin heat exchanger.

To establish a foundational understanding of PCM melting behaviour and develop an accurate modelling framework, a study of a shell-and-tube heat exchanger from the literature was recreated. Initial two dimensional simulations demonstrated overpredictions in PCM melting due to neglected temperature variations within the fluid channel. This was addressed by incorporating a conjugate heat transfer model, which coupled the PCM and fluid domains. An additional three-dimensional model further highlighted the complex dynamics of axial and lateral convection plumes, emphasizing the need for detailed modelling in future designs. These validations provided critical insights into PCM melting and informed the development of subsequent models.

The melting behaviour of PCM within plate-fin heat exchangers, which are commonly used in thermal management systems, was investigated by using numerical simulations across various extended surface configurations and operating conditions. The incorporation of fins on both the fluid and PCM sides demonstrated significant improvements in the thermal performance, leading to enhanced heat transfer and

increased PCM melting efficiency, when compared to the simple rectangular geometries that are commonly found in the literature.

An effectiveness-NTU model was developed based on these findings, providing reliable performance estimates at a fraction of the computational cost of full Computational Fluid Dynamics (CFD) simulations. The model results were compared to the CFD simulations under aerospace-relevant conditions, achieving good agreement for rectangular cavities and capturing the melting dynamics of finned PCM layers. Its flexibility allows for rapid adjustments to design parameters, making it a practical tool for sizing and optimizing compact heat exchangers. This work advances the understanding and modelling of PCM-based heat exchangers, offering a foundation for efficient, lightweight thermal management solutions tailored to high-performance aircraft applications.

Contents

1	Introduction	1
1.1	Thermal management in aerospace	1
1.1.1	Thermal management systems	1
1.1.2	Heat exchangers	4
1.2	Phase change materials	6
1.2.1	Thermal energy storage	6
1.2.2	Use in heat exchanger	8
1.3	Project outline	9
1.3.1	Project focus	9
1.3.2	Aims and objectives	11
1.3.3	Thesis structure	12
2	Design of Traditional Heat Exchangers	14
2.1	Introduction	14
2.2	Traditional heat exchangers for aerospace	15
2.2.1	Heat exchanger classification	15
2.2.2	Compact heat exchanger geometries	15
2.3	Predicting the performance of compact heat exchanger	19
2.3.1	Analytical models	19
2.3.2	Experimental methods	22
2.3.3	Computational Fluid Dynamics	24
2.3.4	Heat exchanger network modelling	29
2.4	Applications of CFD modelling of compact heat exchanger	29
2.4.1	Heat exchanger modelling over different scales	30
2.4.2	Overcoming heat exchanger design challenges with CFD	33
2.5	Summary	36
3	Review of Phase Change Materials for Energy Storage	38

3.1	Introduction	39
3.2	Phase change materials	40
3.2.1	Selection of PCM	40
3.2.2	Heat transfer and phase change phenomenon	43
3.2.3	The mushy zone	44
3.2.4	Constrained and unconstrained melting	46
3.3	PCM heat exchanger geometries	47
3.3.1	Applications of PCM materials	47
3.3.2	Effect of geometries	50
3.3.3	Effect of operating conditions	52
3.3.4	Performance improvement of LHTES Units	53
3.4	Predicting the performance of PCM heat exchanger	60
3.4.1	Analytical models	61
3.4.2	CFD models	63
3.4.3	Enthalpy-porosity model	70
3.5	Summary	74
4	Modelling of a PCM Shell and Tube Heat Exchanger	76
4.1	Introduction	77
4.2	Problem Description	77
4.3	Solution Independence Study on 2D Geometry	79
4.3.1	Problem Setup	79
4.3.2	Mesh Independence and Time Step Independence Study	81
4.3.3	Description of melting process	84
4.3.4	Effect of discretization	85
4.3.5	Effect of material properties definition	86
4.3.6	Effect of software packages	87
4.3.7	Comparison with literature results	89
4.4	3D Conjugate Heat Transfer Simulation	91
4.4.1	Problem Setup	91
4.4.2	Mesh Independence	92
4.4.3	Comparison with literature results	92
4.4.4	Effect of temperature drop over length of heat exchanger	94
4.5	Detailed 3D PCM section analysis	96
4.5.1	Problem Setup	96
4.5.2	Mesh Independence Study	96
4.5.3	Comparison with 2D Simulation	97

4.5.4	Melting Regimes	98
4.5.5	Movement of the phase change front	103
4.6	Summary	106
5	Improving the melting of PCM with finned surfaces in fluid layer	108
5.1	Introduction	108
5.2	Aerospace demand profile	109
5.3	Melting in between two heated plates	110
5.3.1	Onset of convection	110
5.3.2	Melting in a plate heat exchanger	115
5.4	PCM in compact heat exchanger - plate fin geometry	122
5.4.1	Validation of compact heat exchanger modeling	122
5.4.2	Melting of PCM in offset strip fin compact heat exchanger . .	131
5.4.3	Melting of PCM in continuous fins compact heat exchanger . .	141
5.5	Effect of different fluid channel geometries	146
5.5.1	Hydrothermal performance of finned and unfinned surface at initial condition	147
5.5.2	Energy Storage and Effectiveness during melting	149
5.6	Summary	152
6	Enhancement of melting by using extended surfaces in PCM	153
6.1	Introduction	154
6.2	Melting in a 2D rectangular cavity	154
6.2.1	Problem setup	154
6.2.2	Mesh and time Independence	156
6.2.3	Effect of cavity size on melting	157
6.2.4	Effect of varying the dimensions and operating conditions on the melting of PCM	159
6.2.5	Onset of convection	164
6.3	Melting in a finned PCM compact heat exchangers	166
6.3.1	Problem setup	166
6.3.2	Mesh and Time Independence Study	168
6.3.3	Description of the melting process	172
6.4	Comparison with heat exchanger with plain PCM layer	174
6.4.1	Comparison of the melting over time	174
6.4.2	Overall heat transfer performance	175
6.5	Summary	175

7	Development of a low cost model for PCM compact heat exchanger	178
7.1	Introduction	179
7.2	Effectiveness-NTU model with phase change based on a linear Stefan Condition	179
7.2.1	Development of the model	179
7.2.2	Validation of effectiveness-NTU model	189
7.3	Alternative approach based on energy conservation	193
7.3.1	Development of effectiveness-NTU model based on energy conservation	194
7.3.2	Comparison with linear Stefan effectiveness-NTU model . . .	197
7.4	Incorporating fins into the effectiveness-NTU model	197
7.4.1	Treatment of finned PCM cavity	198
7.4.2	Addition of fins to the full Effectiveness-NTU model	212
7.5	Using the effectiveness-NTU model as a design tool	216
7.5.1	Motivation	216
7.5.2	Determining the dimensions of PCM compact heat exchanger geometries	216
7.5.3	Determining optimal fin spacing for uniform melting of PCM over length of heat exchanger	220
7.6	Summary	226
8	Conclusion	227
8.1	Summary of results	227
8.2	Implication of results	231
8.3	Recommendations for future work	231
	References	233

List of Figures

1.1	Schematic of an air Environmental Control System (ECS) Passenger Air Conditioner (PACK) for a Boeing 737-800 aircraft based on flow direction and temperature. (Adapted from Jennions et al. (2020)) . . .	2
1.2	Tubular heat exchanger geometries: (a) Shell and tube heat exchanger; (b) Tube-fin heat exchanger	5
1.3	Platular heat exchanger geometries: (a) Plate heat exchanger; (b) Plate-fin heat exchanger.	5
1.4	Types of thermal energy storage.	7
1.5	Phase change diagram in terms of temperature and stored heat for (a) Isothermal phase change; (b) Non-isothermal phase change for a typical solid/liquid PCM.	8
1.6	Example geometries for PCM heat exchangers for HVAC applications taken from the literature	9
1.7	Integration of PCM heat exchanger in thermal management system to reduce peak load.	10
2.1	Plate heat exchanger geometry	16
2.2	Plate fin heat exchanger layer structure.	17
2.3	Example of fin geometries used within Tube-fin heat exchangers . . .	18
2.4	Examples of common Printed circuit heat exchanger channel geometries.	19
2.5	Test rig used for steady-state steam-to-air experiments. (Adapted from Shah (2003))	23
2.6	Process of developing a CFD model.	24
2.7	Summary of the Finite Volume Method	28
2.8	2D Computational domain for modelling transient response in wavy channels in Zheng et al. (2014)	31
2.9	Computational domain used by Menéndez Pérez et al. (2022) to study the effect of vortex generators for louvered tube-fin heat exchangers .	31

2.10	Modelling of additively manufactured inter-layer heat exchanger by Greiciunas (2019). The left hand side shows the geometry used for modelling, while right hand side shows the corresponding temperature contours within the cold layer for (a) inter-layer core; (b) Pin-fin core; (c) simplified porous core.	32
2.11	Use of optimization methods for improving the thermo-hydraulic performance of heat transfer surfaces: Topology optimization results of winglets for tube-fin heat exchanger compared to traditional designs taken from Kobayashi et al. (2019)	35
2.12	Heat exchanger designs facilitated by additive manufacturing: Use of TPMS surfaces (Taken from Al-Ketan et al. (2020))	36
3.1	Flow chart showing different stages involved in the design and development of a latent heat storage (Adapted from Sharma et al. (2009)).	39
3.2	Classification and Types of PCM materials	40
3.3	Heat Transfer Phenomena occurring during melting of PCM based on Jany & Bejan (1988)	44
3.4	Different Types of Phase Change: (a) Distinct, (b) Continuous, (c) Crystalline.	45
3.5	Microstructure during (a) solidification and (b) melting of paraffin wax as observed by Yang et al. (2019)	46
3.6	Flow description of constrained and unconstrained melting in a cylinder heated from the outside.	47
3.7	Applications of PCM. Graphic is based on the following reviews: Oró et al. (2012), Diaconu et al. (2022). Mehling et al. (2022), Nazir et al. (2019), Xu et al. (2020), Xie et al. (2018), AL-Saadi & Zhai (2013), Yuan et al. (2021), Price (2003), Jaguemont et al. (2018)	49
3.8	Different configurations of LHTES units	50
3.9	Classification of heat transfer enhancement techniques for LHTES . .	55
3.10	Popular types of fin geometries used in PCM-Shell and Tube Heat Exchanger	56
3.11	Popular types of fin geometries used in plate heat exchangers	56
3.12	Use of encapsulated PCM within a packed bed heat exchanger	58
3.13	Increasing the effective thermal conductivity by using porous medium based on examples from the literature.	59
3.14	Improving the melting by changing the contact surface angle based on examples from the literature.	60

3.15	Solid-Liquid phase change process schematic	61
3.16	Summary of fixed grid phase change models.	64
3.17	Apparent Heat Capacity formulation $c_a(T)$ (red curve) for a piece-wise relaxation of the enthalpy relation $H(T)$ (grey curve).	66
3.18	Comparison of experimental and numerical result for the melting front over time for a square cavity. Reproduced from Faden et al. (2018)	71
3.19	Impact of porous medium on the local velocity	73
4.1	Geometry of the problem studied in Hosseini et al. (2012) (not to scale, dimensions given in mm).	77
4.2	Experimental and numerical results for shell and tube heat exchanger study from Hosseini et al. (2014) for $T_{in} = 343.15 K$	78
4.3	Experimental data for Dynamic Viscosity and Thermal Conductivity extracted from Hosseini et al. (2014)	79
4.4	Geometry, boundary condition and example mesh (Mesh 1) used in 2D setup.	80
4.5	Mesh independence: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).	82
4.6	Time independence: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).	83
4.7	Description of the melting process in 2D domain over time: (a) Temperature Contours, (b) Streamlines overlaid on the different phases of PCM.	84
4.8	Discretization: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).	86
4.9	Effect of constant and variable material properties - Viscosity (μ) and Thermal Conductivity (k): (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).	87
4.10	Comparison with OpenFOAM: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).	88
4.11	Comparison of current numerical obtained from Ansys FLUENT, and results from literature (2D numerical by Seddegh et al. (2016) , experimental and numerical by Hosseini et al. (2012)): (a) Average Temperature ($\bar{T}$); (b) Liquid Fraction ($\bar{\phi}$) and (c) Position of the melt front at 30 min intervals.	90
4.12	Geometry and Boundary Condition of the full 3D domain.	91

4.13	Mesh independence (3D long): Average temperature and average liquid fraction over time	92
4.14	Comparison of current 3D simulations and experimental and numerical results from Hosseini et al. (2014).	93
4.15	Variation in the average liquid fraction and average temperature over the length of the heat exchanger at different time steps.	94
4.16	Isocontours of $\phi \approx 0.5$ taken at intervals of $t = 30$ min showing the position of the melting front at different locations along the length of the heat exchanger.	95
4.17	Geometry and Boundary Condition for Shell only model.	96
4.18	Mesh Independence: Position of Melt Front at on XY plane at $Z = 50$ mm (left) and on symmetry plane (right)	97
4.19	Comparison of 2D and 3D simulation: (a) Position of melt front; (b) Average Temperature ($\bar{T}$); (c) Average liquid fraction ($\bar{\phi}$).	98
4.20	Conductive Regime: Contours of temperature with in plane streamlines in the X normal direction. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	99
4.21	Onset of convection: Contours of temperature with in plane streamlines in the X normal direction. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	100
4.22	Coarsening Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	101
4.23	Early convection in cylinder: Contours of temperature with in plane streamlines in the Z normal direction ($z=50$ mm). This corresponds to a location in the cylinder where the peak in the solid-liquid interface is initially off-centred. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	102
4.24	Mixed Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	103

4.25	Lateral Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.	104
4.26	Shape of the liquid PCM volume obtained from the iso-volume of temperature in range $T = 324.15 - 343.15$ K coloured based on normalized axial velocity.	105
5.1	Demand profile for an aircraft PCM heat exchanger as provided by project partner, where $Pr = \frac{\mu c_p}{k}$	109
5.2	Diagram of the problem considered for measuring the onset of convection. Note: not to scale.	111
5.3	Mesh information for independence study for onset of convection study in a 2D bounded cavity	112
5.4	Mesh Independence Study: Comparison of the location of the solid liquid interface over time ($t = 100 - 500$ s) based on the melting thickness (a), obtained by creating an iso-surface of $\phi = 0.5$	112
5.5	Effect of buoyancy on melting based on two different modelling assumptions: with buoyancy and no buoyancy.	114
5.6	Effect of buoyancy on average properties.	115
5.7	Geometry and Boundary conditions used for the plate heat exchanger. Note: not to scale.	115
5.8	Mesh Independence Study for plate heat exchanger: Average properties over time.	117
5.9	Mesh Independence: melting thickness (a) over the length of the heat exchanger (z) over time.	117
5.10	Melting thickness (a) and normalized melting thickness ($a/a_{\max}$) over the length of the heat exchanger (z) over time at $Re=200$	119
5.11	Temperature drop (ΔT) between the inlet and outlet of the water channel over time t at $Re=200$	119
5.12	Effect of Reynolds number on bulk properties: (a) Temperature drop (ΔT) in fluid channel, (b) Average liquid fraction (ϕ) over time of PCM.	120
5.13	Position of the melt front (a) over the length of the heat exchanger (z) over time (t) for different Reynolds number values.	121
5.14	Total fraction of melted PCM ($\bar{\phi}$) at the end of the melting cycle (t_{end}).	122

5.15	Thermal boundary layer thickness (δ_t) for different values of Reynolds value for $t=0$ s, normalized by the plate spacing.	123
5.16	Strip Fin geometry and flow domain for a single channel with 10 periodic flow lengths used for CFD analysis.	124
5.17	Picture of final mesh (Medium).	125
5.18	Comparison of Temperature (T) and Velocity Magnitude ($\mathbf{u}$) along central line.	126
5.19	Boundary layer resolution in the wake of a fin in the middle of the domain	126
5.20	Comparison of j and f factor obtained using CFD with experimental results from Kays & London (1964)	128
5.21	Contours of Velocity Magnitude (left) and Temperature (right) along the length of the water channel at $Re=200$ for laminar and $k-\omega SST$ models.	130
5.22	Effect of turbulence modelling on the j and f factor prediction.	131
5.23	Velocity magnitude ($\mathbf{u}$) at different locations along the length of the strip-fin compact heat exchanger	132
5.24	Contours of normalized velocity magnitude for different values of the Reynolds number for the strip-fin geometry. White line shows the average value of the velocity magnitude in the domain	132
5.25	Model of the PCM-Compact Heat Exchanger that will be used	133
5.26	Medium mesh used for PCM-Compact Heat Exchanger simulations.	134
5.27	Initial flow and temperature profile in water channel: contours of velocity magnitude (left) and temperature (right) at $Re = 200$ for the strip fin geometry.	135
5.28	Mesh Independence Study : Variation of averaged properties over time for the strip-fin compact heat exchanger.	136
5.29	Time Independence Study: Variation of averaged properties over time for the strip-fin compact heat exchanger.	137
5.30	Mesh Independence Study: Variation in the melting thickness (a) over the length (z) of the strip-fin compact heat exchanger.	137
5.31	Change in temperature (T) along the length (z) and depth (x) of the strip-fin compact heat exchanger at the interface between PCM and metal plate. White shadow indicates the location of the fins underneath the PCM layer.	138

5.32	Change in the thickness of melted PCM along the length (z) and depth (x) of the strip-fin compact heat exchanger at $t = 100$ s. This was obtained by creating an isosurface of $\phi = 0.5$ and by extracting the position of this isosurface. White shadow indicates the location of the fins underneath the PCM layer.	139
5.33	Variation in the melting thickness (a) and normalized melting thickness (a/a_{max}) over the length of strip-fin compact the heat exchanger (z) over time.	139
5.34	Variation of average properties over time for different values of the Reynolds value ($Re = [50, 100, 150, 200]$).	140
5.35	Variation in the melting thickness (a) over the length of the strip-fin compact heat exchanger (z) for different values of the Reynolds number ($Re = [50, 100, 150, 200]$) and time.	141
5.36	Model of the PCM-Compact Heat Exchanger with a continuous fin geometry.	142
5.37	Initial flow and temperature profile in water channel: contours of velocity Magnitude (left) and Temperature (right) at $Re = 200$ for the continuous fin geometry.	143
5.38	Variation in the melting thickness (a) and normalized melting thickness (a/a_{max}) over the length of the continuous fin compact heat exchanger (z) over time for continuous fin at $Re = 200$	144
5.39	Variation along the depth (x) and length (z) of the continuous fin compact heat exchanger at $t = 100$ s. The white shaded area indicates the location of the fin in the fluid directly underneath the PCM.	145
5.40	Effect of Reynolds number on bulk properties over time for a continuous fin geometry.	146
5.41	Variation in the melting thickness (a) over the length of the continuous compact heat exchanger (z) for different values of the Reynolds number ($Re = [50, 100, 150, 200]$) and time.	147
5.42	Geometry and Dimensions of the compared cases: (a) No fins, (b) Continuous Fins, (c) Strip Fins.	148
5.43	Hydrothermal performance of different heat transfer surfaces at the initial condition.	149
5.44	Effect of different finned surfaces on bulk properties at $\dot{m} = 2.21 \times 10^{-3}$ kg/s over time.	149

5.45	Variation in the melting thickness (a) over the length of the heat exchanger (z) over time for different fluid side fin geometries time at $\dot{m} = 2.21 \times 10^{-3} \text{ kg/s}$	150
5.46	Overall thermal performance comparison of different finned surfaces over range of mass flow rates after $t = 100\text{s}$	151
6.1	(a) Geometry of heat exchanger with finned PCM layer. Red square represents the area approximated by the simplified cavity problem. (b) Simplification of finned heat exchanger problem, where the hot fluid is represented as a constant temperature (T_H) boundary condition. Half the cavity is modelled by applying a symmetry boundary condition to reduce computational cost. Finally the fins are modelled as solid lines.	154
6.2	Mesh Independence study for constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature ($\bar{T}$). The study was conducted with a maximum time step size of $\Delta t_{\max} = 0.1 \text{ s}$	156
6.3	Time Independence study for constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature ($\bar{T}$). The study was conducted by using the medium mesh.	157
6.4	Effect of size [mm] on melting in constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) heat stored in PCM (E).	158
6.5	Streamlines of melted PCM coloured by normalized velocity magnitude ($ \mathbf{u} /\max[\mathbf{u}]$) and Temperature (T) for $5 \times 5 \text{ mm}$ cavity at $t=100 \text{ s}$. The position of the melt front is shown by a black line.	158
6.6	Dimensions of cavity used for different Aspect Ratio (AR)	159
6.7	Effect of Aspect Ratio on melting in constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) heat stored (E) in PCM.	160
6.8	Position of the melt front ($\phi = 0.5$) over time for increasing aspect ratio.	160
6.9	Comparison of position of melt front ($\phi = 0.5$) for (a) $AR = 0.5$ and (b) $AR = 2$	161
6.10	Effect of wall temperature on melting in constrained cavity at $AR=2$: (a) Average liquid fraction ($\bar{\phi}$), (b) Heat stored in PCM (E), (c) Position of the melt front at $t = 100 \text{ s}$	162
6.11	Effect of varying the aspect ratio (AR) and wall temperature (T_w) on the liquid fraction (ϕ) at the end of the melt cycle. Crosses (x) represent the data points used to generate the contour plot.	162

6.12	Effect of varying the surface area and wall temperature (T_w) on the liquid fraction (ϕ) at the end of the melting cycle. Crosses (x) represent the data points used to generate the contour plot.	163
6.13	Comparison of conduction only (- - -) and with convection (—) modelling assumption, for $AR = 1$ and $AR = 2$ at $T_w = 301.15K$	164
6.14	Comparison of conduction only (- - -) and with convection (—) modelling assumption, for different wall temperature $T = 297.15 - 301.15$ K at $AR=2$	165
6.15	Model of the PCM-Compact Heat Exchanger with finned PCM layer.	167
6.16	Picture of the medium mesh	168
6.17	Mesh independence Study: average properties over time.	169
6.18	Mesh Independence Study: position of the melting front over.	169
6.19	Initial time step independence study with constant time step size.	170
6.20	Time step independence study where the time step is updated every N th timestep	170
6.21	Change in the position of the melting front over the depth x of the heat exchanger.	171
6.22	Change in melting over the length of the heat exchanger shown by: (a) Average liquid fraction ($\bar{\phi}$) over time, (b) Position of the melt front over the length of the heat exchanger.	172
6.23	Comparison of average properties over time for different Reynolds number: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature drop (ΔT).	173
6.24	Comparison of the position of the melt front over the length of the heat exchanger for different Reynolds number at $t = 100$ s at (a) Section 1 and (b) Section 8.	174
6.25	Geometries used for comparison.	175
6.26	Comparison of bulk properties over time for $Re = 200$: (a) Average liquid fraction ($\bar{\phi}$); (b) Effectiveness (ϵ).	176
6.27	Comparison of the average heat transfer performance of finned and plain PCM-CHX over the melting cycle based on: (a) Average Effectiveness ($\bar{\epsilon}$); (b) Stored Heat in PCM (E_{PCM}).	176
7.1	Diagram of the heat transfer problem in a PCM heat exchanger.	180
7.2	Visualisation of the effect of the modelling assumption on the temperature profile compared to the original model	183

7.3	Comparison of the melting thickness a between a linear approximation of the Stefan condition and a 1D CFD model	185
7.4	Discretization of the length of the heat exchanger into elements. . . .	185
7.5	Flowchart outlining the different steps of the convergence algorithm within the elemental loop.	186
7.6	Flowchart presenting the full PCM- ϵ -NTU model.	188
7.7	Strip Fin heat exchanger geometry with 1/8-13.95 fins in fluid layer. (Dimensions are in mm).	189
7.8	Correlations for j and f factor for 1/8-13.95 fins obtained from previous CFD results in Chapter 5.	190
7.9	Mesh and Time independence study of PCM- ϵ -NTU model for Strip Fin geometry	190
7.10	Comparison of $\epsilon - NTU$ and CFD model results over time at $Re = 200$: (a) Average Liquid Fraction, (b) Temperature Drop, (c) Local melting thickness.	191
7.11	Comparison of ϵ -NTU model (—) results with CFD results (---) over range of Reynolds number for Strip Fin Geometry.	192
7.12	Schematic of the energy conservation model.	194
7.13	Flowchart of the new PCM ϵ -NTU model based on Energy Conservation	196
7.14	Comparison of Energy conservation model with Linear Stefan ϵ -NTU model for offset strip fin geometry at $Re=200$	197
7.15	Diagram of a finned PCM layer showing a detailed view of an element of the fin.	199
7.16	Choice of length scales for different fin geometries: (a) Idealized plain rectangular fin, and (b) Plain triangular fin. (Figure adapted from Shah (2003)).	202
7.17	Shape of the melt front for CFD results for two common fin geometries at $AR = 1$, $T_w = 301.15K$ and $A_c = 25 \text{ mm}^2$	202
7.18	Geometry considered for testing the surface effectiveness model. . . .	204
7.19	Flowchart of analytical model of fins in PCM	205
7.20	Comparison of analytical and CFD results for the rectangular fin cavity for varying aspect ratio at constant wall temperature $T_w = 301.15K$ and constant free flow area $A_c = 25 \text{ mm}^2$: (a) Liquid fraction over time; (b) position of melt front.	207

7.21	Comparison of analytical and CFD results for the rectangular fin cavity for varying aspect ratio for varying temperature at the end of the melting cycle ($t = 100s$) at constant free flow area $A_c = 25 \text{ mm}^2$: (a) Liquid Fraction prediction of analytical model; (b) Liquid Fraction prediction of CFD model; (c) Percentage Difference between CFD and analytical results.	208
7.22	Comparison of analytical and CFD results for the rectangular fin cavity for varying surface area for varying temperature at the end of the melting cycle ($t = 100s$): (a) Liquid Fraction prediction of analytical model; (b) Liquid Fraction prediction of CFD model; (c) Percentage Difference between CFD and analytical results.	209
7.23	Comparison of analytical and CFD results for the triangular fin cavity for aspect ratio $AR = 1$ and $AR = 1.75$ at $A_c = 25 \text{ mm}^2$ and $T_w = 301.15K$: average liquid fraction over time.	211
7.24	Comparison of analytical and CFD results for the triangular fin cavity for varying aspect ratio AR at $A_c = 25 \text{ mm}^2$ and $T_w = 301.15K$: Position of the melt front over time	211
7.25	Flowchart of $\epsilon - NTU$ model incorporating fins in fluid and PCM layer.	213
7.26	Geometry of finned PCM compact heat exchanger used for validating the $\epsilon - NTU$ model.	214
7.27	Comparison of the $\epsilon - NTU$ model with the CFD results for a compact heat exchanger with fins in the PCM layer for $Re = 50$ and $Re = 200$: (a) Liquid Fraction $\bar{\phi}$, (b) Temperature Drop ΔT	214
7.28	Comparison of the drop in the liquid fraction in the cavities over the length of the heat exchanger for the $\epsilon - NTU$ model (—) and the CFD results (- - -).	215
7.29	Effect of varying the dimensions of the heat exchanger on (a) the required length (l_z); and (b) Heat Exchanger Volume (V); to store 500 kJ of heat in 100 s in the PCM based on an operating Reynolds number of $Re=200$	217
7.30	Effect of varying the dimensions of the heat exchanger on the Heat Exchanger Volume (V) for varying Aspect Ratio (AR).	218
7.31	Effect of varying the dimensions of the heat exchanger on the length (l_z) of the heat exchanger for varying Aspect Ratio (AR).	219

7.32	Flowchart of algorithm for finding optimal dimensions of heat exchanger based on an optimal melting criteria ϕ_{opt} . The optimal dimensions are determined by using a bisection algorithm as shown further down in Figure 7.33. To calculate the liquid fraction and heat flux, the energy conservation $\epsilon - NTU$ model described in Figure 7.25 is used.	221
7.33	Algorithm used to find optimal length z_{opt} of fin cavity such that at the end of the melt cycle, the full cavity is melted.	222
7.34	Comparison of different initial aspect ratio on average properties over time: (a) Liquid Fraction, (b) Temperature Drop in fluid layer.	223
7.35	Effect of Aspect ratio on the dimensions of the heat exchanger with varying fin spacing: (a) Change of optimal fin spacing over the length of the heat exchanger; (b) Variation in the dimensions of the heat exchanger.	224
7.36	Comparison of the effect of using variable and constant fin spacing on the average liquid fraction over time for similar heat exchanger dimensions (l_x, l_y, l_z) at $AR = 4$	225
7.37	Variation of the compactness (β) over the normalized length of the heat exchanger $(l_z/l_{z,max})$ for both constant and variable fin spacing.	225

List of Tables

2.1	$\epsilon - NTU$ relationship for common flow arrangements, where $C_r = C_{\min}/C_{\max}$	22
3.1	Selection criteria for phase change materials (Adapted from Sharma et al. (2009)) and Nazir et al. (2019))	42
3.2	Selection of PCM based on material type (Adapted from Sharma et al. (2009) and Nazir et al. (2019))	42
4.1	Simplified thermophysical material properties (based on Hosseini et al. (2012) and as implemented in Ansys FLUENT)	78
4.2	Size of different Meshes	81
4.3	Summary of considered schemes	85
4.4	Computational resources used for the 2nd order accurate simulations of a 2D cylinder	89
5.1	Material Properties of RT18HC according to Rubitherm	110
5.2	Summary of numerical schemes used for the melting of PCM in a cavity. WB: With Buoyancy; NB: No Buoyancy.	111
5.3	Material Properties used in compact heat exchanger simulations	116
5.4	Mesh Independence of the initial solution	117
5.5	Fin properties of 1/8-13.95 offset strip fin geometry from Kays & London (1964)	123
5.6	bulk properties measured as part of the mesh independence study: Temperature and Pressure Drop.	127
5.7	Mass flow rate and equivalent Reynolds number for offset strip fin (OSF), continuous fin (CF) and No Fin (NF) case.	147
6.1	Material Properties used in 2D cavity simulations	155
6.2	Mesh sizes for mesh independence study of 2D cavities.	156
6.3	Dimensions of cavity for constant plate spacing	163

6.4	Material Properties used in compact heat exchanger simulations . . .	168
6.5	Mesh Independence Study at initial condition	169
7.1	Summary of geometric parameters for the determination of the shape factor for different cavity geometries. The red shape for each geometry represents the simplified geometry of the melting front used within the model.	206
7.2	Simulation condition for example case 1 to illustrate how to find PCM heat exchanger dimensions	217
7.3	Additional simulation condition for example case 2 to illustrate the optimum variable fin spacing algorithm	223

Nomenclature

Abbreviation

1D	One dimensional
2D	Two dimensional
3D	Three dimensional
ACM	Air Cycle Machine
CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulations
ECS	Environmental Control System
EE	Electric/electronic systems
FE	Finite Element
FV	Finite Volume
HPWS	High Pressure Water Separator
LES	Large Eddy Simulations
LG-PCM	Liquid/gas PCM
LHTES	Latent Heat Thermal Energy Storage
LMTD	Logarithmic Mean Temperature Difference
MEA	More Electric Aircraft
NTU	Number of transfer units
PCM	Phase Change Material
PHX	Primary Heat Exchanger
PV	PACK Valve
RANS	Reynolds Average Numerical Simulations
SHX	Secondary Heat Exchanger
SL-PCM	Solid/liquid PCM

SS-PCM Solid/solid PCM

TMS Thermal Management System

TPMS Triply Periodic Minimal Surface

English Letter Symbols

$\dot{m}$ Mass flow rate [kg/s]

A Heat transfer surface area [m²]

a Melting thickness (distance from the heated wall) [m]

A_c Cross sectional area [m²]

A_{mush} Mushy zone constant [kg/(m³s)]

AR Aspect ratio: $AR = \frac{x}{y}$

b Plate spacing [m]

C Heat capacity rate [W/K]

c_p Specific heat capacity [J/(kgK)]

d Dendritic length [m]

d_h Hydraulic diameter [m]

E Heat [J]

f Fanning friction factor: ratio between the local shear stress and the local flow kinetic energy density

g Gravity [m/s²]

H Total enthalpy [J/kg]

h Specific enthalpy [J/kg]

h heat transfer coefficient [W/(m²K)]

j Colburn factor: $j = \frac{Nu}{RePr^{1/3}}$

k Thermal conductivity [W/(mK)]

L Latent heat of phase change [J/kg]

l Fin length [m]

L_f Fin length [m]

m Fin effectiveness parameter

m Mass [kg]

n Number of layers

Nu Nusselt number: $Nu = \frac{hL}{k}$

P	Perimeter [m]
P	Pressure [Pa]
Q	Heat Load [W]
q	heat transfer rate [W]
R	Thermal resistance [K/W]
Re	Reynolds number: $Re = \frac{\rho u L}{\mu}$
S	Shape factor
S_{ij}	Stress mean tensor
St	Stefan Number: $St = \frac{c_p(T-T_m)}{L}$
T	Temperature [K]
t	time [s]
t_w	Plate thickness [m]
U	Overall heat transfer coefficient [W/(m ² K)]
u	Velocity [m/s]
V	Heat exchanger volume [m ³]
Pr	Prandtl Number: $Pr = \frac{\mu c_p}{k}$

Greek Symbols

α	Thermal diffusivity [m ² /s]
α_{eff}	Effective thermal diffusivity [m ² /s]
β	Surface density [m ²]/[m ³]
β	Thermal expansion coefficient
Δ	Difference
δ	Fin thickness [m]
δ_{ij}	Kronecker delta
ϵ	Effectiveness
η_f	Fin efficiency
η_0	Extended surface effectiveness
μ	Kinematic viscosity [Pa.s]
μ_t	Turbulent viscosity [Pa.s]
$\nu_{\Gamma_{ls}}$	Propagation velocity of the interface

ϕ	Liquid fraction
ρ	Density [kg/m ³]
θ	Angle [rad]

Subscripts

0	Initial
B	Bulk
c	Cold
f	Fin
f	Final
f	Fluid
h	Hot
i	Inner
in	Inlet
l	Liquid
m	Melting
o	Outer
out	Outlet
p	Plate
PC	Phase change
PCM	Phase change material
s	Solid
w	Wall
max	Maximum
min	Minimum

CHAPTER 1

Introduction

Contents

1.1 Thermal management in aerospace	1
1.1.1 Thermal management systems	1
1.1.2 Heat exchangers	4
1.2 Phase change materials	6
1.2.1 Thermal energy storage	6
1.2.2 Use in heat exchanger	8
1.3 Project outline	9
1.3.1 Project focus	9
1.3.2 Aims and objectives	11
1.3.3 Thesis structure	12

1.1. Thermal management in aerospace

1.1.1. Thermal management systems

Heat is generated within an aircraft by both occupants and onboard systems, while exterior factors such as solar heating or aerodynamic heating due to skin friction at supersonic speeds can further increase internal temperatures ([Ahlers \(2020\)](#)). In accordance with data reported in [van Heerden et al. \(2022\)](#), around 60% of heat generated within an advanced fighter aircraft is produced by the engine and 20% by the Environmental Control System (ECS). The remaining part of the heat is produced by the electrical and hydraulic systems of the aircraft. This heat must be dissipated

to maintain a comfortable environment for the occupants and enables the systems to operate at an acceptable temperature limit, preserving component life and performance. For this purpose, thermal management systems (TMS) are designed to address the diverse operating conditions of a flight, with a focus on minimizing weight, enhancing heat transfer efficiency, and reducing drag and fuel consumption (Asli et al. (2024)). This leads to a complex design optimization problem, where multiple systems have to be incorporated.

Figure 1.1 shows an environmental control system (ECS) PACK for a Boeing 737-800, which can be used as part of the TMS. An air conditioner pack is an air cycle refrigeration system comprised of a combined compressor and turbine machine, valves for temperature and flow control, and heat exchangers. In this system, bleed air is supplied at high temperature and pressure where it is split into two streams by using a PACK valve (PV). One part is supplied within the primary heat exchanger (PHX), while the remaining part is mixed back to the airflow downstream of the turbine to control its temperature. The PHX is used to cool the bleed air by using ram air inlet before the air is compressed within the Air Cycle Machine (ACM) compressor. This air is cooled for a second time in the secondary heat exchanger (SHX), before being expanded in the ACM turbine. As the air expansion occurs in

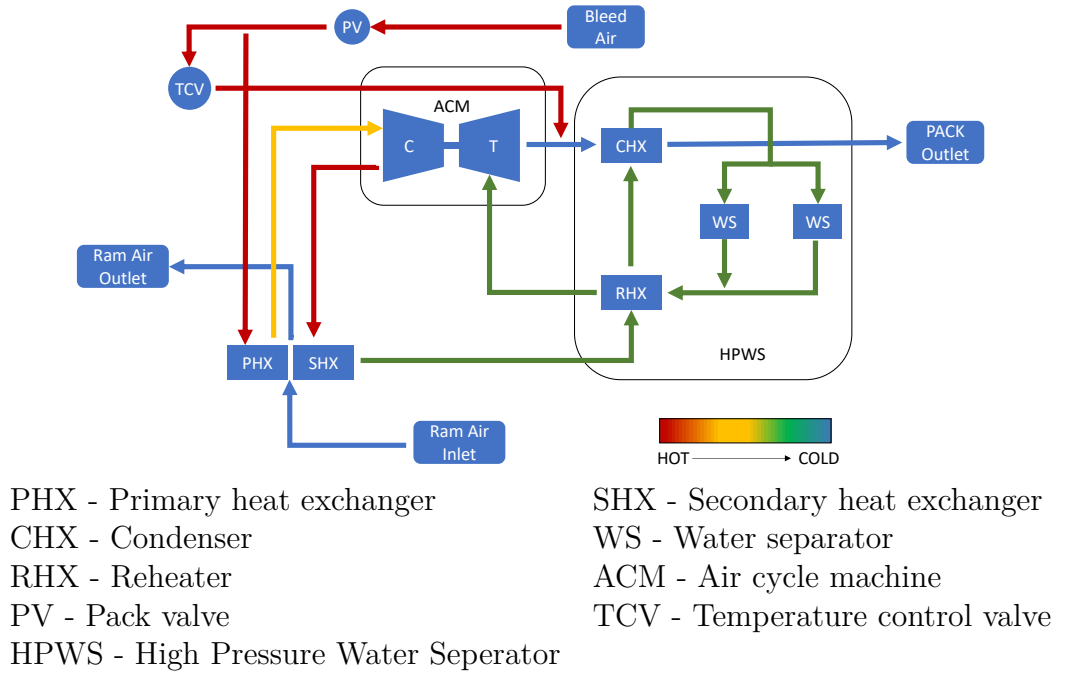


Figure 1.1: Schematic of an air Environmental Control System (ECS) Passenger Air Conditioner (PACK) for a Boeing 737-800 aircraft based on flow direction and temperature. (Adapted from Jennions et al. (2020))

the turbine, the temperature of the air reduces, leading to potential condensation. To avoid liquid water from reaching the cabin or avionics, a high pressure water separator (HPWS) comprised of a condensor and a reheater is used to extract liquid water before the air enters the turbine.

Fuel stored inside of the wing can be used as an alternative heat sink to ram air due to its large thermal capacitance, where it is used to store intermittent heat generated by hydraulic systems over the course of the flight. However, as the fuel is consumed, the cooling capacity diminishes, requiring careful management to ensure sufficient system cooling and prevent fuel degradation. Additionally, energy can be recovered by routing the engine fuel supply line to absorb residual engine heat before combustion, maximizing fuel efficiency and cooling.

Military aircraft share similarities with commercial aircraft, such as comparable ground conditions and the need for an environmental control system (ECS). On top of this, military aircraft rely more heavily on electronics, often packed with sensitive components like sensors and radar systems that are highly vulnerable to temperature fluctuations. Weapon systems further complicate thermal management by generating intense heat spikes during use, adding to the aircraft's thermal load. Additionally, ram-air cooling is restricted in some military designs, as it compromises stealth capabilities. Consequently, more heat must be managed by the fuel, which not only cools hydraulic systems, but also serves as a critical cooling reservoir for electronic and weapon systems (see [van Heerden et al. \(2022\)](#)).

[Ahlers \(2020\)](#) identifies multiple significant changes to aircraft systems in recent years that create challenges for the TMS. Composite materials have increasingly been adopted due to their potential to address many of the challenges with traditional metallic materials. However, composites have low thermal conductivity, which hinders the dissipation of heat through the aircraft skin into the atmosphere. The growing use of electrical/electronic (EE) systems has significantly increased heat loads, from passenger in-flight entertainment systems in commercial aircraft to sophisticated avionics for navigation, communication, and control. Military aircraft face additional challenges in managing the heat generated by high-power weapon and defence systems while maintaining stealth capabilities. Additionally, the shift towards More Electric Aircraft (MEA) architectures has driven substantial increases in thermal loads. To deal with this, Boeing introduced a no-bleed air system on the B787, replacing traditional pneumatic systems with electric alternatives, such as electric compressors for cabin pressurization and air conditioning, electric resistance

blankets for wing de-icing, and engine-mounted generators for startup power. This approach reduces engine bleed-air extraction, which improves fuel efficiency, reduces weight, and simplifies system architecture, thereby enhancing reliability and lowering maintenance costs (Sinnott (2007)).

Reducing CO₂ emissions to meet net-zero targets will drive substantial changes in civil aviation in the coming years, with an emphasis on alternative fuels to kerosene, including hybrid and fully electrified aircraft (see Asli et al. (2024)), as well as hydrogen-powered options (see Srinath et al. (2022)). Military aviation is expected to see significant advancements in avionics and integrated weapon systems (see van Heerden et al. (2022)). These developments in both civil and military aviation are likely to place considerable demands on already strained thermal management systems (TMS). Consequently, thermal management has become a critical design challenge for future aircraft. Among proposed solutions, incorporating phase change materials (PCM) in heat exchangers has been identified as a viable approach to reduce the load on TMS by evening out transient heat loads by van Heerden et al. (2022).

1.1.2. Heat exchangers

1.1.2.1 Heat exchanger geometries

Heat exchangers are a core component within thermal management systems, responsible for transferring heat away from the source. Among the various types of heat exchanger geometries, the shell and tube configuration (see Figure 1.2.a) is the most common, accounting for 90% of industrial applications (Awais & Bhuiyan (2018)). However, this geometry is difficult to manufacture with a sufficient surface density needed for aerospace applications (Hesselgreaves et al. (2017)). A more compact alternative, the tube-fin heat exchanger (see Figure 1.2.b) has been developed, increasing surface density by adding finned surfaces around the tubes to enhance heat dissipation efficiency.

Plate heat exchangers (see Figure 1.3.a) are built from multiple thin plates stacked together in a frame and sealed with gaskets. Fluid circulates through small channels formed by the corrugated surfaces of these plates, significantly enhancing the thermal efficiency of the heat exchanger. A more compact version is the plate-fin heat exchanger shown in Figure 1.3.b, which incorporates plates and fins of various geometries within the fluid channels to further increase the surface area for heat

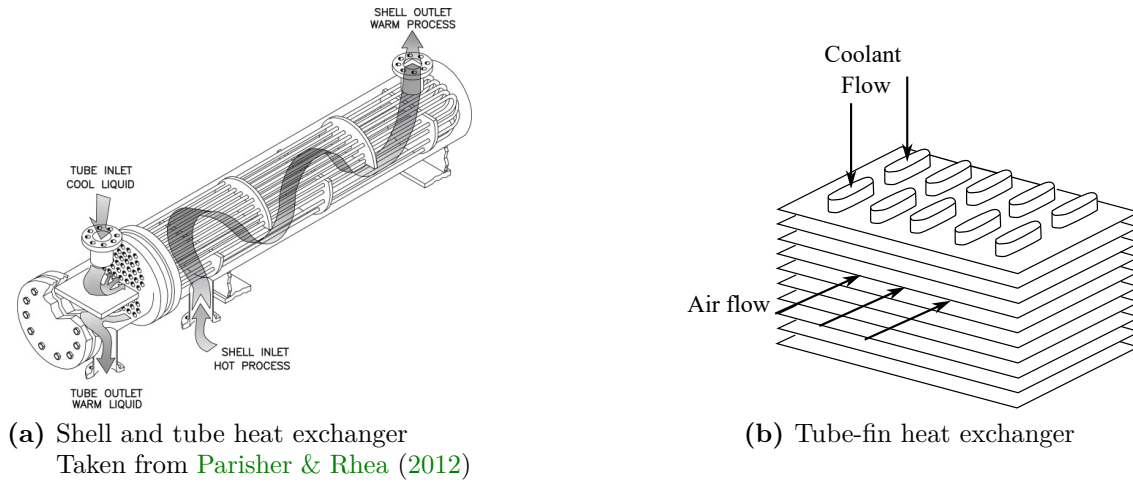


Figure 1.2: Tubular heat exchanger geometries: (a) Shell and tube heat exchanger; (b) Tube-fin heat exchanger.
Reprinted from Pipe Drafting and Design, Third Edition, Roy. A. Parisher, Robert A. Rhea, Chapter 6 Mechanical Equipment, p.117, Copyright (2025), with permission from Elsevier.

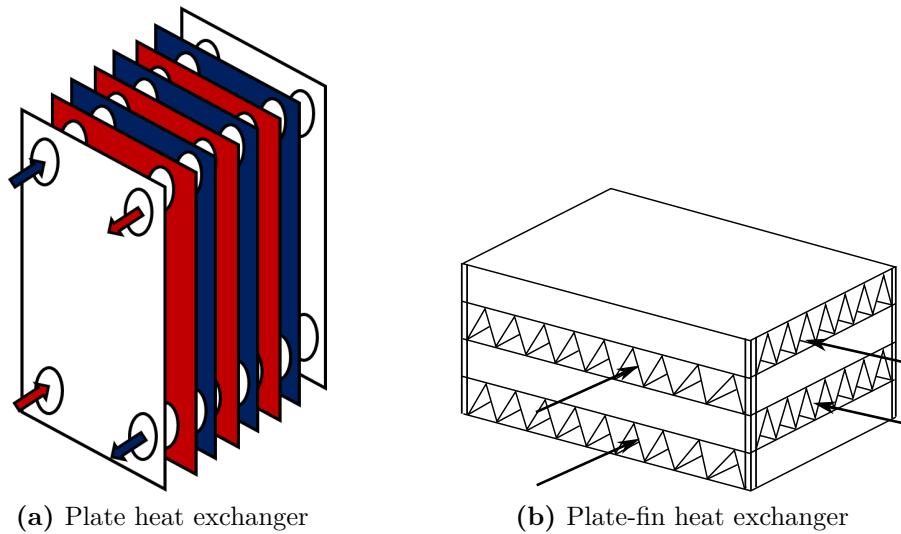


Figure 1.3: Platular heat exchanger geometries: (a) Plate heat exchanger; (b) Plate-fin heat exchanger.

transfer. This geometry is commonly used within aircraft due to its reliability, compactness and effectiveness in operating with low temperature differences between the fluid stream ([Hesselgreaves et al. \(2017\)](#)).

1.1.2.2 Performance Prediction

A solid understanding of the heat exchanger response, operating under various conditions within the thermal management system, is essential to design an op-

timized system. This requires extensive modelling and experimental work to get a close prediction of the heat exchanger performance. Analytical models such as the effectiveness-NTU ($\epsilon - NTU$) and the Logarithmic Mean Temperature Difference (LMTD) models can provide valuable insights on choosing appropriate sizes for the heat exchanger or obtaining the pressure and temperature drop for existing designs. [Kays & London \(1964\)](#) provides a comprehensive overview of each method, alongside an experimental data set for the thermo-hydraulic performance of various corrugation geometries that can be used as an input for the analytical models.

While analytical models provide useful insights into flow and heat transfer, they often fall short in capturing complex phenomena such as flow maldistribution. Computational Fluid Dynamics (CFD) is increasingly used in industry, offering a cost-effective and time-efficient alternative for modelling various scenarios. CFD enables detailed visualization of flow and temperature fields around complex geometries, which is more challenging to achieve through experimental methods ([Hesselgreaves et al. \(2017\)](#)). [Bhutta et al. \(2012\)](#) summarizes recent CFD studies in the literature, highlighting its application in predicting the performance of header designs, corrugations and heat transfer surface enhancements.

1.2. Phase change materials

1.2.1. Thermal energy storage

Thermal energy storage can be subdivided into sensible, latent and thermo-chemical energy storage, as shown in Figure 1.4. Sensible heat energy storage occurs when a material is heated such that its temperature increases. The amount of sensible heat that is stored in the material is:

$$E_{\text{sensible}} = \int_{T_0}^{T_f} mc_p dT, \quad (1.1)$$

where m is the mass, c_p is the sensible specific heat, T_0 and T_f are the initial and final temperatures of the material respectively. Common materials include molten salts, water, thermal oils, metals and rocks.

Latent heat storage involves the absorption or release of energy when a material undergoes a phase change. In the case of a pure material, this process results in an isothermal phase change, as illustrated in Figure 1.5.a. The quantity of latent heat that can be stored in a material is determined by its mass m and the latent heat of

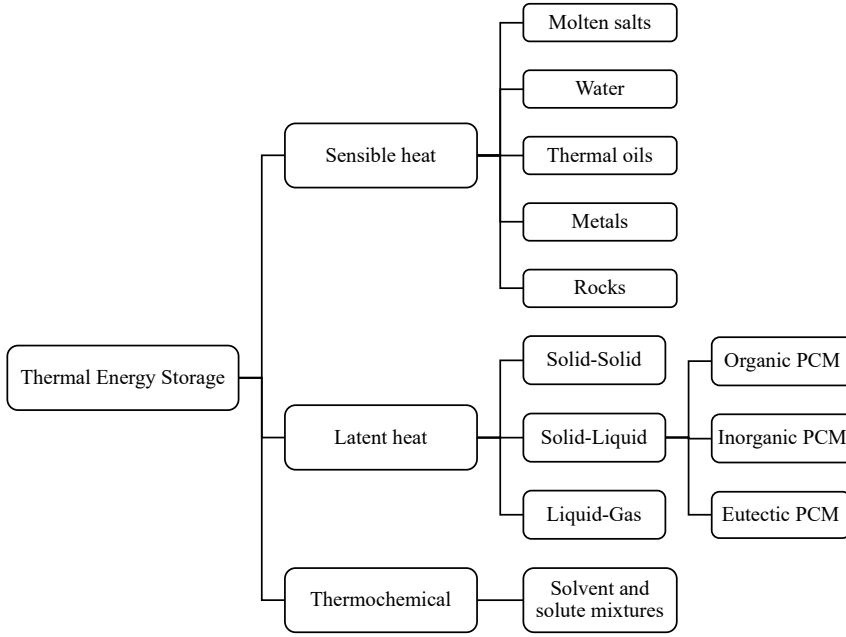


Figure 1.4: Types of thermal energy storage.

phase change L :

$$E_{\text{latent}} = mL. \quad (1.2)$$

Materials with a high latent heat of phase change are known as phase change materials (PCM), which can undergo transitions between solid/solid (SS), solid/liquid (SL), or liquid/gas (LG) states. SL-PCM typically have a lower latent heat than LG-PCM, but experience a smaller volume change, simplifying the design for storage. As such they have received a significant amount of attention in the literature.

It is important to note that PCM are rarely pure substances, such that the phase change occurs over a small range of temperature referred as the solidus T_s and liquidus T_l temperatures, as shown in Figure 1.5.b. This region is often referred to as the mushy region, where part of the material is solid, while the remainder is liquid. For these materials, the amount of stored heat is described by the following relationship:

$$E_{\text{stored}} = \underbrace{\int_{T_0}^{T_s} mc_p dT}_{\text{Solid}} + \underbrace{\int_{T_s}^{T_l} mc_p dT}_{\text{Mushy}} + mL + \underbrace{\int_{T_l}^{T_f} mc_p dT}_{\text{Liquid}} \quad (1.3)$$

Heat can also be stored in thermochemical storage that rely on exothermic and endothermic reactions, making them suitable for long-term storage of heat. In his

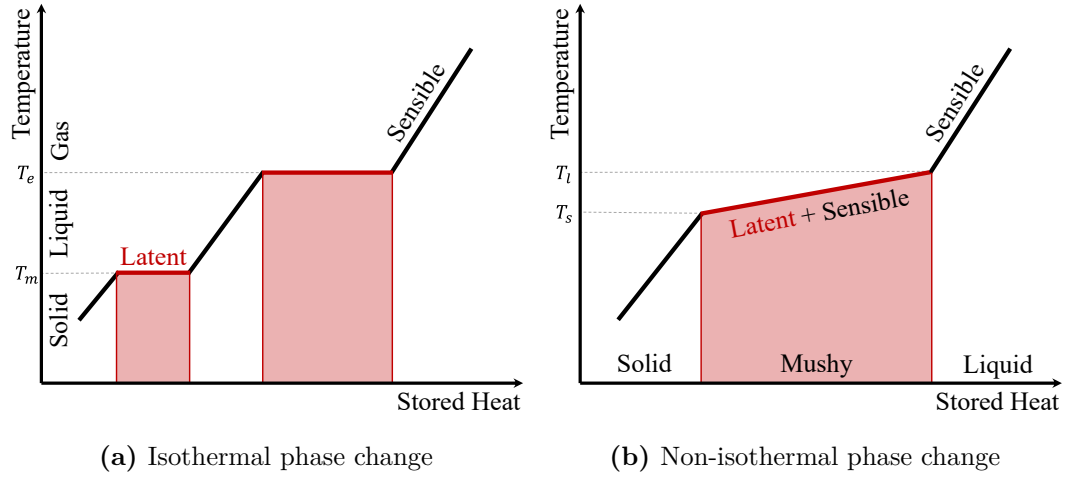
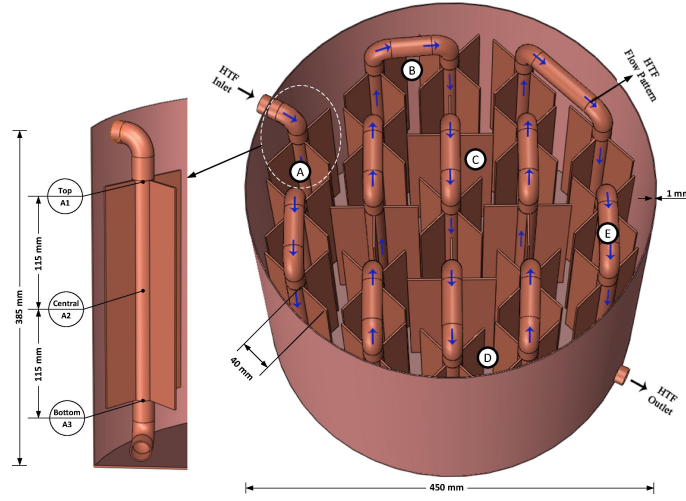


Figure 1.5: Phase change diagram in terms of temperature and stored heat for (a) Isothermal phase change; (b) Non-isothermal phase change for a typical solid/liquid PCM.

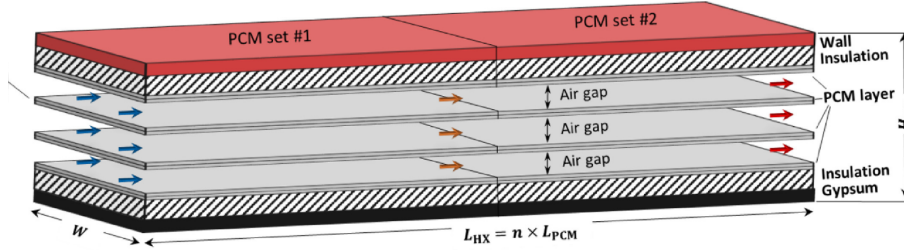
review of recent developments in phase change materials for energy storage, [Nazir et al. \(2019\)](#) compares the various methods of energy storage and concludes that latent heat storage is the most promising option. This is due to the large storage density when compared to sensible heat, its commercial viability due to low material costs and longer lifespan time compared to thermochemical storage.

1.2.2. Use in heat exchanger

PCM benefit from a low volume-to-energy ratio and have low thermal conductivity, resulting in large scales over which the phase change occurs. This has facilitated the design of small and cost-effective domestic space heating or cooling units that operate over the cycle of a day, where the PCM is typically encapsulated in simple geometries to limit pumping power costs. Commonly referenced designs in the literature include conventional heat exchanger geometries such as shell-and-tube heat exchangers (See Figure 1.6.a) and plate heat exchangers (See Figure 1.6.b), where the PCM is encapsulated in one of the fluid sides. To enhance the overall thermal conductivity of the PCM, multiple studies have explored the incorporation of filler materials with beneficial properties. These fillers can include nanoparticles, extended surfaces, or foams, often composed of metals or graphene.



(a) Shell and Tube system
(Taken from [Khan & Khan \(2017\)](#))



(b) Plate system
(Taken from [Morovat et al. \(2019\)](#))

Figure 1.6: Example geometries for PCM heat exchangers for HVAC applications taken from the literature.

Reprinted from Applied Energy, Volume 206, Zakir Khan, Zulfiqar A. Khan, Experimental investigations of charging/melting cycles of paraffin in a novel shell and tube with longitudinal fins based heat storage design solution for domestic and industrial applications, p.1161 Copyright (2025), with permission from Elsevier.

Reprinted from Energy and Buildings, Volume 199, Navid Morovat, Andreas K. Athienitis, Jose A. Candanedo, Vasken Dermardiros, Simulation and performance analysis of an active PCM-heat exchanger intended for building operation optimization, p.50 Copyright (2025), with permission from Elsevier.

1.3. Project outline

1.3.1. Project focus

Current thermal management systems are characterised by highly transient heat load profiles, where the aircraft can generate high power densities over a short period of

time. This requires the main heat exchanger to be engineered based on the maximum heat load to avoid damage to aircraft components and ensure passenger comfort.

PCM have been found to be a potential solution to this issue based on a review by [van Heerden et al. \(2022\)](#). As visualised in Figure 1.7.a, the PCM could be used to absorb heat from the working fluid when the temperature is above the melting temperature of the PCM, leading to the melting of the PCM. As the thermal load on the aircraft decreases after a certain amount of time, the PCM solidifies and releases heat back into the working fluid. To implement the benefits of PCM, an additional PCM heat exchanger could be integrated upstream of the main heat exchanger, as illustrated by Figure 1.7.b. This integration allows the downstream heat exchanger to be designed around an averaged load, resulting in a lower peak load. This could lead to a reduction in the size of the main heat exchanger.

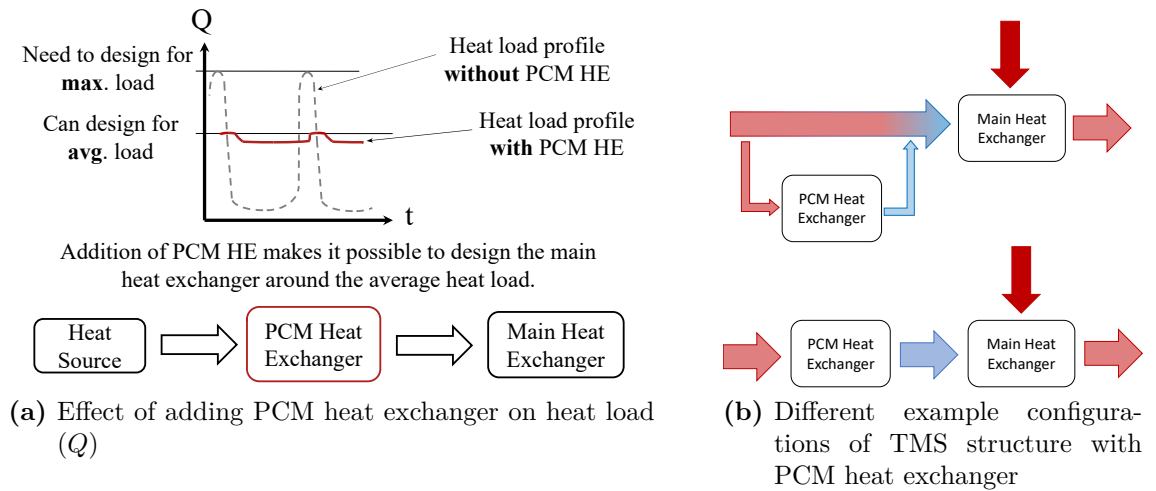


Figure 1.7: Integration of PCM heat exchanger in thermal management system to reduce peak load.

A majority of the heat exchangers incorporating PCM available in the literature are working under operating conditions with a melting and solidification range of multiple hours. This leads to geometries with low compactness levels, such as shell and tube heat exchangers and plate heat exchangers. In aerospace a significant reduction in the loading cycle length is required, such that higher power density needs to be achieved by increasing the heat transfer rate between the heat transfer fluid and the PCM channels. Melting uniformity over the length of PCM heat exchangers is poorly understood, but this problem can have a significant impact on the heat absorption of the heat exchanger. This project will be addressing these key points by embedding and melting a paraffin PCM within plate-fin heat exchanger

geometries, which are typically used in aircraft. The effect of extended surface geometries in both fluid and PCM channels will be studied computationally over a range of operating conditions based on a loading profile for a PCM heat exchanger for aircraft. The knowledge gained from these studies will then be used to develop a semi-analytical model to help in rapidly sizing future PCM heat exchangers.

1.3.2. Aims and objectives

To be able to study this problem, the following aims and objectives have been defined:

Aim 1: Generate a modelling framework that accurately evaluates heat exchangers that incorporate PCM technology in existing latent heat thermal energy storage (LHTES) units:

Obj 1: Identify the challenges in modelling the phase change process that arise due to the thermo-physical properties of PCM and select an appropriate phase change model (Chapter 3).

Obj 2: Create a CFD model for the constrained melting of PCM within a heat exchanger and validate the model against experimental results (Chapter 4).

Aim 2: Develop the theoretical understanding of melting of PCM in compact heat exchangers:

Obj 1: Summarise the current state of research of LHTES and heat exchangers for aerospace application (Chapter 2-3).

Obj 2: Identify improvements in the fluid channel geometry and the operating conditions of the heat exchanger to obtain a more uniform melting of the PCM and increase the heat transfer rate between the PCM and the fluid (Chapter 5).

Obj 3: Improve the heat transfer rate between the PCM and the fluid by incorporating fins into the PCM channel and study the effect of flow rate on the melting uniformity of PCM (Chapter 6).

Aim 3: Develop a low computational cost model aimed at the early design stage for exploring the design space of PCM compact heat exchanger (Chapter 7):

- Obj 1:** Create an Effectiveness-NTU model for predicting the thermal performance of PCM compact heat exchanger.
- Obj 2:** Validate the model by comparing the melting rate and the temperature drop of the fluid side to CFD results for a PCM compact heat exchanger geometry.
- Obj 3:** Illustrate how the model could be used as a design tool by proposing example cases for sizing heat exchanger.

1.3.3. Thesis structure

Chapter 2 and 3 provide an overview of the literature on the design of traditional heat exchanger for aerospace applications and phase change materials for energy storage in particular. This outlines the different modelling techniques that will be used as part of the report. In particular, the finite volume method for describing fluid flow and heat transfer, appropriate boundary conditions to couple various fluid and solid domains, turbulence and phase change models are presented. On top of this, an overview of the $\epsilon - NTU$ is given.

In chapter 4, a shell and tube heat exchanger geometry from the literature will be studied. This will be used to validate the CFD modelling framework, which is based on a three-dimensional conjugate heat transfer model. The results are analysed to discuss the effect of temperature drop on the melting over the length of the heat exchanger. Finally, a more refined model of a smaller section of the the shell is created to analyse the complex behaviour of the fluid PCM plumes in the shell.

Chapter 5 uses the CFD modelling framework to study the effect of different fluid side enhancement techniques based on extended surfaces on a platular heat exchanger geometry subjected to an example loading profile for a military aircraft. The amount of stored heat and the uniformity of melting over a range of mass flow rates is compared for a case without fins, with continuous fins, and with offset strip fins.

Chapter 6 studies the effect of adding fins to the PCM side on a heat exchanger geometry with offset strip fins on the fluid side. For this, the effect of plate and fin spacing is initially studied on a two dimensional geometry of the PCM and fins. The findings are then incorporated into a three dimensional model to study the effect of Reynolds number on the melting uniformity over the length. Results are subsequently compared with results from Chapter 5.

An $\epsilon - NTU$ model for PCM heat exchangers with finned and unfinned PCM layer is developed in Chapter 7. Similar geometries and loading conditions are studied to the ones in Chapter 5 and 6 to validate the results of the low-cost model against high-fidelity CFD simulations. Finally, two example cases are provided that highlight how the model can be used to rapidly size and compare different heat exchanger geometries.

A summary of key results, as well as a recommendation for future work are provided in Chapter 8.

Design of Traditional Heat Exchangers

Contents

2.1	Introduction	14
2.2	Traditional heat exchangers for aerospace	15
2.2.1	Heat exchanger classification	15
2.2.2	Compact heat exchanger geometries	15
2.3	Predicting the performance of compact heat exchanger	19
2.3.1	Analytical models	19
2.3.2	Experimental methods	22
2.3.3	Computational Fluid Dynamics	24
2.3.4	Heat exchanger network modelling	29
2.4	Applications of CFD modelling of compact heat exchanger	29
2.4.1	Heat exchanger modelling over different scales	30
2.4.2	Overcoming heat exchanger design challenges with CFD	33
2.5	Summary	36

2.1. Introduction

In this chapter a comprehensive overview on the use and design of heat exchangers for aircraft is given. Section 2.2 describes the various compact heat exchanger geometries used in aerospace heat exchanger. Section 2.3 describes the various meth-

ods used to design heat exchangers. Finally, Section 2.4 provides an extensive review of the literature of the use of CFD to model compact heat exchangers.

2.2. Traditional heat exchangers for aerospace

2.2.1. Heat exchanger classification

Shah (2003) differentiates multiple ways of classifying heat exchangers, including transfer process, surface compactness, geometry and flow arrangements. In aerospace engineering, a particular focus has been put on the compactness of heat exchangers. By maximizing the heat transfer efficiency, the weight and volume of the heat exchanger can be reduced. Compactness is described by the surface density (β) as the ratio of heat transfer surface (A_i) to the volume of the heat exchanger (V_i):

$$\beta = \frac{A_i}{V_i}, \quad (2.1)$$

and the hydraulic diameter (d_h) of the heat exchanger core:

$$d_h = \frac{4A_c}{P}, \quad (2.2)$$

where A_c is the cross sectional area and P is the perimeter of the flow passage.

For gas-to-fluid heat exchangers, exchangers with a surface density greater than $700 \text{ m}^2/\text{m}^3$ for the gas stream and $400 \text{ m}^2/\text{m}^3$ for the liquid or phase change stream are typically classified as compact heat exchangers. Where $\beta \geq 3000 \text{ m}^2/\text{m}^3$, the fluid flow in the heat exchanger is consistently in the laminar regime such that these are referred to as laminar flow heat exchangers, while the term micro heat exchanger is used for surfaces where $\beta \geq 10,000 \text{ m}^2/\text{m}^3$.

2.2.2. Compact heat exchanger geometries

Another popular way of classifying heat exchangers is based on construction features, with common construction types including tubular, plate-type, extended surface and regenerative heat exchangers. In this section, common compact heat exchanger geometries will be introduced.

2.2.2.1 Plate heat exchanger

Gasketed plate heat exchangers are mainly used for liquid-liquid heat transfer applications. The plate heat exchangers are constructed by compressing thin corrugated plates together into a frame and sealing the edges with gaskets as shown in Figure 2.1. The corrugations provide mechanical support to the plate and result in narrow, highly interrupted flow passages, leading to high heat transfer coefficients and increased shear rates. A wide variety of corrugation patterns are available that can be selected based on design requirements. Due to the restrictive size of the flow passages, clean fluids are required to prevent blockages.

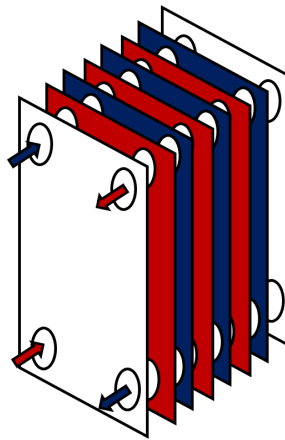


Figure 2.1: Plate heat exchanger geometry

The available materials for gaskets restrict the choice of fluids and the operating conditions of gasketed plate heat exchangers. This can be overcome by using welded plate heat exchangers, where the different plates are welded to each other.

2.2.2.2 Plate-fin heat exchanger

Plate-fin heat exchangers take advantage of extended surfaces in the form of corrugated fins or spacers to improve the heat transfer between two consecutive layers of the heat exchanger separated by a plate (see Figure 2.2.a). The fins lead to a greater surface density and lower hydraulic diameter and help in maintaining the structural integrity of the heat exchanger. A large amount of different fin designs as shown in Figure 2.2.b, can be used in plate fin heat exchangers to fit a wide range of operating conditions and are used in various applications, including aerospace.

The use of fins can lead to a reduction in the heat transfer coefficient depending on the fin spacing. An increase in the heat transfer coefficient of up to four times to its plain counterpart can be obtained by using geometries that interrupt the flow

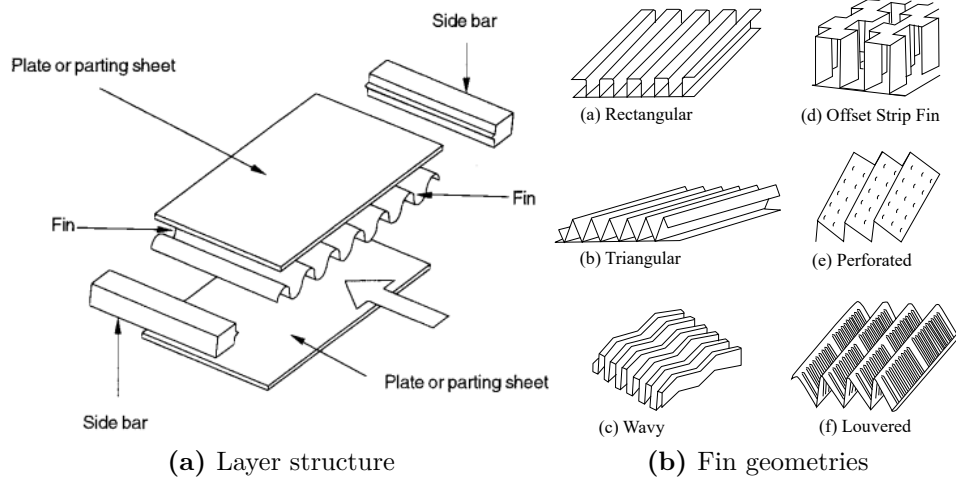


Figure 2.2: Plate fin heat exchanger layer structure.

such as wavy fins, offset strip fins or louvered fins (Shah (2003)). This is achieved by interrupting the thermal boundary layer and increasing the mixing of the flow through secondary flows and swirling (Awais & Bhuiyan (2018)).

Extensive research on fin geometries has been conducted both experimentally and numerically. Kays & London (1964) provides a large database of experimental thermal and hydraulic performance results for a variety of fin geometries. Furthermore, correlations have been built to describe the performance of the heat exchanger as reviewed in Sheik Ismail et al. (2010) for offset strip and wavy fins, and in Qasem & Zubair (2018) for plain, pin, louvered, offset strip and wavy channels.

2.2.2.3 Tube-fin heat exchanger

Tube-fin heat exchangers are typically built for gas-to-liquid heat exchange. The liquid circulates through the pipes and the gas passes between the fins on the outside of the pipe, where fins make it possible to balance the lower heat transfer coefficient of the gas. Tube-fin heat exchangers are therefore used extensively for condensers and evaporators for various applications.

Similarly to plate-fin heat exchangers, a variety of fins can be used, as visualised in 2.3. In general, round or rectangular tubes are used for tube-fin heat exchanger, although elliptical can also be found. These can be internally finned to improve the heat transfer. To further increase the heat transfer, mixing can be induced by using vortex generators in the form of winglets (Awais & Bhuiyan (2018)).

Awais & Bhuiyan (2018) and Qasem & Zubair (2018) provide a review of correlations

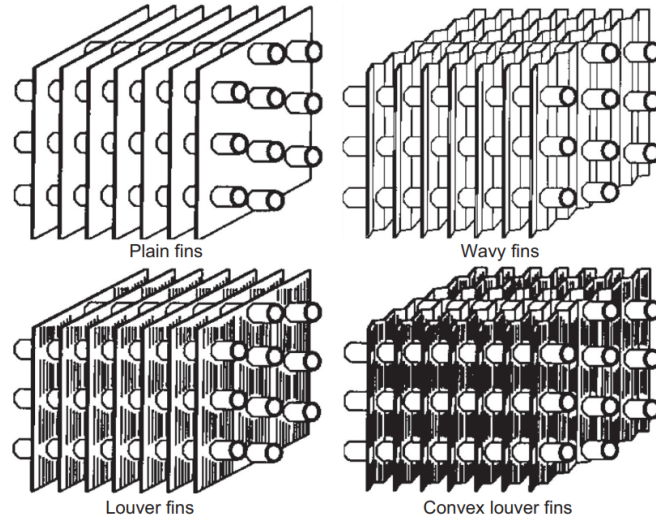


Figure 2.3: Example of fin geometries used within Tube-fin heat exchangers. (Taken from [Awais & Bhuiyan \(2018\)](#))

Reprinted from International Journal of Mass and Heat Transfer, Volume 127, Muhammad Awais, Arafat A. Bhuiyan, Heat and mass transfer for compact heat exchanger (CHXs) design: A state-of-the-art review, p.362, Copyright (2025), with permission from Elsevier.

for characterising the heat and hydraulic performance of tube-fin heat exchangers with a variety of fin geometries.

2.2.2.4 Printed circuit heat exchanger

Printed circuit heat exchangers are created by assembling individual plates with etched fluid passages with a typical hydraulic diameter of 1.5 – 3.0 mm, through diffusion bonding. The diffusion bonding process allows for a heat exchanger with high pressure containment and good anti-corrosion properties, while the size of the etched fluid passages guarantees high surface density.

A variety of internal channel geometries can be created from the manufacturing process as shown in Figure 2.4, depending on desired loading conditions. The etching process allows for various internal flow stream configurations leading to a large range of design freedom. The main drawback of printed circuit heat exchangers is small porosity of the heat exchanger leading to increased size and weight of the system. Due to the small size of the channels, the flow resistance is important, limiting the application range.

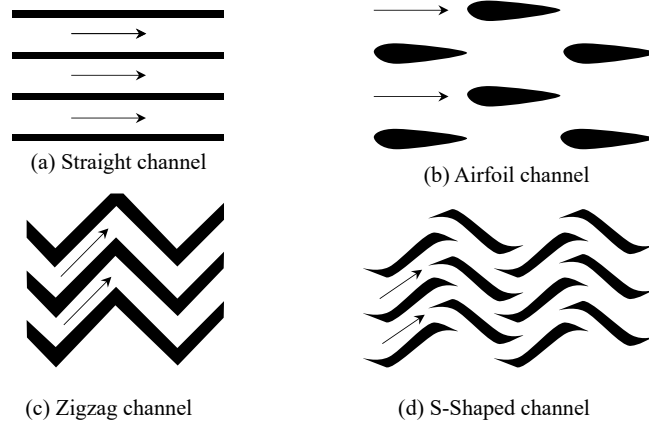


Figure 2.4: Examples of common Printed circuit heat exchanger channel geometries.

2.3. Predicting the performance of compact heat exchanger

Predicting the heat transfer and hydraulic performance is paramount for designing optimum compact heat exchanger. This was initially limited to analytical models and experimental testing of the heat exchanger as reviewed in [Kays & London \(1964\)](#). In recent years, computational fluid dynamics (CFD) has become a useful tool for conducting parametric studies and understanding the flow within the heat exchanger.

2.3.1. Analytical models

Various analytical model such as the Effectiveness-Number of Transfer Units ($\epsilon - NTU$) and Log-Mean Temperature Difference (LMTD) model have been developed to predict the performance of the heat exchanger. These models provide a good initial guess on the performance of the heat exchanger, although the results then need to be validated against experiments.

The models are equivalent, but the $\epsilon - NTU$ model provides an inherent advantage in problems, making it a more popular choice for predicting the performance. The thermal performance for an already existing exchanger can be found analytically by using the $\epsilon - NTU$. For the LMTD method, an initial guess of the outlet temperature has to be provided and the process has to be repeated iteratively until a correct temperature difference is obtained. A quick overview of the $\epsilon - NTU$ will be given here under and a more detailed description will be provided in Chapter 7 for a phase change case.

To perform the heat transfer analysis of a heat exchanger, two basic relationships are required, an energy balance and a rate equation for heat transfer. The heat transfer rate equation can be derived as:

$$\frac{dq}{dA} = U(T_h - T_c) \quad (2.3)$$

where dq/dA is the heat transfer per unit area, U is the overall heat transfer coefficient and $(T_h - T_c)$ is the temperature difference between the hot and cold side. U can be obtained from a thermal resistance network, where $R = (UA)^{-1}$ for the different layers of the heat exchanger:

$$\frac{1}{U_h} = \frac{1}{\eta_{0,h}h_h} + \frac{t_w}{(A_w/A_h)k} + \frac{1}{(A_c/A_h)\eta_{0,c}h_c} \quad (2.4a)$$

$$\frac{1}{U_c} = \frac{1}{\eta_{0,c}h_c} + \frac{t_w}{(A_w/A_c)k} + \frac{1}{(A_h/A_c)\eta_{0,h}h_h}, \quad (2.4b)$$

where h is the film heat transfer coefficient, t_w is the plate thickness, A_w , A_h and A_c are the heat transfer areas of the wall, hot and cold sides respectively, where the area of the wall is taken as an average of the hot and cold sides ($A_w = (A_c + A_h)/2$). Additional resistances can be added to the overall heat transfer coefficient to consider fouling factors.

When extended surfaces are used, a temperature gradient develops along the fins and extends into the fluid. This will lead to a loss in the heat transfer effectiveness, denoted by the term η_0 . Relations have been proposed by [Gardner \(1945\)](#) to take this into account by calculating the surface effectiveness ($\eta_{0,f}$):

$$m = \sqrt{\frac{2h}{k_w\delta}}, \quad (2.5)$$

where k_w is the thermal conductivity of the fin and δ is the fin thickness. This can then be used to calculate the fin efficiency:

$$\eta_f = \frac{\tanh(ml)}{ml}, \quad (2.6)$$

where l is the fin length. The surface effectiveness is then found as:

$$\eta_0 = 1 - \frac{A_f}{A}(1 - \eta_f). \quad (2.7)$$

The energy equation can be defined in terms of two dimensionless parameters, the effectiveness ϵ and the number of transfer units NTU . The flow arrangement and

operating conditions defined in terms of the heat capacity rate ratio $C_{\min}/C_{\max}$ will have an impact on the effectiveness:

$$\epsilon = f \left(NTU, \frac{C_{\min}}{C_{\max}}, \text{flow arrangement} \right) \quad (2.8)$$

The effectiveness (ϵ) is a measure of the thermal performance of the heat exchanger where the actual heat transfer rate q is compared to the maximum heat transfer rate $q_{\max}$:

$$\epsilon = \frac{q}{q_{\max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{\min}(T_{h,i} - T_{c,i})} = \frac{C_c(T_{c,o} - T_{c,i})}{C_{\min}(T_{h,i} - T_{c,i})}, \quad (2.9)$$

The subscripts i and o represent the inlet and outlet, and h and c represent the hot and cold sides respectively. The thermal capacitance is defined as:

$$C = \dot{m}c_p, \quad (2.10)$$

with $C_{\min} = \min[C_h, C_c]$.

The number of transfer units (NTU) is a measure of the non dimensional thermal size of the heat exchanger. It is defined as the ratio of the overall thermal conductance to the smaller heat capacity rate:

$$NTU = \frac{UA}{C_{\min}} \quad (2.11)$$

Over 150 $\epsilon - NTU$ relationships for various heat exchanger geometries and flow arrangements of single-phase exchangers have been published in the literature according to [Shah \(2006\)](#). Table 2.1 gives a quick overview of the most common cases.

Table 2.1: $\epsilon - NTU$ relationship for common flow arrangements, where $C_r = C_{\min}/C_{\max}$.

Type	$\epsilon - NTU$ relationship
Parallel Flow	$\epsilon = \frac{1 - \exp[-NTU(1+C_r)]}{1+C_r}$
Counter Flow	$\epsilon = \frac{1 - \exp[-NTU(1-C_r)]}{1-C_r \exp[-NTU(1-C_r)]}$
Cross flow (both fluids unmixed)	$\epsilon = 1 - \exp\left[\frac{\exp(-nNTUC_r)-1}{nC_r}\right]$ where $n = NTU^{-0.22}$
Cross flow (both fluids mixed)	$\epsilon = \left[\frac{1}{1 - \exp(-NTU)} + \frac{C_r}{1 - \exp(-NTUC_r)} - \frac{1}{N}\right]^{-1}$
Phase change case ($C_r = 0$)	$\epsilon = 1 - e^{-NTU}$

To model the pressure drop in heat exchanger accurately, four main variables have to be taken into account: flow entrance, core friction, flow acceleration and flow exit. A relationship for pressure drop can be derived as:

$$\frac{\Delta P}{P_i} = \frac{G^2}{2} \frac{v_i}{P_i} \left[\underbrace{(K_i + 1 - \sigma^2)}_{\text{entrance effect}} + 2 \underbrace{\left(\frac{v_e}{v_i} - 1\right)}_{\text{flow acceleration}} + \underbrace{\left(f_c \frac{A}{A_c} \frac{v_m}{v_i}\right)}_{\text{core friction}} - \underbrace{(1 - \sigma^2 - K_e) \frac{v_e}{v_i}}_{\text{exit effect}} \right] \quad (2.12)$$

where the subscripts i and e indicate inlet and exit components respectively for the following properties:

ΔP - core pressure drop	P - fluid pressure
G - mass flow velocity ($\dot{m}/A_c$)	K - loss coefficient
σ - ratio of free flow area to frontal area	A - Fluid side area
f_c - core friction factor	v - specific volume
A_c - free-flow area of one side	v_m - mean specific volume

2.3.2. Experimental methods

Analytical solutions to determine the heat transfer and pressure drop performance of the surface can only be obtained for relatively simple geometries. Experimental

techniques have for a long time been the only accurate way of determining the performance of extended surfaces and the effect of headers.

An illustration of an experimental rig for determining the thermal performance of heat exchanger corrugations is shown in Figure 2.5 based on steady state steam-to-air technique described in [Kays & London \(1964\)](#). The tested heat exchanger is made of a known steam and an unknown air side, on which the surface to be tested is mounted. During the experiment, the fluid temperature is measured upstream and downstream of the test section on both fluid sides, which is used to determine the heat transfer coefficient (h) or Colburn factor (j) for the unknown surface. Furthermore, the flow rate and pressure drop of the air side is recorded to measure the friction factor (f).

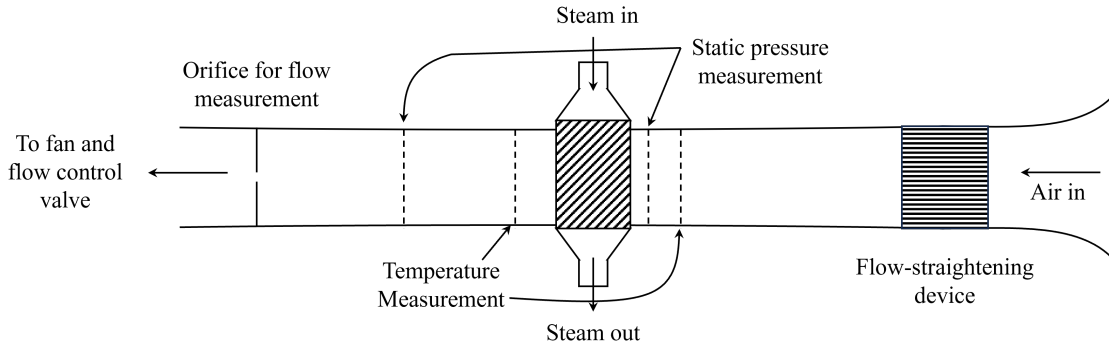


Figure 2.5: Test rig used for steady-state steam-to-air experiments. (Adapted from [Shah \(2003\)](#))

The main source of uncertainty in determining the heat transfer arise from the inaccurate magnitude of thermal resistances and the right range of NTU according to [Shah \(2003\)](#). The first source of uncertainty is reduced by using a fluid with a high thermal capacitance that flows at constant flow rate in the known side, such as steam, water or oil. This ensures that the air side remains the controlling thermal resistance by maintaining $C_r < 0.2$. The wall resistance can be minimized by using a thin material with high thermal conductivity. The second source of errors arises when $NTU < 0.5$, linked to inaccurate temperature and mass flow rate measurements. A high NTU resulting in a low airflow rate can lead to a rollover or a drop-off in the j vs Re curve, where a smaller j value is predicted from the experiment. Great care also needs to be taken when manufacturing the test core, where flaws in the thermal bond of the fins to the primary surface, flow maldistribution due to header or manifold design can have a significant impact on the measurements.

2.3.3. Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is an essential tool in modern heat exchanger design and development. By solving the Navier-Stokes and energy equations, CFD provides detailed insights into temperature distribution, flow patterns and pressure drops. This enables engineers to study the effects of flow maldistribution and novel surface enhancement techniques. As a cost-effective alternative to experimental testing, CFD supports extensive parametric studies, streamlining the design process by allowing engineers to identify and focus on the most promising prototypes for physical testing. Different numerical methods can be employed for this purpose, such as the finite volume (FV) and the finite element (FE) methods to approximate solutions to user-defined geometries. Figure 2.6 outlines the process required for developing a CFD model.

2.3.3.1 Governing equations

Three governing equations are required to capture the flow and heat transfer through the heat exchanger: continuity of mass, momentum and the energy equations. Due to the low Mach number, the flow is assumed to be incompressible. By assuming constant material properties and a negligible effect of gravity, this reduces the equations to the following:

$$\frac{\partial u_i}{\partial x_i} = 0, \quad (2.13a)$$

$$\rho \left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2} \quad (2.13b)$$

$$\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\alpha \frac{\partial T}{\partial x_j} \right). \quad (2.13c)$$

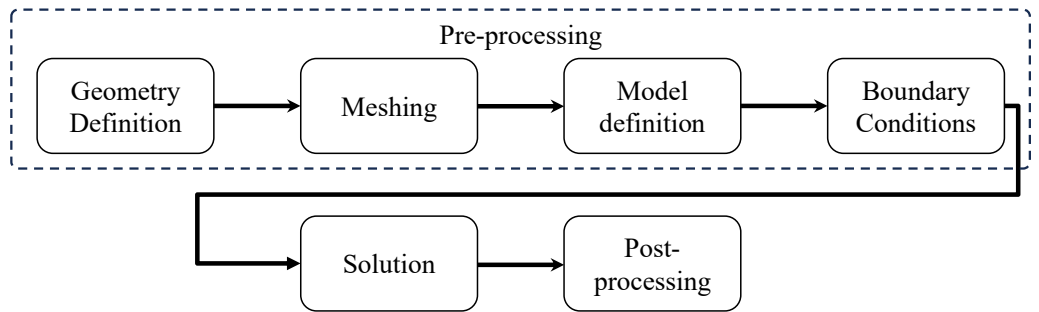


Figure 2.6: Process of developing a CFD model.

u - Velocity	T - Temperature
P - Pressure	$\alpha = k/(\rho c_p)$ - Thermal diffusivity
μ - Dynamic Viscosity	k - Thermal conductivity
ρ - Density	c_p - specific heat capacity

2.3.3.2 Turbulence modelling

Flows become unstable above a certain Reynolds number, leading to the onset of turbulence. In this regime, the flow is chaotic and random, where the velocity, pressure and temperature changes continuously in time. Due to the complex shapes leading to an increased mixing, the onset of turbulence occurs significantly earlier in compact heat exchanger as was noted by [Hiramatsu et al. \(1992\)](#). As a result, it is paramount to capture the turbulence to ensure accurate results.

Equations 2.13 are capable of fully resolving the fluid flow in the turbulent regime at all its scales, known as Direct Numerical Simulations (DNS). However, the computational cost associated with DNS is often too high for modelling the flow around complex heat exchangers geometries. Various turbulence models have been developed to overcome this issue, such as Large Eddy Simulations (LES) and Reynolds Average Numerical Simulations (RANS).

RANS models are most widely used for engineering purposes. In these models, the mean flow is computed by obtaining a time-averaged solution:

$$\bar{u}_i = \frac{1}{T} \int_0^T u_i dt, \quad (2.14)$$

such that the velocity components can be decomposed into a mean velocity component ($\bar{u}$) and a fluctuating part (u'):

$$u_i = \bar{u}_i + u'_i. \quad (2.15)$$

By substituting this into the Navier-Stokes equation, the following RANS equations can be obtained:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad (2.16a)$$

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial \bar{P}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \rho \frac{\partial \overline{u'_i u'_j}}{\partial x_j} \quad (2.16b)$$

The last term in Equation 2.16.b is called the Reynolds stress term, which allows for the turbulent disturbance of the flow. This creates a closure problem that different RANS models try to address by modelling the stress term as a function of the mean flow. This was first tackled by Boussinesq by introducing the following relationship:

$$-\rho \overline{u'_i u'_j} = 2\mu_t S_{ij} - \frac{2}{3}k\rho\delta_{ij}, \quad (2.17)$$

where $S_{ij} = \frac{1}{2} \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right)$ is the stress mean tensor, μ_t is the turbulent viscosity, $k = \frac{1}{2} \overline{u'_i u'_i}$ and δ_{ij} is the Kronecker delta.

Similarly, the energy equation can be rewritten to account for the effect of turbulence as:

$$\frac{\partial \overline{T}}{\partial t} + \overline{u_j} \frac{\partial \overline{T}}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\alpha_{\text{eff}} \frac{\partial \overline{T}}{\partial x_j} \right), \quad (2.18)$$

where $\alpha_{\text{eff}} = \alpha + \mu_t/(\rho Pr_t)$ is the effective thermal diffusivity.

By substituting the Boussinesq approximation into the Reynolds equation, this removes the Reynolds stress term from the equation, but introduces two new variables k and μ_t that have to be estimated. A number of models have been introduced for this purpose, such as the $k - \epsilon$ (see [Launder & Sharma \(1974\)](#)), $k - \omega$ (see [Wilcox \(1988\)](#)) and $k - \omega$ *SST* models (see [Menter \(1994\)](#)). The $k - \epsilon$ model was commonly used in earlier compact heat exchanger studies but was found to underestimate the Nusselt number in the turbulent regime in [Amano \(1987\)](#), linked to the bad prediction of wall bounded flows. [Greiciunas \(2019\)](#) compared the laminar, $k - \epsilon$ (see [Wilcox \(1988\)](#)) and $k - \omega$ *SST* models on a wavy finite length channel. This found a great deviation in the pressure drop and Nusselt number prediction between the $k - \epsilon$ model with the other turbulence models. [Ning et al. \(2022\)](#) simulated novel heat exchanger designs and tested various turbulence models. They found that the $k - \epsilon$ models showed large deviation above 50% with experimental results, while a better agreement was observed with the $k - \omega$ models. The smallest deviation was observed with the $k - \omega$ *SST* model where less than 10% difference was observed with experiments. This is likely due to the capability of the $k - \omega$ *SST* model to capture boundary layer flows by resolving the specific turbulence dissipation (ω) and its ability to model the free stream in a similar fashion to $k - \epsilon$ models ([Marcinkowski et al. \(2024\)](#)). Based on these results, turbulence will be modelled by using the $k - \omega$ *SST* model to validate flow in finned heat exchanger in Chapter 5.

2.3.3.3 Conjugate heat transfer

The conjugate heat transfer assumption is used as part of this project, where the fluid and solid domains are coupled by assuming the temperatures of the fluid and the solid phases are the same at the interface. Further to this, the heat flux entering one region should be equal to the heat flux leaving the other region at the interface.

$$T_f = T_s, \quad (2.19a)$$

$$k_f \frac{dT_f}{dn} = -k_s \frac{dT_s}{dn}, \quad (2.19b)$$

where n is the normal direction to the wall and k_f and k_s are the thermal conductivity of the fluid and the solid, respectively. Heat transfer through the solid material is modelled by using a heat conduction equation:

$$\frac{\partial T}{\partial t} = \frac{\partial}{\partial x_j} \left(\alpha \frac{\partial T}{\partial x_j} \right). \quad (2.20)$$

2.3.3.4 Finite volume method

Different numerical methods exist to solve partial differential equations, including the finite volume, finite element, lattice Boltzmann, etc. The Finite Volume Method (FVM) is the numerical method of choice for a majority of industrial CFD applications. It is valid for any type of grid so that it can be applied to irregular shapes commonly found in industrial designs, and it is inherently conservative and stable for a wide range of applications.

The conservation law for the transport of a general flow variable ϕ will be used to introduce the FVM in this section. The balance equation for unsteady convection-diffusion can be defined as a transport equation:

$$\underbrace{\frac{\partial \rho \phi}{\partial t}}_{\text{Temporal derivative}} + \underbrace{\frac{\partial}{\partial x_j} (\rho u_j \phi)}_{\text{Convective term}} - \underbrace{\frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \phi}{\partial x_j} \right)}_{\text{Diffusion term}} = \underbrace{S_\phi}_{\text{Source term}}, \quad (2.21)$$

where Γ is the diffusivity. Equation 2.21 can be integrated over a control volume, giving:

$$\int_{CV} \frac{\partial}{\partial t} \rho \phi dV + \int_{CV} \frac{\partial}{\partial x_j} (\rho u_j \phi) dV - \int_{CV} \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \phi}{\partial x_j} \right) dV = \int_{CV} S_\phi dV. \quad (2.22)$$

To ensure conservation across the boundaries of each control volume CV , it is nec-

essary to express the fluxes at the surfaces. The balance in the previous equation can be converted into a surface integral form by using the divergence theorem. This results in the following equation:

$$\begin{aligned} \int_{CV} \left(\int_{\Delta t} \frac{\partial}{\partial t} (\rho \phi) dt \right) dV + \int_{\Delta t} \left(\int_S (\rho U \phi) n_j dS \right) dt - \int_{\Delta t} \left(\int_S \Gamma \frac{\partial \phi}{\partial x_j} n_j dS \right) dt \\ = \int_{\Delta t} \left(\int_{CV} S_\phi dV \right) dt, \quad (2.23) \end{aligned}$$

where n_j is the normal vector to the surface element dS . In the FVM, the domain is discretized by splitting the geometry into control volumes with planar faces as shown in Figure 2.7, so that the exact conservation of relevant properties can be calculated for each finite size cell size. The values of all variables are stored in the centroid of the control volume and the solution is interpolated from the cell centres to the face centres, by using appropriate numerical schemes.

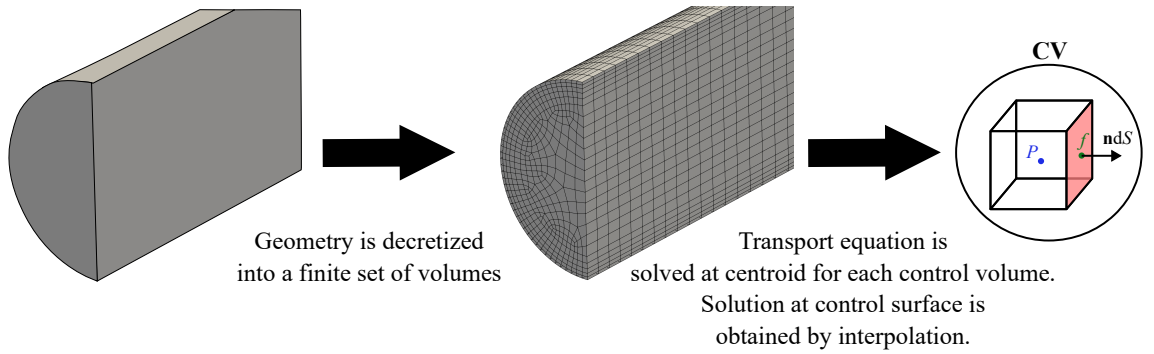


Figure 2.7: Summary of the Finite Volume Method

This creates a set of algebraic equation that are stored within a matrix and which can be solved numerically:

$$\begin{bmatrix} a_{11} & a_{12} & \dots & a_{1n} \\ a_{21} & a_{22} & \dots & a_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ a_{n1} & a_{n2} & \dots & a_{nn} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ \vdots \\ b_n \end{bmatrix} \quad (2.24a)$$

$$A\vec{x} = \vec{b} \quad (2.24b)$$

2.3.4. Heat exchanger network modelling

Heat transfer network modelling permits the performance assessment of the thermal management system as a whole. Although it is beyond the scope of the project it provides the user with initial conditions for HE design and is worth to be noted. It also helps in understanding the behaviour of a heat exchanger design within the system. For this, exergy analysis (EA) and Entropy Generation Minimisation (EGM) methods can be used in conjunction with each other to produce an optimized design. EA combines the first and second law of thermodynamics to provide an understanding on how much work can be extracted from a system as it is brought into equilibrium ([Hayes et al. \(2017\)](#)). EGM focusses on maximizing the efficiency by minimizing irreversibilities in the heat exchange process ([Bejan \(1996\)](#)). This can be used to tune the system based on global constraints by selecting materials, improving flow path design and reducing temperature gradients, as highlighted in [Vargas & Bejan \(2001\)](#), where EGM is used to determine the optimum size for a cross-flow heat exchanger for an ECS.

2.4. Applications of CFD modelling of compact heat exchanger

CFD is increasingly being used at various stages of the heat exchanger design process due to its ability to cut costs by reducing the number of experiments, while providing key insights that help in designing more efficient units. This has led to the development of different models for qualifying the heat transfer and fluid flow, that allow various challenges in heat exchanger design to be studied.

2.4.1. Heat exchanger modelling over different scales

Modelling the whole geometry of a compact heat exchanger by accurately resolving each corrugation is impractical in most cases, due to the multi-scale problem leading to large mesh sizes. As such, different models have been developed to reduce the computational cost.

2.4.1.1 Periodic domain

Patankar et al. (1977) developed a periodic boundary condition applicable to geometries that have periodically repeating features, such as wavy and offset strip fins. This makes it possible to study the thermo-hydraulic performance of a single period by neglecting the effect of entrance length. The velocity components exhibit a periodic behaviour, leading to a linear increase in the pressure drop for each periodic length. The treatment of the fully developed temperature field is more complex and depends on the type of thermal boundary condition that is chosen at the walls. For a constant wall heat flux, the temperature field behaves in a similar manner to the pressure drop, where the temperature gradient is constant. This however assumes that the fluid flow is steady state and laminar, such that a reliable prediction at higher Reynolds number cannot be obtained. For a constant wall temperature, the temperature field is not periodic where the fluid temperature approaches the wall temperature at infinite heat exchanger length. The temperature profile exhibits repeating patterns, such that it can be defined in terms a dimensionless parameter that will be periodically repeating.

The model was applied in Patankar et al. (1977) for a heat exchanger with transverse fins, where large blockage areas and recirculation zones could be identified. Zhang et al. (2004) and Ismail & Velraj (2009) applied this model to study the effect of fin waviness and spacing on the fluid flow and heat transfer. Wang et al. (2009) studied plain and strip fins and compared the numerical simulation results with experimental data from Kays & London (1964), showing good agreement between the two data sets.

2.4.1.2 Finite length channel

A finite length of the flow channel can often provide more detailed information on the fluid flow and heat transfer when compared to periodic model. For instance, transient heat transfer within channels can be evaluated from finite length channels. Zheng et al. (2014) studies seven periodic lengths of wavy fins in a transient laminar

two dimensional model (see Figure 2.8). The flow is shown to develop rapidly over the corrugation, but leads to a chaotic flow pattern that changes over time. Greiciunas (2019) performs a similar study and compares the results with steady periodic domain simulations. They find that the periodic simulations leads to an underestimation of the pressure drop and Nusselt number at higher Reynolds number.

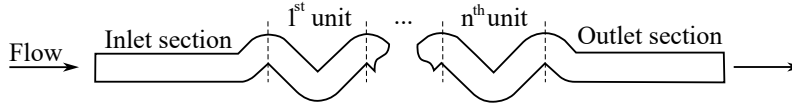


Figure 2.8: 2D Computational domain for modelling transient response in wavy channels in Zheng et al. (2014).

Finite length channel models have been a popular approach for characterising tube-fin heat exchangers, as the small distance between fins allows for cost-effective three-dimensional models. This has been widely applied to study the effects of different geometric parameters and heat transfer performance enhancements. Figure 2.9 highlights an example domain based on work carried out by Menéndez Pérez et al. (2022) on characterising the effect of vortex generators in louvered tube-fin heat exchangers based on j and f factors.

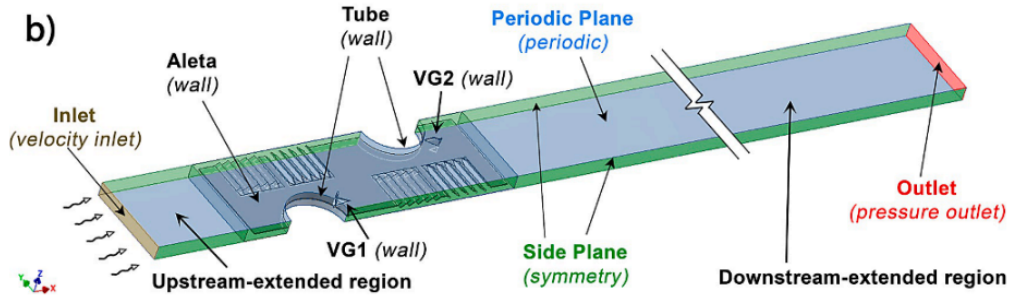


Figure 2.9: Computational domain used by Menéndez Pérez et al. (2022) to study the effect of vortex generators for louvered tube-fin heat exchangers.

Reprinted from Thermal Science and Engineering Progress, Volume 27, Alberto M. Perez, Carlos F.-A. Altamirano, Ruben B. Perez: Parametric analysis of the influence of geometric variables of vortex generators on compact louver fin heat exchangers, p.4, Copyright (2025), with permission from Elsevier.

The finite length channel model allows for more complex heat exchanger problems to be modelled, where the coupling between the two fluid streams and the separating wall can be represented. As such Borrajo-Peláez et al. (2010) applies conjugate heat transfer by modelling a tube fin heat exchanger with an air/water side model and compares this to an air-side only model. It is found that the prediction of Nusselt

number of the two models differs significantly, showing the importance of carefully considering the modelling assumptions.

2.4.1.3 Heat exchanger core modelling

The heat exchanger can be modelled as a whole by approximating the flow as passing through a porous medium. In Greiciunas et al. (2019a) and Greiciunas et al. (2019b), a heat exchanger model based on the porous medium approach is used to model various compact heat exchanger geometries. For this, the pressure drop is evaluated by using a Darcy-Forschheimer Law model or the Power Law model through a source term in the momentum equation. The heat transfer is evaluated by using an analytical model, based on the $\epsilon - NTU$ model, where experimental data or results from heat exchanger corrugation simulation can be used to inform the analytical model. The authors found strong agreement with experimental results in Greiciunas et al. (2019a), where an offset strip fin plate fin heat exchanger is analysed. In Greiciunas et al. (2019b), a novel additively manufactured heat exchanger is evaluated with different modelling techniques, as shown in Figure 2.10. This finds that the porous medium heat exchanger model leads to similar pressure drop findings but overestimates the temperature drop for the more complex geometry.

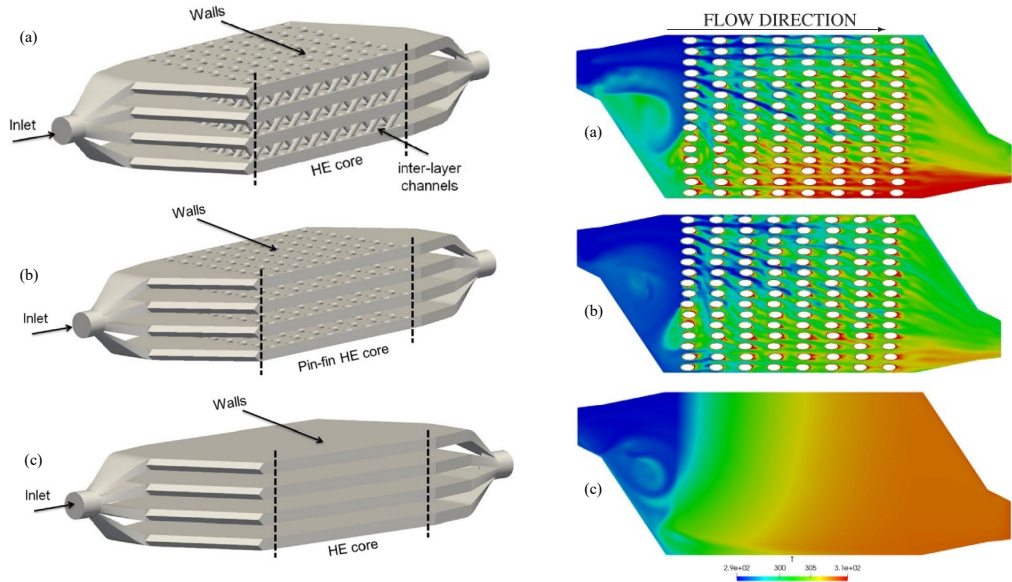


Figure 2.10: Modelling of additively manufactured inter-layer heat exchanger by Greiciunas (2019). The left hand side shows the geometry used for modelling, while right hand side shows the corresponding temperature contours within the cold layer for (a) inter-layer core; (b) Pin-fin core; (c) simplified porous core.

McDermott & He (2024) provides an alternative porous medium approach within a conjugate heat transfer model to determine the thermal performance of printed circuit heat exchanger. Similar to the previous model, a Darcy-Forschheimer source term is used to predict the pressure drop. Additionally, a source term is added to the energy equation based on a volumetric thermal resistance term, which is calculated from the heat transfer coefficient of the surface. When compared to a heat exchanger geometry with airfoil channels, good agreement with experimental results were seen and areas with increased thermal stress were accurately determined by the porous model. This highlights the ability of a porous heat exchanger model to be used to inform design choices.

2.4.2. Overcoming heat exchanger design challenges with CFD

2.4.2.1 Reducing flow nonuniformities

Nonuniformity poses a significant challenge in compact heat exchanger design due to variations in flow and temperature distribution across the exchanger channels. These variations can arise from the geometry of the heat exchanger, manufacturing imperfections, and operational factors like fluid viscosity, density differences, and fouling (Shah (2003)). Flow nonuniformity can occur at various levels: from large-scale discrepancies in flow across the exchanger (gross nonuniformity) to passage-specific variations and manifold-induced inconsistencies. Additionally, multi-stream fluid inlets and insufficient mixing can create nonuniform temperature distributions, especially when fluids of different temperatures enter without full mixing (Singh et al. (2014)). These nonuniformities affect heat transfer efficiency, create hotspots, and increase pressure drop, challenging the optimization of thermo-hydraulic performance.

Flow nonuniformities have been investigated by using CFD in various studies of the header design. Zhang & Li (2003) evaluates the effect of different inlet configurations on plate-fin heat exchangers by using a CFD model created in FLUENT to estimate the effect of introducing baffles to the header for improving the flow uniformity at the inlet. Similar work is underdone by Wen et al. (2006), where a more complex punched baffle is added to the header where larger holes are used in areas of small fluid velocity leading to more uniform flow when compared to the conventional configuration.

2.4.2.2 Improving heat transfer performance of surfaces

Maximizing the thermal and hydraulic efficiency of the heat exchanger is an ongoing research problem, where CFD is used as a tool to understand the influence of geometric changes on the fluid flow and heat transfer. [Awais & Bhuiyan \(2018\)](#) provides an overview of recent advances in heat transfer enhancement techniques with a focus on fin-tube heat exchangers. This highlights that improved heat transfer occurs by introducing interrupted surfaces to create secondary and swirl flows, augmenting turbulence intensity through corrugated and rough surfaces. The thermal boundary layer thickness can also be reduced by using fin interruptions and vortex generators, leading to an increased heat transfer coefficient. [Menéndez Pérez et al. \(2022\)](#) studies the effect of vortex generators in louvered tube fin heat exchangers by varying the geometric parameters. They find that the configuration of the winglets has a significant impact on the performance. The advantage of CFD is highlighted, where the complex fluid flow and the impact on the heat flux is visualized, making it possible to explain the varying heat exchanger performance.

CFD has been used in conjunction with design optimization techniques to improve heat exchanger performance. [Hsieh & Jang \(2012\)](#) found an optimal design for louvered finned-tube heat exchangers by using the Taguchi method. For this, they analysed 18 different designs by varying the geometry of the corrugations, indicating that the fin collar diameter, transverse tube pitch and fin pitch had the highest effects on the performance. [Anibal & Martins \(2024\)](#) used CFD in conjunction with adjoint-based shape optimization to minimize drag and mass in a plate-fin heat exchanger with wavy fins. In [Kobayashi et al. \(2019\)](#), the optimal design of winglets on fin-tube heat exchangers was studied by using topology optimization based on a quality factor of j/f . By comparing a solution to the optimization (shown in Figure 2.11) to conventional winglet designs, the authors observed an increase in the quality factor, leading to enhanced thermo-hydraulic behaviour.

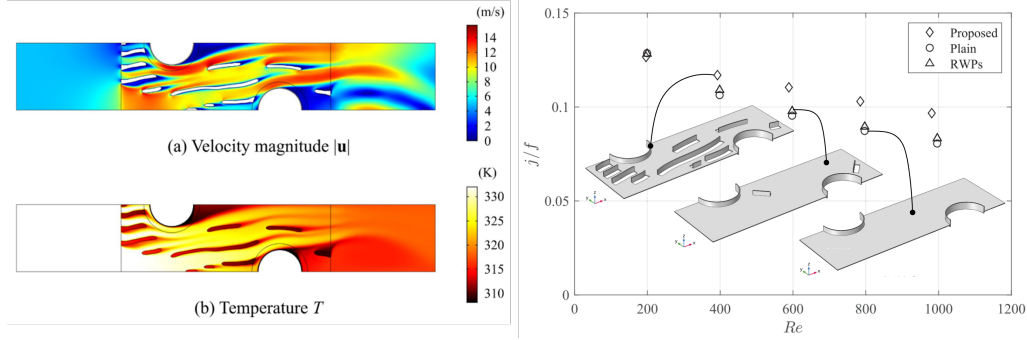
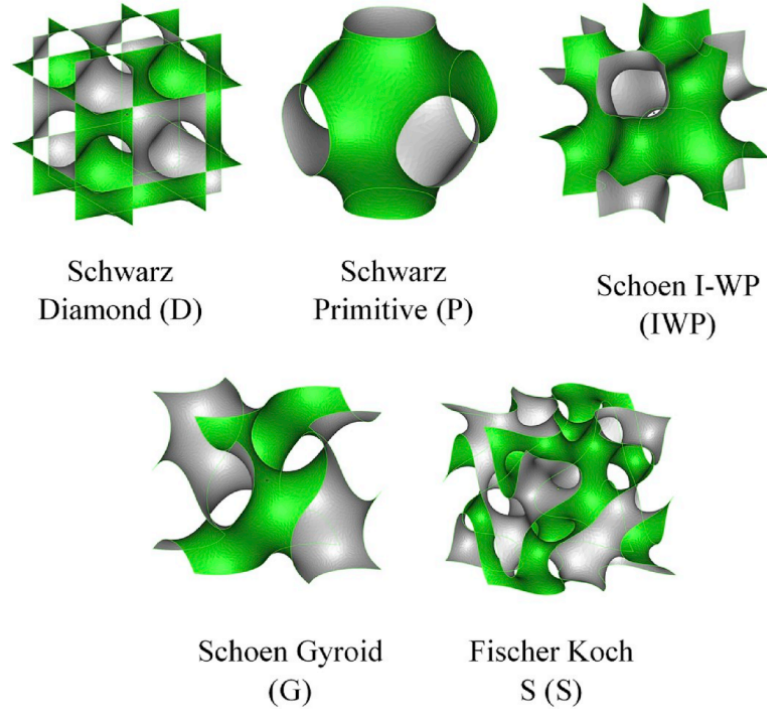


Figure 2.11: Use of optimization methods for improving the thermo-hydraulic performance of heat transfer surfaces: Topology optimization results of winglets for tube-fin heat exchanger compared to traditional designs taken from [Kobayashi et al. \(2019\)](#).

Reprinted from Applied Thermal Engineering, Volume 161, Hiroki Kobayashi, Kentaro Yaji, Shintaro Yamasaki, Kikuo Fujita: Freeform winglet design of fin-and-tube heat exchangers guided by topology optimization, p.7-8, Copyright (2025), with permission from Elsevier.

The development of additive manufacturing has allowed for a greater flexibility in the development of heat exchanger geometries. CFD has been used in conjunction with additive manufacturing to develop novel geometries in an effort to increase the performance while reducing weight and mass of the components. Recently, the number of studies on the use of periodic minimal surface (TPMS) within heat exchangers has exploded. TPMS (see Figure 2.12) can be defined by trigonometric functions based on zero-mean curvature, leading to geometries that are challenging to manufacture with conventional methods. They are characterised by large surface area to volume ratio beneficial to heat transfer applications. [Dixit et al. \(2022\)](#) modelled and manufactured a gyroid TPMS heat exchanger that exhibits a significant increase in the heat exchanger effectiveness when compared to a conventional compact heat exchanger, although the impact on friction was not taken into account in this study. [Attarzadeh et al. \(2022\)](#) conducted a multi-objective optimization study of a TPMS exchanger based on a genetic algorithm to study the effect of operating conditions and geometric features on the temperature and pressure drop. They observe that the performance is particularly sensitive to the wall thickness, pore diameter and periodic length.



(a) Examples of TPMS geometries

Figure 2.12: Heat exchanger designs facilitated by additive manufacturing: Use of TPMS surfaces (Taken from [Al-Ketan et al. \(2020\)](#))
 Reprinted from Journal of the Mechanical Behavior of Biomedical Materials, Volume 102, Oraib Al-Ketan, Dong-Wook Lee, Reza Rowshan, Rashid K. Abu Al-Rub: Functionally graded and multi-morphology sheet TPMS lattices: Design, manufacturing, and mechanical properties, p.2, Copyright (2025), with permission from Elsevier.

2.5. Summary

This chapter provided a comprehensive review of current heat exchanger developments for aerospace applications. Common compact heat exchanger geometries were described and various secondary heat transfer surfaces were introduced to improve the heat transfer performance by disrupting the flow. The following section summarized different methods to estimate the performance of compact heat exchangers, based on analytical models, experiments and CFD simulations. The $\epsilon - NTU$ model is widely used for quickly sizing heat exchangers of varying complexity. However, it relies on experiments or CFD simulations to provide the inputs and the model can overly simplify the geometry leading to inaccuracies. CFD simulations have become a popular way of estimating heat exchanger performance as computational power and CFD codes have become more readily available. By post-processing

CFD results, a greater depth of results can be obtained when compared to analytical models or experiments, making it possible to overcome various heat exchanger design challenges. Early models were focussed on periodic arrays based on the periodic boundary condition developed by [Patankar et al. \(1977\)](#). By modelling a finite length of channels, greater detail can be obtained on the flow development and more accurate results can be generated at higher Reynolds number. The heat exchanger core was approximated as a porous medium in multiple studies, allowing for whole heat exchanger geometries to be modelled. More recently, CFD has been coupled to design optimization tools, pushing heat exchanger development towards its limit.

CHAPTER 3

Review of Phase Change Materials for Energy Storage

Contents

3.1	Introduction	39
3.2	Phase change materials	40
3.2.1	Selection of PCM	40
3.2.2	Heat transfer and phase change phenomenon	43
3.2.3	The mushy zone	44
3.2.4	Constrained and unconstrained melting	46
3.3	PCM heat exchanger geometries	47
3.3.1	Applications of PCM materials	47
3.3.2	Effect of geometries	50
3.3.3	Effect of operating conditions	52
3.3.4	Performance improvement of LHTES Units	53
3.4	Predicting the performance of PCM heat exchanger . .	60
3.4.1	Analytical models	61
3.4.2	CFD models	63
3.4.3	Enthalpy-porosity model	70
3.5	Summary	74

3.1. Introduction

In the last two decades, Latent Heat Thermal Energy Storage (LHTES) has become an attractive solution for storing heat, due to its high energy storage capacity at almost isothermal conditions. LHTES enables the storage and release of heat at almost constant temperature where the unit is designed around a phase change temperature of the phase change material.

Figure 3.1 summarizes the different stages involved in developing LHTES units based on [Sharma et al. \(2009\)](#). The chart displays a difficult process, where careful decisions need to be taken in selecting an appropriate PCM before a heat exchanger unit can be designed for a given application. This is because a wide range of materials are available for which the properties can vary drastically from one another. The material might melt at an appropriate temperature, but not be chemically stable so that a commercialisation would be unsustainable.

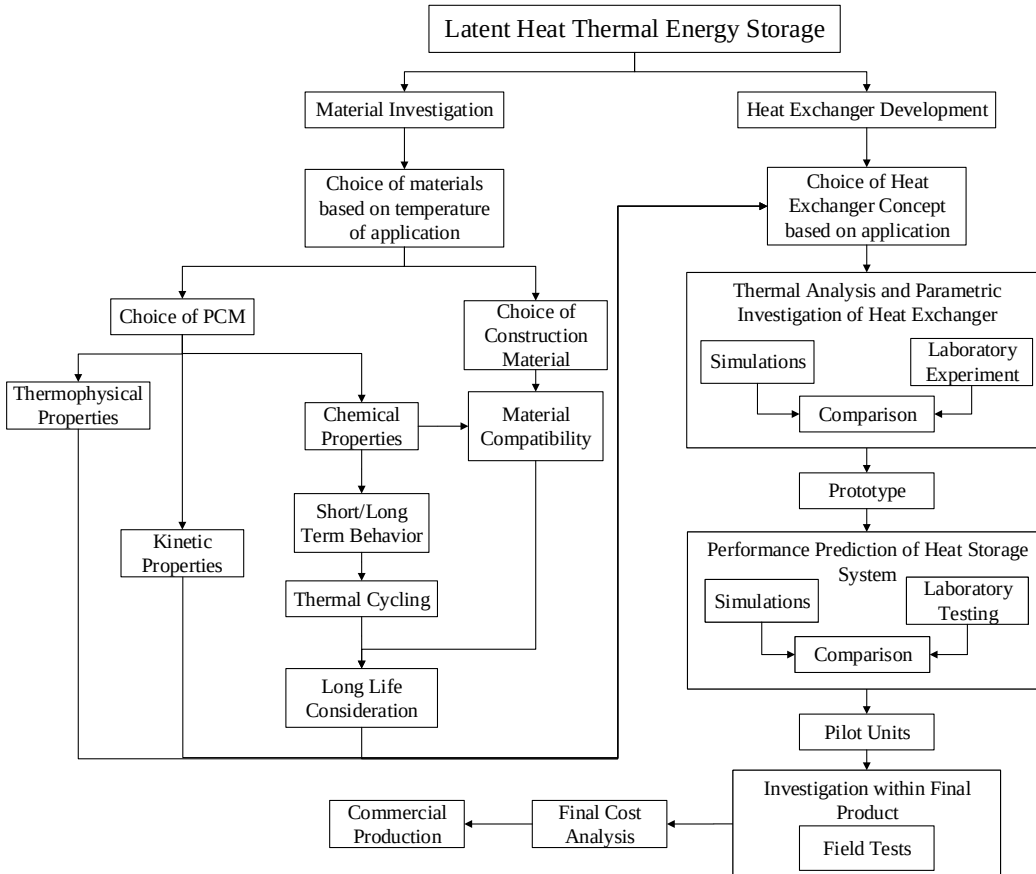


Figure 3.1: Flow chart showing different stages involved in the design and development of a latent heat storage (Adapted from [Sharma et al. \(2009\)](#)).

This chapter will build an understanding on how to select an appropriate phase

change material by providing context on PCM materials and the phase change mechanism in Section 3.2. Following this, an understanding on heat exchanger development will be given by summarizing the literature on existing applications and designs of LHTES units in Section 3.3. Section 3.4 details how heat exchangers are designed by using various analytical and numeric models to qualify the heat transfer and phase change in PCM heat exchanger, with a particular focus on the enthalpy-porosity model that will be used in Chapter 4-6.

3.2. Phase change materials

3.2.1. Selection of PCM

3.2.1.1 Phase Change Mechanism

PCM can be classified based on the phase change mechanism and the material type as shown in Figure 3.2.

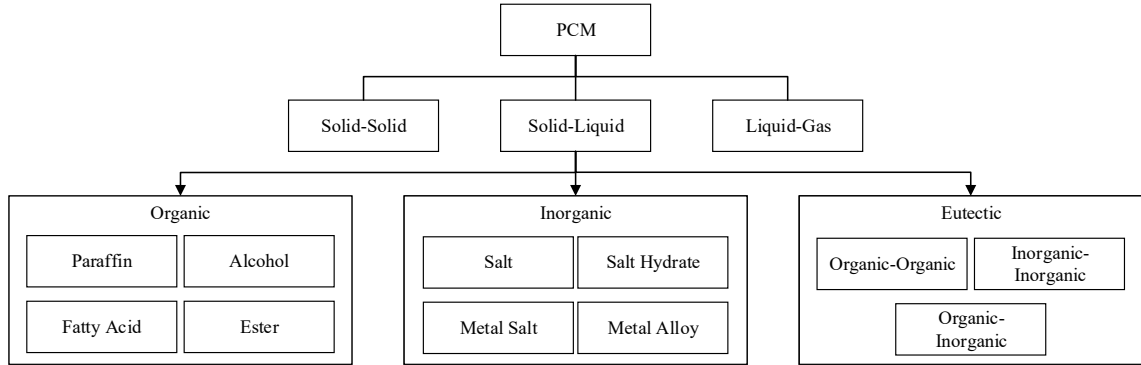


Figure 3.2: Classification and Types of PCM materials

Solid-solid PCM (SS-PCM) rely on the transformation from one crystalline form of the material to another, with promising materials including pentaerythritol, pentaglycerine, Li_2SO_4 and KHF_2 (Sharma et al. (2009)). This type of phase change comes with inherent advantages as they are leakage free, do not require any encapsulation, have great chemical stability and experience smaller volume variation than other types of phase change. This makes it possible for the materials to be directly blended with other material, for example with concrete or gypsum where the composite can be used for thermal energy storage in construction materials (Fallahi et al. (2017)). Unfortunately, SS-PCM are typically associated with lower latent heat, such that less energy can be stored per unit mass and volume. According to Fallahi et al. (2017), this is linked to the mobility restrictions of the molecules in the

solid phase. Liquid-gas (LG) phase change is characterized by a transformation from connected molecules to separated particles. While LG-PCM, have the highest energy density, their applications are limited due to high pressure and volume changes during phase change leading to containment problems (Sharma et al. (2009)). In solid-liquid phase change materials (SL-PCM), the internal molecular arrangement is changed from an ordered crystalline structure to a disordered arrangement. As a result, a relatively large amount of heat can be stored and released by the material during phase change when compared to SS-PCM. The material is usually separated from the working fluid by encapsulating the material, as the SL-PCM experience a significantly smaller volume expansion than LG-PCM, in the order of 10% according to Nazir et al. (2019). This has made SL-PCM the most wide spread used phase change mechanism in LHTES units and this project will focus on this type of materials.

SL-PCM can further be subdivided into types of material. Nazir et al. (2019) differentiates between organic, inorganic and eutectic PCM. Organic materials include paraffin waxes, alkanes, esters, fatty acids and alcohols. Inorganic PCM include salts, salt-hydrates and metal salts, compounds and alloys. Eutectic materials are created from a mixture of components at the eutectic point based on their phase diagram. The mixture can be made of organic, inorganic materials or a mixture of both. Each type of material comes with advantages and disadvantages, which will be discussed in the following section.

3.2.1.2 Selection Criteria

Table 3.1 summarizes the different material property criteria that are key to selecting a PCM for a given application to ensure the LHTES will have the planned performance over its lifecycle. The criteria can generally be subdivided into thermo-physical, chemical, economic and environmental properties (Nazir et al. (2019)).

Notably, PCM are generally characterised by relatively low thermal conductivity, which is a significant obstacle for the design of LHTES, leading to slow phase change. Additionally, supercooling has been an issue with certain PCM, especially inorganic salt hydrates, where the solidification of the material is postponed. This can interfere with heat absorption and makes them impracticable for industrial applications, where a narrow and accurate working temperature range is required (Nazir et al. (2019)). Table 3.2 summarizes the advantages and disadvantages of organic, inorganic and eutectic materials in terms of material properties.

Table 3.1: Selection criteria for phase change materials (Adapted from [Sharma et al. \(2009\)](#)) and [Nazir et al. \(2019\)](#)).

Thermo-physical properties	• Suitable phase-transition temperature • High latent heat of phase-transition • High thermal conductivity • High specific heat capacity • Phase stability • High density • Small thermal expansion • Limited supercooling
Chemical properties	• Long-term chemical stability • Compatibility with encapsulation materials • Limited toxicity • Limited flammability
Economic properties	• Inexpensive • Commercial availability
Environmental properties	• Low impact on environment over product life • Recyclable

Table 3.2: Selection of PCM based on material type (Adapted from [Sharma et al. \(2009\)](#) and [Nazir et al. \(2019\)](#)).

PCM	Advantage	Disadvantage
Organic	• Good energy storage capacity • Non-corrosive and low supercooling properties • Low volume change • Good chemical stability • Low cost • Can come from renewable sources	• Low thermal conductivity • Flammability • Can become unstable at high temperature • Low phase change temperature
Inorganic	• High energy storage capacity • Higher thermal conductivity • Large temperature range available • Relatively inexpensive	• Less stable than organic: - Phase segregation can occur - Can react with encapsulation material - Incongruent melting and precipitation
Eutectic	• Increased stability • Adjustable phase change temperature based on choice of materials for mixture	• Can have similar drawbacks to the organic or inorganic PCM • Expensive

Additives can be added into the PCM to improve its material properties. For instance a small amount (1 – 3% in weight) of thickening and gelling agents have been used to address some of the stability problems involved in using PCM, including supercooling, agglomeration and leakage. A thorough review of current achievements as well as current challenges is provided in [Cong et al. \(2022\)](#). Multiple materials are available with a wide range of operating temperature, including medium to high temperature applications. However, the addition of thickening agents has profound impacts on the material properties of the PCM. For instance, the addition of the agent can lead to a change in the viscosity profile and the material compatibility of the PCM.

Simultaneously meeting all design criteria is complicated. Engineers have to make informed decisions by conducting trade-off studies based on the desired properties, availability and cost of the materials. To address this, decision-taking tools have been introduced in the literature. [Barreneche et al. \(2014\)](#) use the CES Selector software package to create a database of three hundred PCM materials to compare the melting temperature and enthalpy of fusion for the different materials. [Khare et al. \(2012\)](#) uses the same tool for selecting high temperature PCM with a phase change temperature between 420°C and 750°C, based on high energy density, volumetric heat of fusion and thermal conductivity. The authors used a multi-objective optimisation technique to minimise the mass per unit energy storage and cost, before considering the environmental footprint of the materials by using the CES EcoAudit tool. [Socaciu et al. \(2016\)](#) use an analytic hierarchy process (AHP) for ranking ten commercial PCM based on a wide range of thermophysical properties, where different weights are attributed based on the importance of each criteria.

3.2.2. Heat transfer and phase change phenomenon

Figure 3.3 provides an example problem of a horizontal PCM cavity with gravity acting downwards taken from [Jany & Bejan \(1988\)](#). One side of the domain is continuously heated while all other walls are kept adiabatic. Initially, the PCM is fully solid, and as the temperature of the wall is higher than the temperature of phase change of the material, the PCM will start to melt. At this early stage, the heat transfer is dominated by conduction, so that the solid/liquid interface of the PCM develops normal to the heated boundary (see Figure 3.3.a). As time progresses, the melted pool of PCM starts to convect, where the hotter PCM is convected towards the top of the cavity. As this happens, the shape of the interface adapts based on

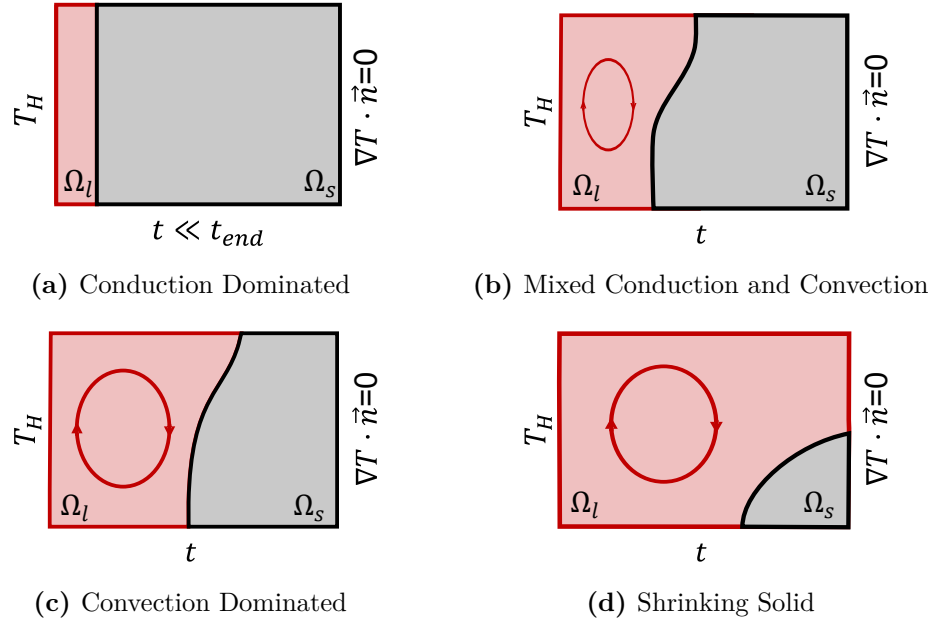


Figure 3.3: Heat Transfer Phenomena occurring during melting of PCM based on Jany & Bejan (1988)

the convection of the PCM, where more of the PCM is melted at the top of the cavity than at the bottom (see Figure 3.3.b). Convection becomes more dominant in the rest of the cavity in Figure 3.3.c. In Figure 3.3.d, the PCM shrinks in the lower half of the cavity until the melting of the PCM is complete when all of the PCM is fully liquid.

3.2.3. The mushy zone

In Figure 3.3, the phase change front is assumed to be a sharp interface. In most engineering applications, this is rarely the case, as isothermal phase change is limited to eutectic or pure materials, so that a third region, the mushy zone develops. Voller et al. (1990) distinguishes between 3 different types of phase changes shown in Figure 3.4:

- **Distinct:** Distinct solid and liquid phases separated by a smooth continuous front, where the phase change occurs at a distinct temperature *e.g. freezing of pure water; rapid solidification of pure metals.*
- **Continuous:** Liquid and solid phases are fully dispersed in the phase change region with no distinct interface between the solid/liquid phases. In this case, the phase change occurs over a range of temperatures. *e.g. solidification of polymers or glasses.*

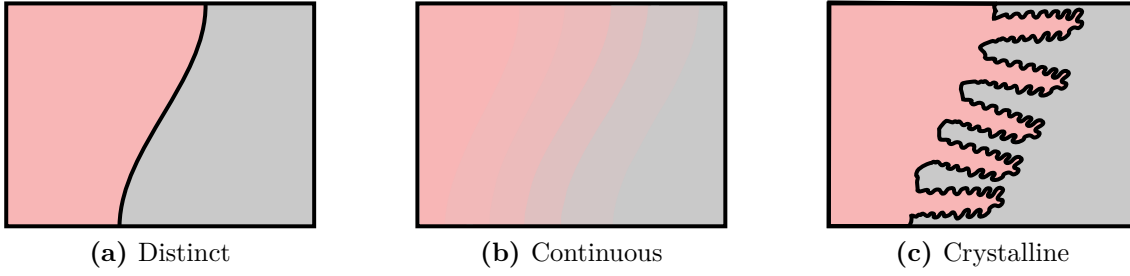


Figure 3.4: Different Types of Phase Change: (a) Distinct, (b) Continuous, (c) Crystalline.

- **Crystalline:** Crystalline structure consisting of columnar or equi-axed grains, so that the solid/liquid interface results in a complex shape that is not necessarily continuous. *e.g. solidification of most metal alloys, paraffin wax.*

Yang et al. (2019) study the microstructural evolution of a paraffin wax during melting and solidification. They observe three regions during the phase change process, linked to the finite temperature range at which the material solidifies and melts: solid, liquid and a mushy zone. As can be observed in Figure 3.5, the authors observe two different structures during the phase change. During solidification, a dendritic growth with some free crystals is observed, while during melting no distinct crystals are apparent. These structures are typically 3 order of magnitude smaller than the width of the mushy zone, such that this becomes a multiscale problem.

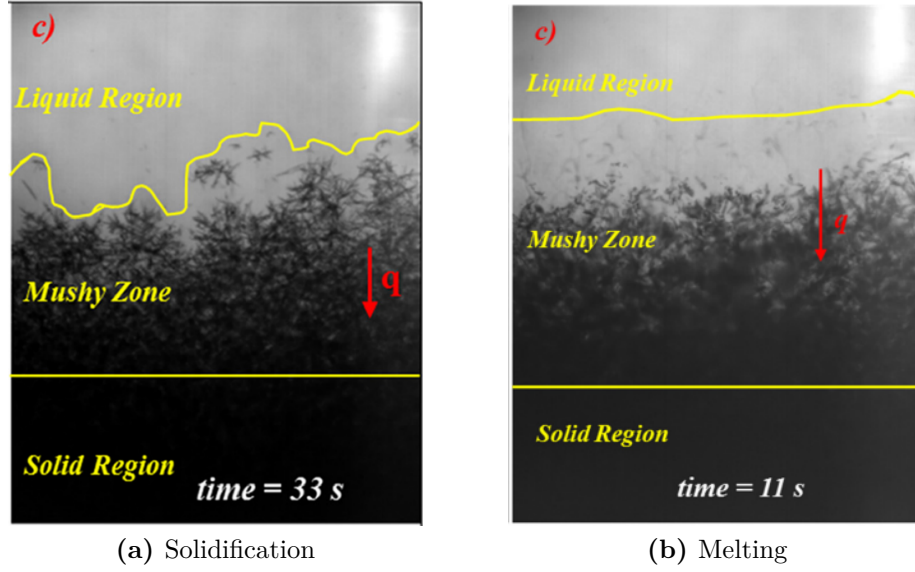


Figure 3.5: Microstructure during (a) solidification and (b) melting of paraffin wax as observed by Yang et al. (2019). This highlights the differences in microstructure between solidification and melting.

Reprinted from International Journal of Heat and Mass Transfer, Volume 141, Bei Yang, Aikifa Raza, Fengwu Bai, Tiejun Zhang, Zhifeng Wang: Microstructural evolution within mushy zone during paraffins melting and solidification, p.772, 774, Copyright (2025), with permission from Elsevier.

3.2.4. Constrained and unconstrained melting

In certain cases, the solid PCM can sink to the bottom of the encapsulation if it is not constrained by the geometry (see Figure 3.6 for a comparison of constrained and unconstrained melting). In constrained melting, the PCM is maintained in place by the geometry of the encapsulation, where the main driving force of the melting is the natural convection of the PCM pool. In unconstrained melting, often referred to as close contact melting (CCM), the solid PCM will be pushed towards one of the heated surface during melting due to the difference in density of the solid and liquid phase of the PCM. As the solid PCM approaches the wall, a thin liquid film layer is created between the heated surface and the solid PCM.

An experimental study by Abdi & Rastan (2023) compares constrained and unconstrained melting in finned rectangular enclosures. Due to the increased convection induced by the sinking of the solid PCM, as well as the lower distance between heated wall and the solid PCM, unconstrained melting is seen to have a higher energy absorption rate than unconstrained melting.

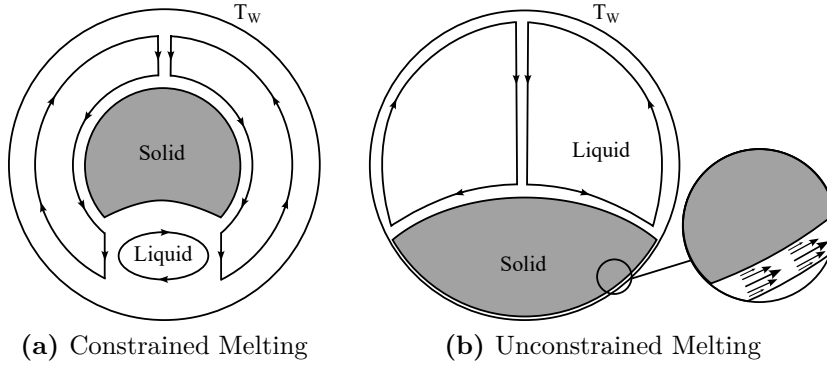


Figure 3.6: Flow description of constrained and unconstrained melting in a cylinder heated from the outside.

3.3. PCM heat exchanger geometries

The current state of the art of LHTES units is reviewed in this section, including common geometries, the effect of operating conditions on the melting of PCM and performance improvement strategies.

3.3.1. Applications of PCM materials

The ability of PCM to stabilize temperatures and store thermal energy makes them highly valuable across various applications that require consistent temperature regulation, energy efficiency, and reliable thermal storage. Figure 3.7 highlights various applications of PCM in the literature and industry. They are most widely used in building construction and Heating, Ventilation and Air Conditioning (HVAC) systems, where they provide intermittent thermal storage to help regulate indoor temperatures. PCM can be encapsulated within the insulation of walls, ceilings, roofs and floors (Oró et al. (2012)), reducing the need for air conditioning and heating. This has been shown to lead to significant energy savings in studies by Lee et al. (2018) and Xie et al. (2018). In HVAC systems, PCM help in shifting heating and cooling loads to off-peak hours, easing system demands and enhancing efficiency. For example, Bruno et al. (2014) found that a 13.5% reduction in energy demand could be achieved by pushing peak loads to off-peak times, provided an appropriate PCM material is chosen.

PCM are critical to solar power applications, where they absorb excess heat during the day and release it at night. In concentrated solar power (CSP) farms, they help manage short-term thermal transients and enable extended operation into the

night, as discussed in [Xu et al. \(2015\)](#). Furthermore, in orbital spacecraft, PCM are essential for passive thermal management and protection, particularly given the low power availability for active systems and the periodic heat exchange with the environment ([Diaconu et al. \(2024\)](#)). They are integrated into the thermal control systems of spacecraft to absorb and store heat when exposed to intense sunlight and gradually release it when the craft moves into shadowed areas, stabilizing internal temperatures and protecting sensitive electronics and instruments from thermal stress.

Less commonly, PCM are applied to cold chain logistics for the transport of refrigerated items in the food and beverage industry, as well as the medical industry. In these applications, they provide emergency cooling in cases of power outages as noted in [Mehling et al. \(2022\)](#). Additionally, PCM are used in electronics and electric vehicles to manage battery and device temperatures, thereby extending the lifespan and performance of these systems.

Applications in aircraft are limited to a small number of studies, focusing mainly on mitigating waste heat from individual avionic systems (see [Lv et al. \(2022\)](#)) or managing battery thermal performance in the context of aviation electrification aimed at achieving net-zero goals ([Li et al. \(2025\)](#), [Kim et al. \(2023\)](#)). Additionally, several studies have explored the integration of PCM heat exchangers and sinks as thermal capacitors for directed energy weapons (DEW) ([Patel et al. \(2010\)](#), [Homitz et al. \(2012\)](#)). DEW can generate high power densities over short durations, typically ranging from $10^1 - 10^2$ seconds, which can quickly overwhelm the aircraft's thermal management system. PCM can help alleviate this issue by temporarily storing excess heat and smoothing out the thermal load. The integration of PCM at a multi-component level of aircraft systems, similar to traditional thermal management system (TMS) architectures, has not been extensively studied in the literature, resulting in redundant PCM heat sinks.

Due to the limited applications of PCM within transportation, the mass and volume is rarely a restrictive factor in the design of LHTEs units. Additionally, a significant amount of the studies design the system to operate over multiple hours, where the PCM melts over the cycle of a day. This has limited the development of compact heat exchangers which would be most applicable to aircraft where the mass and volume of the system is a limiting design criteria and shorter melting cycles are desired.

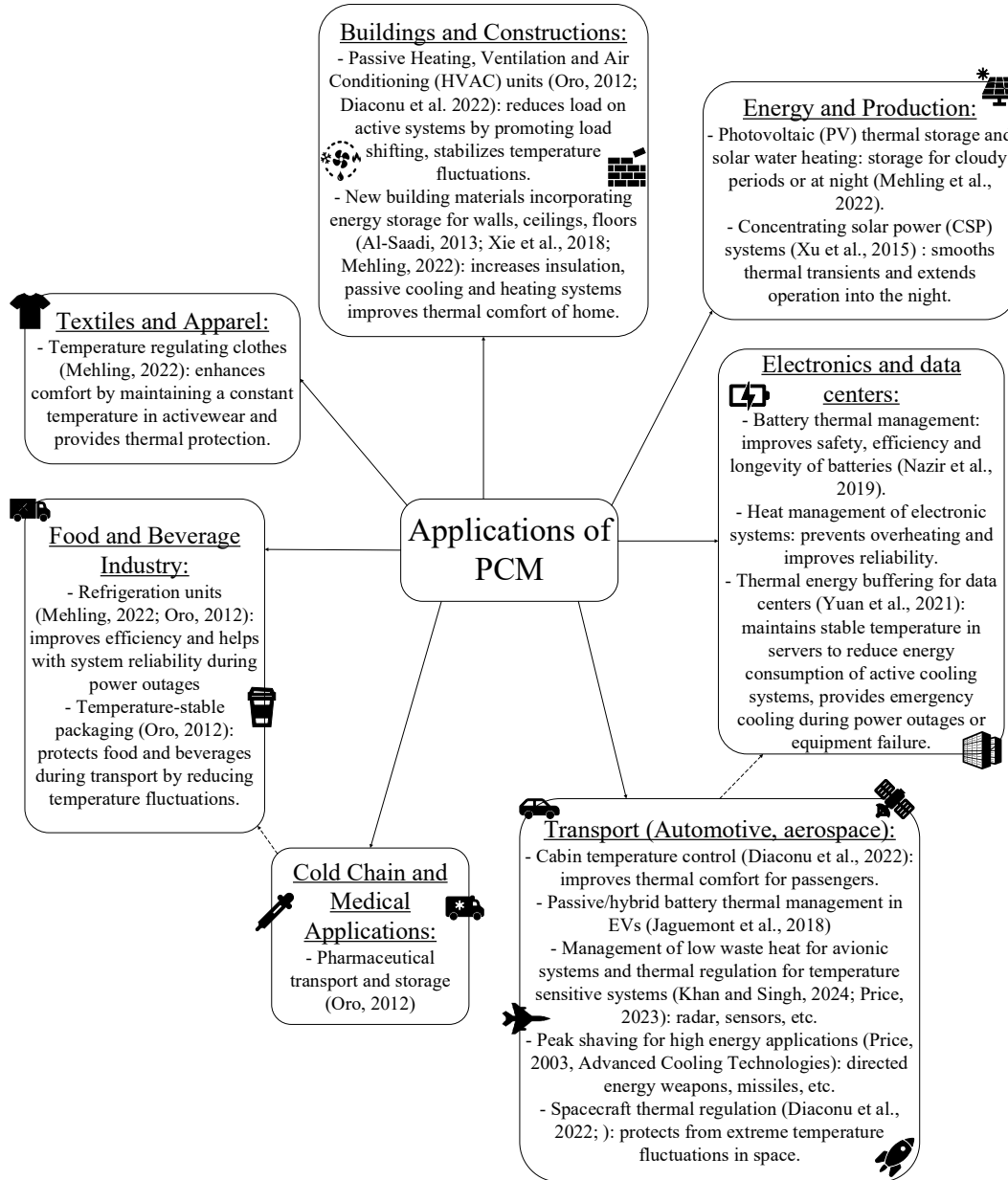


Figure 3.7: Applications of PCM. Graphic is based on the following reviews: Oró et al. (2012), Diaconu et al. (2022). Mehling et al. (2022), Nazir et al. (2019), Xu et al. (2020), Xie et al. (2018), AL-Saadi & Zhai (2013), Yuan et al. (2021), Price (2003), Jaguemont et al. (2018)

3.3.2. Effect of geometries

The choice of container geometry has a great impact on the heat transfer performance of the PCM heat exchanger. [Agyenim et al. \(2010\)](#) distinguishes between two main types of heat exchanger geometries: rectangular and cylindrical units. Cylindrical units are more widely spread in the literature, linked to its compatibility with piping structures, making it easy to incorporate within pre-existing systems.

Within cylindrical units, different configurations are shown in the literature: (a) concentric tubes with encapsulated PCM in the shell and fluid flowing in the tube, (b) concentric tubes with PCM in tube and fluid flowing around the PCM, (c) multitube, (d) triplex tube units where the fluid flows within and around the PCM. These are visualised in Figure 3.8.a-d.

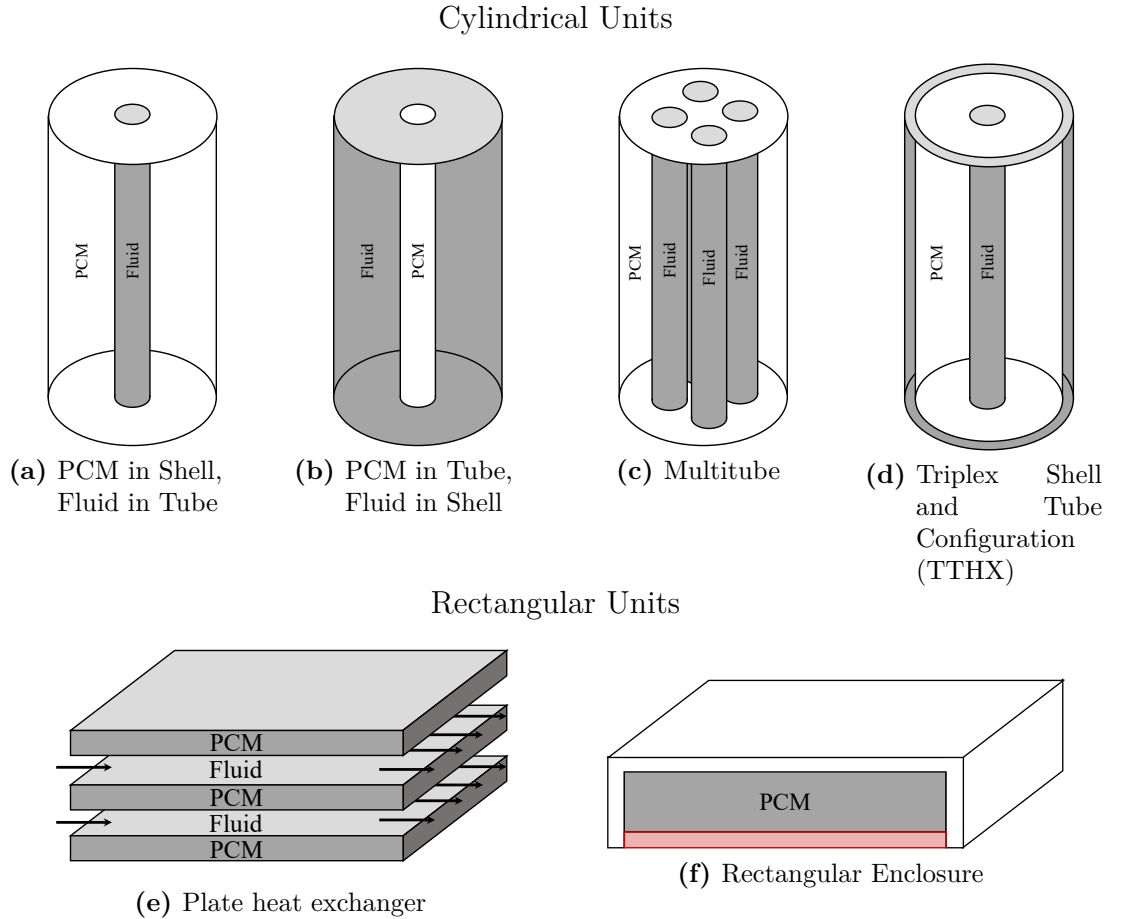


Figure 3.8: Different configurations of LHTES units

[Esen et al. \(1998\)](#) compared geometry type (a) and (b) for various operating conditions and geometric parameters. They found that the PCM in shell and fluid in tube configuration generally led to a lower melting time as there is no heat loss

to the environment in this configuration. During solidification, the heat transfer is mostly dependent on conduction, so that no change in the solidification time was observed. Multitube configurations are a convenient choice for medium to large installations, where a more significant mass of PCM is required as studied by [Esapour et al. \(2016\)](#). This showed that as the number of tubes inside the shell increased, natural convection becomes increasingly dominant, which in turn increases the melting rate. To maximize this effect, the tubes should be placed towards the bottom of the shell. [Baal & Ünal \(2013\)](#) compared the melting inside a triplex tube and double tube heat exchanger. They noted that the triplex geometry could lead to a significant improvement in melting rate as the PCM is heated from two points, so that a reduction in size could be achieved for small to medium sized units.

For the shell and tube design, a shell to tube diameter ratio can be determined as follows [Kalapala & Devanuri \(2018\)](#):

$$R = \frac{\text{Diameter of shell}}{\text{Diameter of tube}} \quad (3.1)$$

In [Cao & Faghri \(1991\)](#), the performance of a shell and tube heat exchanger was analysed by conducting a parametric study where the shell to tube diameter ratio was varied. This showed that as R increases, the total melting time increased, but this also lead to an increase in the energy storage density. In a similar study, [Lacroix \(1993\)](#) studied the effect of varying the shell radius, the mass flow rate and the fluid inlet temperature. This showed that the shell radius needs to be considered carefully as this directly impacts on the storage capacity of the LHTES. [Seddegh et al. \(2017\)](#) find that a value of $R = 5.4$ offers a good compromise between charging time and stored energy fraction.

Plate heat exchangers (see Figure 3.8.e) are also widely studied in the literature, where the PCM is macro-encapsulated within rectangular plates while fluid flows around the plates. [Liu et al. \(2011\)](#) studies the effect of geometric parameters on the melting process of PCM in a plate heat exchangers. They examine the effect of gap between slabs, slab length and thickness at constant PCM mass and find that the geometric parameters only show a small impact on the heat transfer rate. A study by [Saeed et al. \(2019\)](#) finds that the thickness of PCM in between fluid channels has an important impact on the melting and freezing rate of PCM, where a smaller spacing leads to an increase in the energy storage and release rate. This is in line with findings by [Gürel \(2020\)](#) and [Hoseinzadeh et al. \(2018\)](#), but contradicts the findings of [Liu et al. \(2011\)](#). They note that increasing the packing density of fluid

channels can negatively impact the energy density of the heat exchanger due to a reduced volume of PCM, although in the cases studied by the authors, only a small impact on the volume reduction was noted.

Only a limited amount of studies follow the design criteria of high energy density and low melting time that are of interest in this thesis. [Barz & Emhofer \(2021\)](#) conduct a study on the phase transition behaviour of four commercial PCM by using a plate-fin heat exchanger geometry with offset strip fins in both the water and PCM channels at various constant heating and cooling rates, leading to melting time ranging from 6-50 min. The geometry is chosen due to its good heat transfer performance criteria, and operated such that the fluid temperature can be assumed to be constant. The study therefore does not consider the effect of the geometry on the melting for designing efficient heat exchangers. A recent study by [Yuan et al. \(2024\)](#) develops a numerical model for studying plate-fin heat exchanger geometries for mitigating transient thermal shocks in lasers. The operating conditions are similar to the ones that will be studied in this thesis, with a melting cycle of 25 s and a small temperature difference between the PCM phase change temperature and the fluid temperature. They study the effect of initial temperature, fluid temperature and flow rate, but the study does not consider the impact of varying the heat exchanger geometry on the melting.

3.3.3. Effect of operating conditions

Due to the effect of natural convection, the orientation of the heat exchanger is important as gravity will change the flow properties during melting. This was noted by [Seddegh et al. \(2016\)](#) who compared a horizontal and a vertical shell and tube heat exchanger. This showed that the horizontal heat exchanger was more efficient at the beginning of the melting process while losing in efficiency once the top half of the cylinder was melted. A similar pattern was not observed for the vertical configuration, where the efficiency stayed relatively constant over time. As conduction is the main mode of heat transfer in solidification, the difference in efficiency for the horizontal and vertical heat exchangers was small. [Agyenim & Hewitt \(2010\)](#) studied a horizontal shell and tube heat exchanger for both melting and solidification. This showed that the flow was mostly dominated in the radial direction and that the temperature gradients were negligible (2.5-3.5 % of radial direction), so that a 2D approximation of the shell and tube system was appropriate. However, a drop in the melt fraction over the length of the heat exchanger has been observed in studies by

Promoppatum et al. (2017) and Herbinger et al. (2018), diverging from the findings from Agyenim & Hewitt (2010). This is linked to a drop in the temperature between the inlet and the outlet, so that the heat absorbed from the fluid decreases along the length of the the heat exchanger and the PCM melts less uniformly. A more in depth analysis of this phenomenon is required and this will in part be covered in this thesis. A common solution to the problem of melting uniformity is to use multiple PCM with varying melting temperatures along the length of the heat exchanger, for instance studied in Peiró et al. (2015).

Both the effect of inlet temperature and Reynolds number of the fluid on the PCM have been extensively studied in the literature. For instance, Hosseini et al. (2012) determine that an increase in the inlet temperature leads to an increase in the melting and solidification rate. This is linked to the driving force of heat, where a greater temperature difference between the fluid and the PCM will lead to a higher heat transfer rate. An improvement in the energy absorption rate was also observed when the fluid flow rate is increased, although the effect becomes negligible past a certain value (Cao & Faghri (1991), Shahsavar et al. (2021)). The flow rate did not seem to affect the solidification rate significantly in Esapour et al. (2016) and Liu et al. (2017), indicating that the improvement in the heat transfer coefficient is more noticeable in natural convection driven systems. Liu et al. (2011) studies the effect of fluid mass flow rate for a plate heat exchanger and find that increasing the mass flow rate significantly shortens the melting time. The improvement in the melting is linked to a more uniform melting of the PCM, suggesting that there is an optimal operating flow rate for each design at a given operating temperature at which point no further improvement of the melting of the PCM can be achieved.

The direction of the flow of the fluid can have a significant impact on the melting rate of the PCM for triplex- and multitube configurations. This is studied in Belusko et al. (2016), where it was found that the counterflow configurations achieved higher heat transfer rates than parallel flow configurations.

3.3.4. Performance improvement of LHTES Units

One of the main shortcoming of PCM is the low thermal conductivity of the material, which decreases the rate at which energy can be absorbed and released by the PCM. Counteracting this low thermal conductivity has been a major research focus for the development of the LHTES units. The research has mainly been focusing on increasing the contact surface area between the fluid and the PCM, or by decreasing

the average diffusion distance within the PCM, so that a more efficient heat transfer can be achieved. For this, multiple strategies have been described in the literature, which have been summarized in Figure 3.9.

3.3.4.1 Reduction in diffusion length

Diffusion length in PCM refers to the characteristic distance over which heat or mass diffuses into the material during the phase change process. Due to the low thermal conductivity of the material, the diffusion length has an important impact on the melting and solidification rate of the PCM.

The most common strategy for improving the performance of LHTES is to reduce the diffusive length of the PCM. For this, extended surfaces, heat pipes and the encapsulation of the PCM within metal capsules can be used. The use of metal fins within the PCM has been one of the most widely studied LHTES improvement strategies in the literature, due to the ease of manufacture, installation and maintenance. The fins help in achieving a more uniform penetration of the heat. As such various geometries of fins have been developed, which are summarized in Figure 3.10 for shell and tube heat exchangers and Figure 3.11 for plate heat exchangers.

[Agyenim et al. \(2009\)](#) investigates the use of fins within a horizontal shell and tube heat exchanger. They find that the addition of longitudinal fins helps in reducing the melting time when compared to an unfinned shell. [Deng et al. \(2019\)](#) study the effect of longitudinal fins in a horizontal shell and tube heat exchanger, by varying the number and orientation of fins. They observe that an increase in the number of fins leads to a increase in the melting rate, where the melting time reduces by 70.5% for $N = 10$ when compared to the case without fins. They find that using angled fins is recommended for $N > 6$, while straight fins are better for a lower number of fins. In [Pu et al. \(2020\)](#), PCM is melted in a vertical shell and tube heat exchanger with circular fins of various lengths and placement. This study finds that the addition of fins helps in reducing the melting time and identify an optimal arrangement and length of fins when the mass of PCM is kept constant. They find that distributing the fins more densely at the bottom of the shell increases the melting rate. Certain types of fins are more efficient depending on the orientation of the heat exchanger. As such, longitudinal and tree-shaped fins are more appropriate for horizontal heat exchanger, while circular fins are more effective in vertical arrangements. This is due to the fins separating the shell into different sections so that multiple pools of melted PCM are enabled inside the shell (see [Dekhil et al. \(2021\)](#)). In recent studies

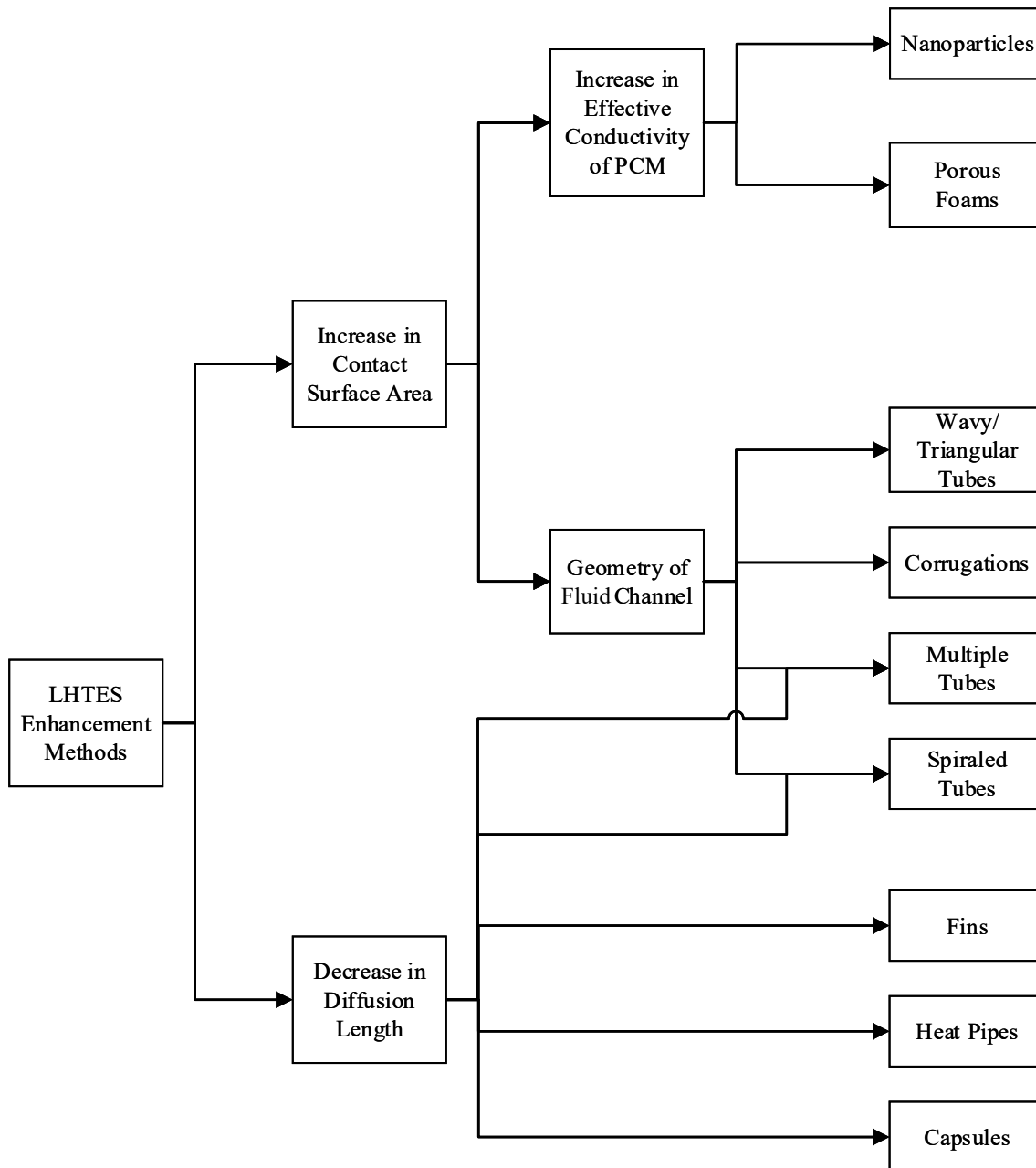


Figure 3.9: Classification of heat transfer enhancement techniques for LHTES

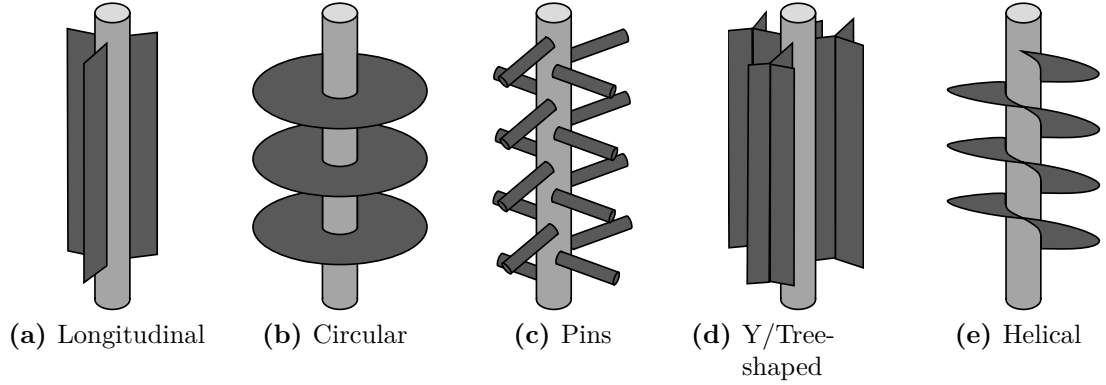


Figure 3.10: Popular types of fin geometries used in PCM-Shell and Tube Heat Exchanger

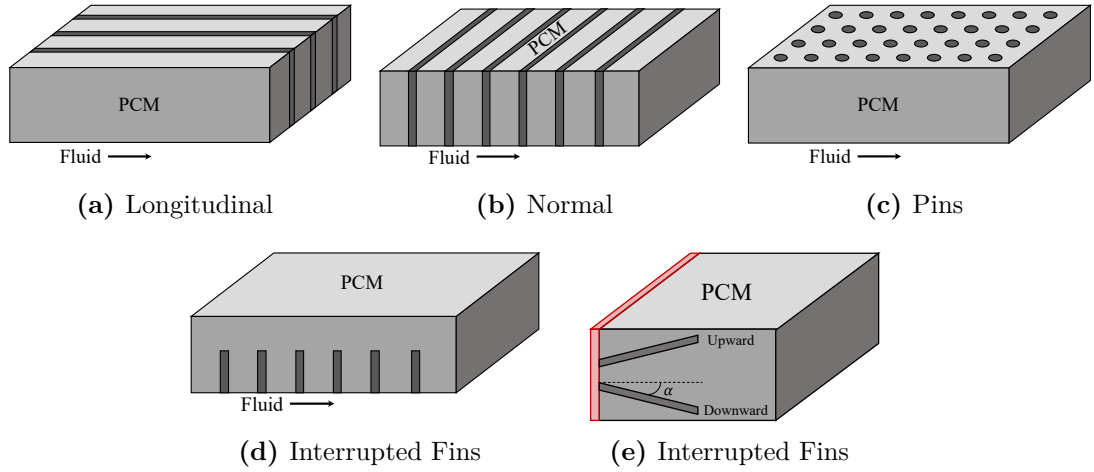


Figure 3.11: Popular types of fin geometries used in plate heat exchangers

by Sciacovelli et al. (2015) and Pizzolato et al. (2017), topology optimization was used to find the optimal fin geometry to certain applications, resulting in tree-shape like fin geometries. Helical fins have been studied by Zonouzi & Dadvar (2022) who found that for a vertical orientation, the helical fins increased the energy absorption rate of the unit due to the helicity of the melted PCM. Beck et al. (2015) found that longitudinal fins were more adapted than helical fins when the shell and tube system is in a horizontal orientation, as the longitudinal geometry leads to stronger buoyancy currents as the flow of PCM is less restricted.

The addition of fins to rectangular units has been widely studied in the literature. Fins were found to increase the melting rate of the PCM, linked to the increased heat transfer area. Gharebaghi & Sezai (2007) conducted a parametric study on a horizontal plate heat exchanger with fins in the normal direction of the fluid where

the geometry of the finned cavity was varied. They found that increasing the fin density led to a decrease in the melting time, making these geometries applicable to cases where large capacity storage units have to be designed around small temperature differences between the fluid and PCM temperature. [Lacroix & Benmadda \(1998\)](#) conduct a study on a rectangular enclosure where interrupted fins are attached to the heated wall (see Figure 3.11.d). The authors compare heating the top and bottom walls and they find that fins attached to the bottom finned wall lead to enhanced melting when compared to the top heated wall, due to an increase in natural convection. Additionally, an optimal distance between fins is observed for which the melting time is minimized. [Lacroix & Benmadda \(1997\)](#) study the melting and solidification of PCM in a finned vertical enclosure, where the number and length of horizontal fins is varied. They find that longer fins significantly enhance the melting, even though they reduce the development of larger thermal plumes when compared to shorter fins. For all studied cases, the increased number of fins was found to increase the melting and solidification rate. The effect of fin angle and length was studied in [Ji et al. \(2018\)](#) for a rectangular vertical enclosure (see Figure 3.11.e). They find that a small downward angle of $\alpha = -15^\circ$ can lead to an enhancement of the PCM melting of 23.8% when compared to the $\alpha = 0^\circ$ case, as this decreased the non-uniform distribution of heat underneath the fin, while an upwards angle increases the melting time. Similarly to previous studies, increasing the fin length increased the melting rate.

PCM can be encapsulated by covering the PCM with a suitable coating or shell materials. This helps in separating the material from its surrounding, such that the reaction of the PCM with its surrounding can be avoided ([Salunkhe & Shembekar \(2012\)](#)). The size of the PCM encapsulation can range between macro (> 1 mm), micro ($0 - 1000 \mu\text{m}$) and nano ($0 - 1000$ nm) scale, where a smaller diameter leads to more rapid melting. As the capsules are entirely surrounded by fluid, this leads to the emergence of unconstrained melting which has been shown to increase the melting rate of encapsulated PCM by [Hou et al. \(2021\)](#). The capsules can be set within a packed bed heat exchanger as shown in Figure 3.12, allowing for a great contact surface area between the fluid and the PCM capsules ([de Gracia & Cabeza \(2017\)](#)). Alternatively, a slurry can be created by mixing micro- or nano-capsules with the heat transfer fluid with further details described in the review by [Salunkhe & Shembekar \(2012\)](#).

[Motahar & Khodabandeh \(2016\)](#) studies the use of heat pipes to enhance the melting and solidification of PCM in a vertical cylindrical test cell, while [Ladekar et al. \(2017\)](#)

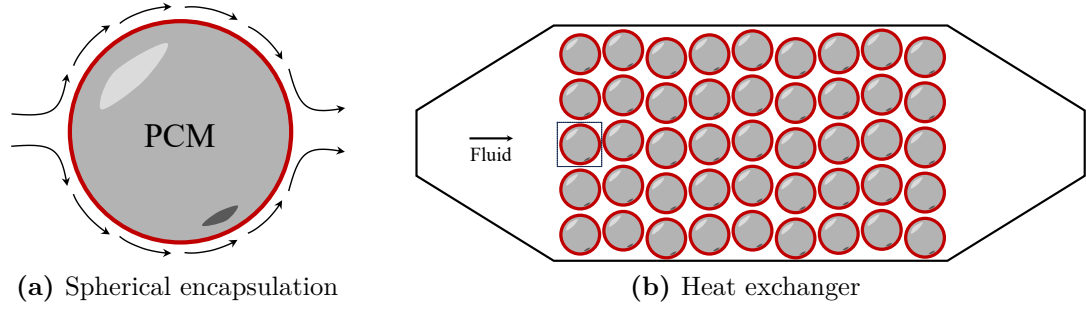


Figure 3.12: Use of encapsulated PCM within a packed bed heat exchanger

use heat pipes within rectangular geometry. Both authors found a reduction in the melting time when heat pipes were used.

3.3.4.2 Increase in effective conductivity of PCM

Some improvement strategies have been focusing on the increase of the effective thermal conductivity of the PCM. This is achieved by adding a material with high thermal conductivity to the PCM by embedding the PCM in a porous foam or by mixing nanoparticles into the PCM. These additives are often made of metals, such as copper, aluminium and nickel, or carbon based materials, such as expanded graphite, graphene or carbon fibres (Hamidi et al. (2023)).

Mesalhy et al. (2005) studied the effect of matrix porosity (β) on the melting of PCM by varying the porosity between 85% and 95%. They note that while decreasing the porosity leads to an increased melting rate, the effect of natural convection also diminishes. As the porosity is decreased, this leads to a smaller increase in the melting rate. Xu et al. (2017) studied the effect of placement of porous media within a shell and tube heat exchanger. By only placing the porous medium in the bottom half of the cylinder (see Figure 3.13.a), where the PCM melts more slowly due to an absence of convective currents, the melting rate can be improved drastically. The authors recommend a filling height of 70% of the shell to maximize the storage capacity of the unit. Mahdi et al. (2022) investigated the thermal performance of a shell and tube heat exchanger where metal foam strips and fins were added to the PCM, as shown in Figure 3.13.b. The results show a maximum of 58% improvement in the melting rate when a mixture of fins and foam strip were used. Qureshi et al. (2021) compared the melting of PCM in various triply periodic minimal surfaces (TPMS) and a Kelvin-cell based geometry with $\beta = 90\%$. The results identify an improvement in the melting rate for the TPMS surfaces when compared to the Kelvin-cell surface, with an IWP geometry achieving the most rapid melting.

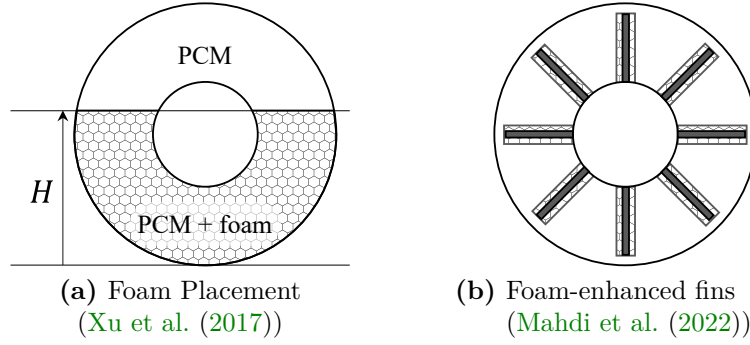


Figure 3.13: Increasing the effective thermal conductivity by using porous medium based on examples from the literature.

The addition of nanoparticles for heat transfer applications has been growing in interest in the recent years, including for use in LHTES units. [Elbahjaoui & El Qarnia \(2017\)](#) study the thermal enhancement of nanoparticles within a plate heat exchanger during the melting process. The addition of nanoparticles was found to decrease the melting time and improves the sensible heat storage. [Abdulateef et al. \(2021\)](#) added nanoparticles to a triplex tube heat exchanger with triangular or longitudinal fins. A 10% volume fraction of Al_2O_3 nanoparticles improved the melting time by 12% for the longitudinal fins and 22% for the triangular fins.

3.3.4.3 Increasing the contact surface area of fluid channel

Increasing the contact surface area by varying the geometry of the fluid channel can have an effect on the melting of the PCM. [Santos et al. \(2021\)](#) studied the effect of different ridge geometries in the channels of a plate heat exchanger on the melting of PCM. This found that small dimples could achieve double the heat transfer rate for a small increase in pressure drop when compared to a smooth plate. An improvement in the heat transfer can be achieved by disturbing the flow of the fluid through corrugations as described in [Mahani et al. \(2020\)](#), where the addition of triangular corrugations led to a higher solidification and melting rate. [Talebizadehsardari et al. \(2021\)](#) looked at the case of triangular zigzag plates for LHTES units. The results showed that the wave amplitude played a key role in increasing the heat transfer rate, mostly linked to the increased contact surface area with the PCM.

3.3.4.4 Hybrid strategies

Hybrid strategies that combine increasing the surface area and decreasing the diffusion length can be found in the literature. As previously described, multiple tubes

can help in increasing the efficiency of LHTES systems in bigger units. [Ardahaie et al. \(2019\)](#) achieves this by using a flat spiral tube heat exchanger, where a significant increase in the surface area is achieved without increasing the volume of the LHTES unit. The authors show that increasing the surface area and decreasing the diffusion length by varying the number of flat spiral coils in the heat exchanger can improve the performance of the heat exchanger, especially at the early stage of melting. Various designs of triplex shell and tube designs are analysed in the thesis by [Al-mudhifar \(2019\)](#), that are combined with various fin geometries to improve the melting and solidification rate of the heat exchanger. [Salunkhe & Shembekar \(2012\)](#) provides a couple of example studies where fins have been added to capsules.

3.4. Predicting the performance of PCM heat exchanger

Problems involving phase change have attracted a large amount of interest due to their wide applications in engineering, mathematics and physics. Different computational strategies for modelling phase change problems are discussed in this section.

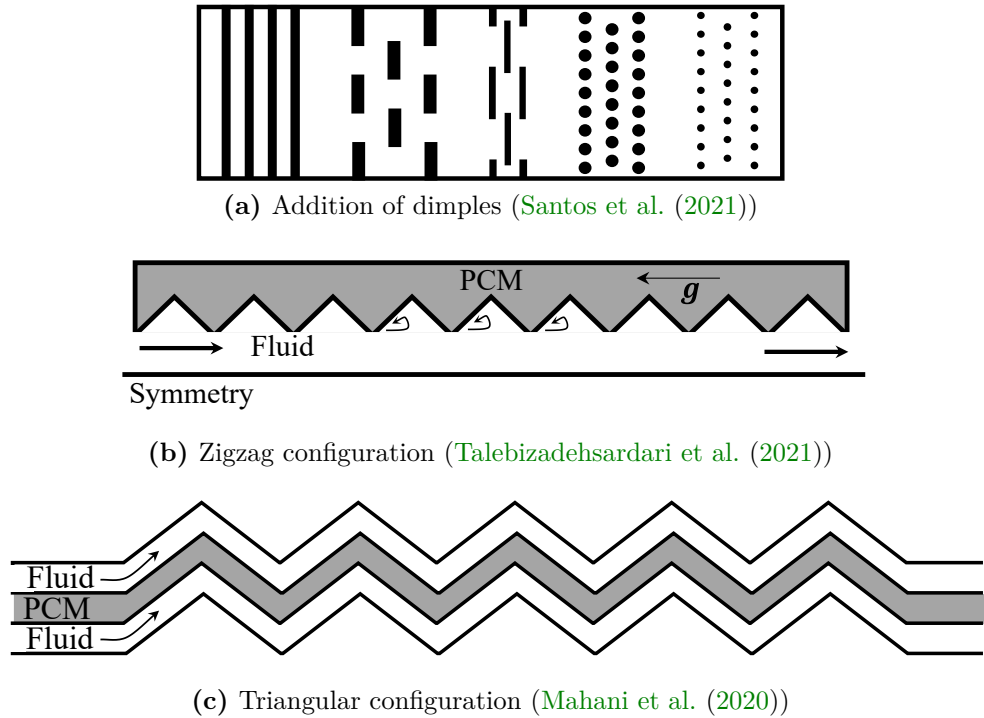


Figure 3.14: Improving the melting by changing the contact surface angle based on examples from the literature.

3.4.1. Analytical models

3.4.1.1 Stefan condition

The Stefan Problem aims at describing the melting or solidification of a pure material mathematically, via a moving boundary separating the two phases as described in Figure 3.15. [Stefan & Akad \(1889\)](#) first formulated a model to capture this moving boundary for the solidification of water and melting of ice. The equation for pure conduction in the liquid phase Ω_l and solid phase Ω_s is given by:

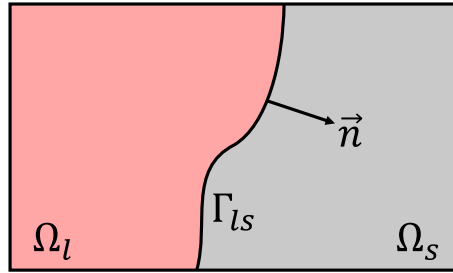


Figure 3.15: Solid-Liquid phase change process schematic

$$\rho_s c_{ps} \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_j} \left(k_s \frac{\partial T}{\partial x_j} \right) \quad \text{in } \Omega_s, \quad (3.2a)$$

$$\rho_l c_{pl} \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_j} \left(k_l \frac{\partial T}{\partial x_j} \right) \quad \text{in } \Omega_l. \quad (3.2b)$$

Following this, two boundary conditions are defined at the solid/liquid interface Γ_{ls} . The first condition identifies the melting front based on the melting temperature:

$$T = T_m \quad \text{on } \Gamma_{ls}, \quad (3.3)$$

The second condition balances the heat at the moving interface and is defined as:

$$k_s \frac{\partial T}{\partial x_j} - k_l \frac{\partial T}{\partial x_j} = \rho_s L \nu_{\Gamma_{ls}} \quad \text{on } \Gamma_{ls}, \quad (3.4)$$

where T is the temperature, ρ is the density, c_p is the specific heat and k is the thermal conductivity, n_i is the normal of the interface, $\nu_{\Gamma_{ls}}$ is the propagation velocity of the interface and L is the latent heat per unit mass of the material.

[Stefan & Akad \(1889\)](#) used this formulation to find the analytical solution for a semi-infinite 1D problem with constant thermo-physical properties. In this problem, the rate of melting is governed by the Stefan number St , defined as the ratio between sensible and latent heat:

$$St = \frac{c_p(T - T_m)}{L}. \quad (3.5)$$

A limited amount of analytical solutions to the Stefan problem are available, but these are limited to relatively simple geometries and boundary conditions, where the boundary is semi-infinite (see [Hu & Argyropoulos \(1996\)](#) for a summary of analytical solutions in rectangular and cylindrical geometries). For more complex problems, as found in most engineering applications, it is necessary to solve the Stefan problem numerically. These are subdivided into front tracking methods, where the mesh is adapted to fit the two phases, and fixed grid methods, where the spatial grid is fixed and a new variable is introduced to track the phase change interface. As the solid-liquid interface becomes less distinct, it is harder to explicitly track the phase change front. Due to the higher computational cost and lower stability associated with front tracking methods, fixed grid methods were selected for this work.

3.4.1.2 Effectiveness-NTU model

A range of analytical and semi-analytical models have been developed to study the effect of a wide range of geometry changes on the effectiveness of the PCM heat exchanger. For instance, the effectiveness-NTU ($\epsilon - NTU$) model has been applied to shell and tube heat exchangers ([Bruno et al. \(2010\)](#), [Tay et al. \(2012\)](#), [Tay et al. \(2014\)](#), and [Liang et al. \(2020\)](#), [Tian et al. \(2021\)](#)), plate heat exchangers ([Belusko et al. \(2012\)](#)) and packed bed heat exchangers ([Amin et al. \(2014\)](#), [Aziz et al. \(2022\)](#) and [Bejarano et al. \(2018\)](#)). In these studies, a good agreement with experimental or numerical results obtained from higher fidelity simulation was observed, while the models showed a significant reduction in the computational cost. As a result, the model was applied for parametric studies in [Tian et al. \(2021\)](#) and design optimization in [Liang et al. \(2020\)](#).

In the $\epsilon - NTU$ models found in the literature, the position of the melt front is used to define the location of a perfect heat sink. This is then used as part of a thermal network to determine the temperature drop within the heat exchanger. In the case of a heat exchanger with PCM, finding an analytical solution of the solid/liquid interface position over time is complicated, due to the change in boundary conditions over the length of the heat exchanger and the time of melting. [Bruno et al. \(2010\)](#), [Tay et al. \(2012\)](#) and [Amin et al. \(2014\)](#) overcome this issue by defining the position of the melt front in terms of a time independent phase change fraction parameter. This significantly simplifies the problem, as the effectiveness can be calculated at different melt ratios of the PCM domain. This is sufficient to determine the average

effectiveness of a heat exchanger at different mass flow rates (see Tay et al. (2012)) and can predict the performance of more complex geometries, such as when fins are added to the shell as studied in Tay et al. (2014). The main shortcoming of the phase fraction methodology is that the melting time of the PCM is not taken into account. Two different geometries compared by using this simplification can have a significantly different melting time, due to the highly nonlinear melting of the PCM. Therefore, additional experimental or simulation work has to be completed to ensure that the final design can achieve the energy density required for the application in a given melting/solidification cycle.

This problem is addressed by Tian et al. (2021), where a dynamic model based on energy conservation is proposed. In the model, the energy released by the working fluid is directly absorbed by the PCM in the form of latent heat by using a Stefan condition. As a result, an ordinary differential equation has to be solved numerically, leading to a slightly higher computational cost than the previous model. However, more complicated time dependent operating conditions can also be modelled. To account for the buoyancy that occurs in the PCM during the melting cycle, the authors use an empirical model based on the effective thermal conductivity of the PCM. Bejarano et al. (2018) develops an $\epsilon - NTU$ model that is capable of studying partial charging and discharging cycles accurately within spherical capsules filled with PCM. This is achieved by subdividing the PCM into spherical sublayers.

An $\epsilon - NTU$ model will be used to develop a sizing tool for PCM plate-fin heat exchangers in Chapter 7. A full description of the model will be given in that chapter.

3.4.2. CFD models

3.4.2.1 Summary of fixed grid methods

A summary of different fixed grid methods will be provided in this section. In fixed grid methods, both the solid and liquid phases are modelled in the same computational domain, so that the phase change problem can be solved by a single set of equations. This is achieved by defining a phase fraction ϕ , where the velocity and the temperature are related to the properties of each phase in each cell. This approach leads to a significant decrease in complexity of the problem for cases where the solid/liquid interface becomes less distinct Voller et al. (1990), so that fixed grid methods are typically associated with lower cost.

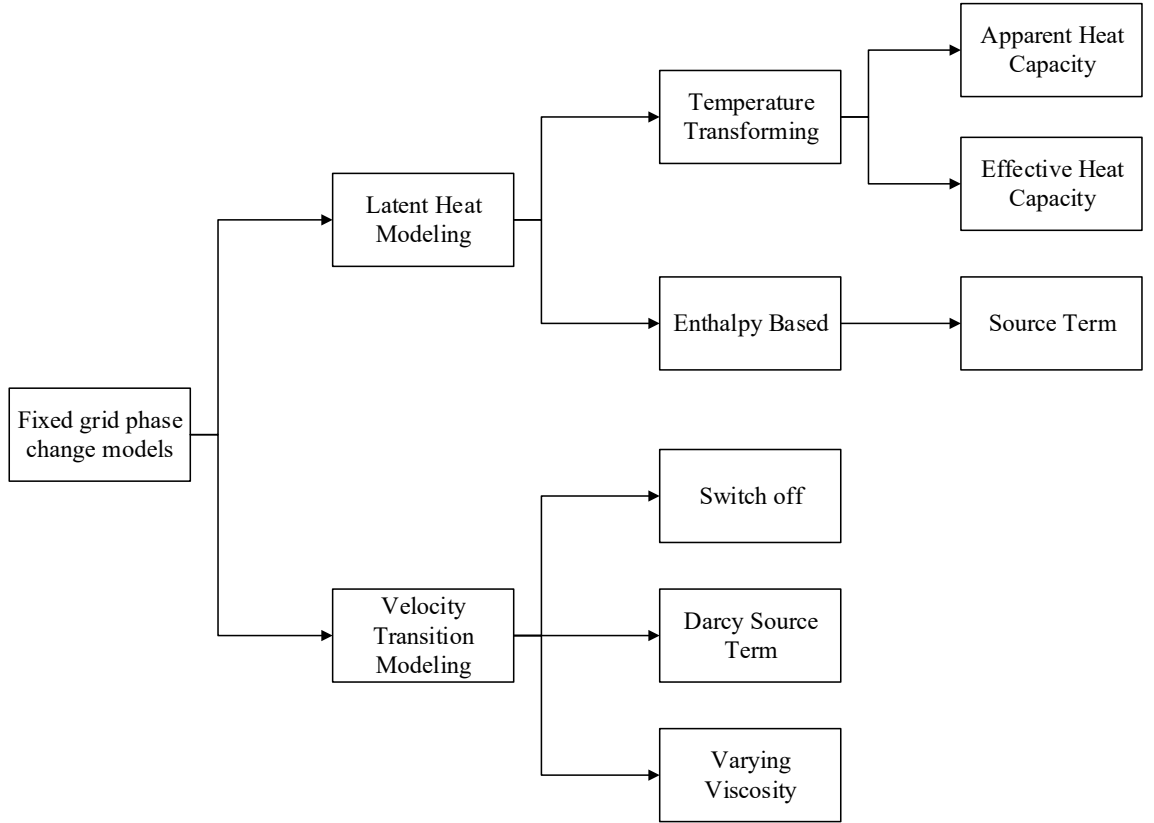


Figure 3.16: Summary of fixed grid phase change models.

To appropriately model the phase change in fixed grid methods, a relationship for the velocity needs to be defined to account for the velocity transition during melting and solidification. Similarly, the release or absorption of latent heat on the temperature field needs to be taken into account. Different techniques have been developed to take these two properties into account and this has been summarized in Figure 3.16.

3.4.2.2 Latent heat

Temperature Transforming Methods

In temperature transforming models, the latent heat formulation is incorporated into the specific heat capacity (c_p) term of the energy equation provided in Equation 3.6.

$$\rho c_p \left(\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right), \quad (3.6)$$

In the apparent heat capacity method, the phase change is modelled by increasing the specific heat of the PCM during the phase change. The specific heat is referred to as the apparent heat capacity c_a which is defined based on the total enthalpy (H) in the system:

$$c_a = \frac{dH}{dT}. \quad (3.7)$$

It is worth noting that equation 3.7 needs to be differentiable, which suggests that $H(T)$ has to be continuous. This is not necessarily the case for phase change materials, where the total enthalpy might contain jump discontinuities, e.g. for isothermal phase change. To overcome this issue, a small artificial temperature range at the jump can be introduced to ensure that the function is piece-wise continuous throughout the temperature domain as described in [Voller et al. \(1990\)](#). Considering a constant specific heat $c_{p,i}$ for each phase as shown in Figure 3.17, the total enthalpy H for a phase change temperature difference of $2\epsilon_T$ can be defined, so that the apparent specific heat increases during the phase change:

$$H(T) = \begin{cases} c_{p,s}(T - T_{\text{ref}}) & T \leq T_m - \epsilon_T, \\ \int_{T_s}^{T_l} c_p dT + \phi(T)L & T_m - \epsilon_T < T \leq T_m + \epsilon_T, \\ c_{p,l}(T - T_{\text{ref}}) & T > T_m + \epsilon_T. \end{cases} \quad (3.8)$$

During the phase change, both the specific heat and latent heat addition is commonly simplified to a constant, giving the following apparent heat capacity definition:

$$c_a(T) = \begin{cases} c_{p,s} & T \leq T_m - \epsilon_T, \\ 0.5(c_{p,s} + c_{p,l}) + \frac{1}{2\epsilon_T}L & T_m - \epsilon_T < T \leq T_m + \epsilon_T, \\ c_{p,l} & T > T_m + \epsilon_T. \end{cases} \quad (3.9)$$

For small phase change intervals, the apparent heat capacity method can lead to problems in the convergence of the model as reported in [Ma & Zhang \(2006\)](#). Furthermore, small time steps are needed for predicting the phase change accurately. This is attributed to the fact that latent heat is not accounted if the temperature change in a control volume is too abrupt when the phase change occurs. To solve this issue, [Poirier & Salcudean \(1988\)](#) propose to spatially average the apparent heat capacity in an effective heat capacity:

$$c_{\text{eff}} = \frac{\int c_a dV}{V}. \quad (3.10)$$

This ensures that the latent heat is taken into account at each time step, allowing for larger time step sizes to be used.

Enthalpy Based Methods

In the enthalpy based method, the energy equation is rewritten in terms of the total

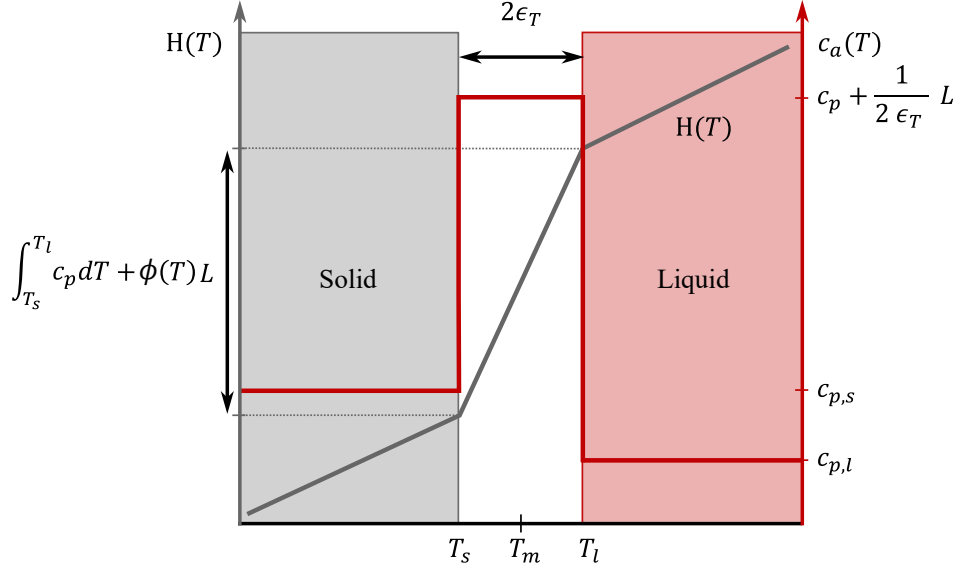


Figure 3.17: Apparent Heat Capacity formulation $c_a(T)$ (red curve) for a piece-wise relaxation of the enthalpy relation $H(T)$ (grey curve).

enthalpy H , as:

$$\frac{\partial \rho H}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j H) = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right), \quad (3.11)$$

where the total enthalpy H is the addition of the sensible and latent part. Following Voller et al. (1990), this can be modelled as:

$$H = (1 - \phi) \int_{T_{\text{ref}}}^T \rho c_{p,s} dT + \phi \int_{T_{\text{ref}}}^T \rho c_{p,l} dT + \phi \rho L, \quad (3.12)$$

where the latent heat component is linearly interpolated based on the temperature. In this form, the formulation of Equation 3.11 is valid across all ranges of the phase change, including in cases where discontinuities are present, e.g. in isothermal phase change. A common approach for this method is to group the additional latent heat terms into a source term:

$$S = \frac{\partial \rho \Delta H}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j \Delta H) \quad (3.13)$$

Voller & Prakash (1987) note that enthalpy based methods can lead to the oscillation in the temperature solution over time for a small range of the mushy zone (2ϵ). This happens when the transient term of the latent heat source term is determined by using an iterative update. To overcome this, an appropriate treatment of the source term is described in Voller et al. (1990).

3.4.2.3 Velocity transition

To accurately model the phase change process, it is important to take the change in velocity between the flow in the liquid phase and the static behaviour of the solid into account. This is achieved by adjusting the momentum equations, which in its basic form is:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_j} + \mu \frac{\partial^2 u_i}{\partial x_j^2} + \rho \vec{g}. \quad (3.14)$$

Voller et al. (1987) identifies 3 main methods for dealing with the drop in velocity during solidification: switch off method, variable viscosity method and porous source term method. These will be summarized here.

Switch off Method

In the switch off method, the velocity is defined in terms of temperature by using a step function, where the velocity is simply set to zero when the material is solid. This can be done by overwriting the velocity value in control volumes in which the temperature is above the melting temperature, or by using the predicted values of latent heat contribution to the enthalpy Voller et al. (1987). Switch off methods have been identified to be relatively easy to implement, but the method can lead to an discontinuous velocity field as described in Nazzi Ehms et al. (2019).

Variable Viscosity Method

In the variable viscosity method initially proposed by Gartling (1978), the viscosity of the mixture is increased to a large value as the material solidifies. As this happens, the velocity of the material is reduced. In Gartling (1978), this is done by using a temperature-dependent viscosity where an arbitrary large value for the viscosity in the solid regime is chosen. Voller et al. (1987) proposes an alternative method, where the variable viscosity is dependent on the liquid fraction:

$$\mu = \mu_l + B(1 - \phi), \quad (3.15)$$

where μ is the effective viscosity, μ_l is the liquid viscosity and B is the large value used to emulate the solid phase. Voller et al. (1987) argues that this formulation is more convenient for isothermal phase change problems. Further formulations for the viscosity have been defined to get a more physical representation of the phase change. These have been summarized in Nazzi Ehms et al. (2019) review paper on fixed grid numerical models for phase change.

Porous Source Term Method

A porous source term can be incorporated into the momentum equation, where the source term is responsible for slowly increasing the velocity as the material melts. This is achieved in [Voller & Prakash \(1987\)](#) by approximating the mushy region as a porous medium. Here the following source term, inspired by the Kozeny-Carman equation, is used:

$$S_A = -A_{mush} \frac{(1 - \phi)^2}{\phi^3 + q} u_i, \quad (3.16)$$

where A_{mush} is the mushy zone constant, which depends on the morphology of the porous medium and q is a constant value to avoid division by zero when the material is solid. This is usually set to a small number, with most codes using a value of 0.001.

[Nazzi Ehms et al. \(2019\)](#) argues that the smooth transition of the source term method leads to a more stable solution when compared to the switch off method and the method is less prone to numerical problems when compared to the variable viscosity method according to [Voller & Prakash \(1987\)](#). As a result, the method is widely popular for treating the changing velocity field in the mushy region.

Mushy Zone Constant

The choice of mushy zone constant plays a vital role on predicting the melting rate of the PCM, as studied by multiple studies. [Hameter & Walter \(2016\)](#) find that a lower value of the mushy zone constant leads to a faster melting rate, due to the higher value of the source term as the material melts. Based on the Ansys user guide, a mushy zone constant varying between 10^3 and 10^9 is advised, with no instructions on how to choose this quantity. In their paper on microstructural evolution of the mushy region for melting and solidification, [Yang et al. \(2019\)](#) provide guidance on how to choose an appropriate value of the mushy region:

$$A_{mush} = \frac{180\mu}{d^2}, \quad (3.17)$$

where d corresponds to the dendrite length. The authors observe that this characteristic length varies based on the local liquid fraction in their experiments and advise to calculate this as:

$$d = C_1 \phi_l^{C_2} [\mu m], \quad (3.18)$$

where C_1 and C_2 are coefficients that should be determined from experimental results, and ϕ_l is the local liquid fraction. As the microstructure varies between melting and solidification, the coefficients are different for both cases.

Koyuncu (2023) investigates the effect of mushy zone constant on the melting of PCM by simulating the same case for different values of A_{mush} . This found a great variation in the average temperature of the PCM over time, where $10^7 \text{ kg m}^{-3} \text{ s}^{-1}$ is the only value showing good agreement with experimental results. Following this, the dendrite length is measured for different types of PCM by calculating an average value from different measurements and this is used to calculate the mushy zone constant based on Equation 3.17. The values for the mushy zone constant that are recovered from this calculation ($A_{mush} = 2.01 \times 10^7 \text{ kg m}^{-3} \text{ s}^{-1}$) are in good agreement with the mushy zone constant that was determined when comparing simulation results with experiments. Furthermore, the author studies the micro structure of different organic and inorganic PCM. This finds that the different materials exhibit different structures, where crystals and filaments are observed in the mushy zone.

3.4.2.4 Modelling of unconstrained melting

Unconstrained melting requires more complex modelling assumptions than constrained melting, where the use of the Boussinesq approximation is found to be sufficient to model the convection in the PCM. Nemati et al. (2023) reviews different modelling strategies for unconstrained melting based on the enthalpy-porosity model and provides a modelled example of unconstrained melting where the different strategies are applied. The authors note that for unconstrained melting, the main driving force for buoyancy is the density difference between the liquid PCM and the solid PCM and not the changes in density within the liquid pool. As such, the Boussinesq approximation, which only accounts for the density changes in the melted PCM is insufficient for modelling the melting of PCM in these cases, such that the melting rate is substantially underestimated. This is confirmed in the simulations carried out by the authors. Additional modelling strategies are considered, based on single phase and multiphase volume of fluid (VOF) simulations. A common method used in the literature is to define the material to have a different solid and liquid density in a single phase model. This simplifies the problem significantly, where the layer of air that is usually present in the cavity to allow for the thermal expansion of the PCM is not modelled. However, this violates mass conservation and leads to uncertainties in the solution. The VOF models model the PCM and the gas layer as two different phases, where the expansion of the PCM is accounted,

leading to an increased computational cost.

The authors suggest a further method, where a body force source term is added to the momentum equation to account for the change in density, maintaining mass conservation:

$$\vec{F} = \vec{g} \begin{cases} \min \left(\rho_s, \frac{\rho_s - \rho_l}{T_s - T_l} (T - T_l) + \rho_l \right) & T < T_l \\ \frac{\rho_l}{1 + \beta(T - T_l)} & T \geq T_l \end{cases} \quad (3.19)$$

This provides similar results in predicting the position of the melt front to the VOF model once the layer of air required for the VOF simulation is accounted for, at a fraction of the computational cost. However, all the methods in the study share a similar problem, where a large film thickness is observed, which is not present in the experimental results. Additionally, the shape of the melt front is not well reconstructed when compared to the experimental results.

The difference in experimental and numerical results in [Nemati et al. \(2023\)](#) could be linked to the Darcy Source term approach used to model the mushy region, where the velocity is suppressed by introducing a source term. This means that the sinking velocity of the solid region is also suppressed in the mushy region. [Faden et al. \(2018\)](#) addresses this issue, based on previous work by [Rösler \(2014\)](#). In their paper, the authors prescribe a solid settling velocity $\vec{u}_{set}$ within the Darcy Source term, which accounts for the falling solid PCM.

$$\vec{S}_A = \frac{A_{mush}(1 - \phi)^2}{\phi^3 + \epsilon} (\vec{u} - \vec{u}_{set}). \quad (3.20)$$

The value of the settling velocity is determined based on a force balance of the solid PCM acting on the liquid PCM and the enclosure, requiring the explicit knowledge of the interface position. An immersed boundary method to reconstruct the surface of the PCM. Following this method, [Faden et al. \(2018\)](#) accurately predict the settling velocity as well as the position of the solid PCM within a square cavity (see Figure 3.18). The method requires additional steps at each time step, leading to a non negligible increase in the computational cost. This limits the use of the model to small geometries, such that this thesis will focus on constrained melting of PCM.

3.4.3. Enthalpy-porosity model

Due to its wide availability in commercial and open-source software, computational stability and accuracy, the enthalpy-porosity model was selected to model the melt-

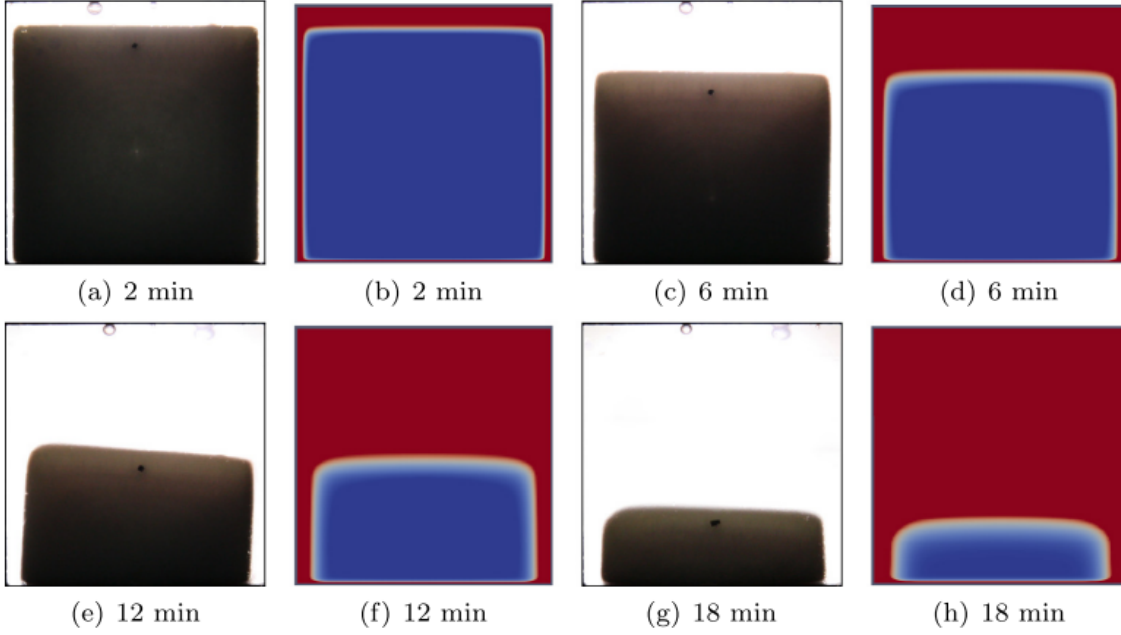


Figure 3.18: Comparison of experimental and numerical result for the melting front over time for a square cavity. Reproduced from [Faden et al. \(2018\)](#).

ing of PCM in this work. The enthalpy-porosity model was firstly created by [Voller & Prakash \(1987\)](#) for applications in the metallurgical industry to accurately predict the moving boundary phase change problem. The model is developed from the enthalpy method, where the temperature is defined as a function of the enthalpy. In the enthalpy model by [Gartling \(1978\)](#), the velocity reduction as the fluid turns to solid is modelled by significantly increasing the viscosity of the cell. [Voller & Prakash \(1987\)](#) consider modeling the fluid that undergoes a phase change as a pseudo-porous media by using a Darcy source approach to simulate the convective motion of the fluid.

For a laminar flow, the heat equation for transient conjugate heat transfer is given in terms of enthalpy by:

$$\frac{\partial \rho H}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j H) = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) \quad (3.21)$$

where k is the thermal conductivity of the fluid, ρ is the density and H is the total volumetric enthalpy defined as:

$$H = h + \Delta H$$

$$h = h_{ref} + \int_{T_{ref}}^T c_p dT$$

where h is the sensible heat defined and c_p is the specific heat. ΔH is the latent heat contribution, which in the current model is a function of the liquid fraction.

By substituting the relationship for H in equation 3.21, the following convection-diffusion phase change equation is given:

$$\frac{\partial \rho h}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j h) = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) - \frac{\partial \rho \Delta H}{\partial t} - \frac{\partial}{\partial x_j}(\rho u_j \Delta H) \quad (3.22)$$

where the ΔH terms are collected into a source term so that:

$$\frac{\partial \rho h}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j h) = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + S_h \quad (3.23)$$

ΔH is defined in terms of the liquid fraction ϕ and the heat storage capacity L , where the $\phi = 0$ when the material is solid, $\phi = 1$ when the material is liquid and $0 < \phi < 1$ when the material is melting/solidifying.

$$\Delta H = f(T) = \begin{cases} L & T \geq T_l \\ \phi L & T_l > T \geq T_s \\ 0 & T < T_s \end{cases} \quad (3.24)$$

with T_s , the solidus temperature (material is fully solid) and T_l , the liquidus temperature (material is fully liquid).

The liquid fraction is interpolated at each iteration based on the instantaneous temperature T , T_s and T_l .

$$\phi(T) = \begin{cases} 1 & T \geq T_l \\ \frac{T-T_s}{T_l-T_s} & T_l > T \geq T_s \\ 0 & T < T_s \end{cases} \quad (3.25)$$

The momentum and continuity equations for incompressible, unsteady laminar flow are respectively given by:

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} \right) = -\frac{\partial P}{\partial x_j} + \mu \frac{\partial^2 u_i}{\partial x_j^2} + S_A + S_b \quad (3.26)$$

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3.27)$$

Where ρ is the density, P is the pressure, μ is the liquid viscosity. S_A is a source term which changes the velocity based on the phase of the cell, while S_b models the effect of gravity on the melted region.

The phase change can be divided into three phases during the melting process. At the beginning, the material is fully solid, so that there is no fluid flow. As the material starts melting above the solidus temperature, a porous zone forms, called the mushy zone. When the material is fully melted, the PCM can be considered as a fluid.

In the enthalpy-porosity model, this is accounted for with a source term S_A following a similar approach to the Carman-Koseny equation, which relates the pressure drop to the porosity and the velocity. S_A is given by:

$$S_A = -A_{\text{mush}} \frac{(1 - \phi)^2}{\phi^3 + q} u_i \quad (3.28)$$

where A_{mush} is the mushy zone constant which depends on the morphology of the porous medium (large number, usually ranged between 10^4 and $10^8 \text{ kg m}^{-3} \text{ s}^{-1}$) and describes the rate of velocity change due to melting/solidification. q is a constant which is added to avoid a division by zero and is set to a small number. Ansys FLUENT uses a value of 0.001 (*ANSYS FLUENT Theory Guide 2009*).

The velocity that is computed in the model is an averaged velocity over the area of the cell u_i . In between the solidus and liquidus temperature, the actual velocity of the liquid velocity around the solid PCM will be higher than the averaged velocity in the cell due to the melted material circulating around the solid material, so that the area of liquid PCM is smaller than the cell area (see Figure 3.19).

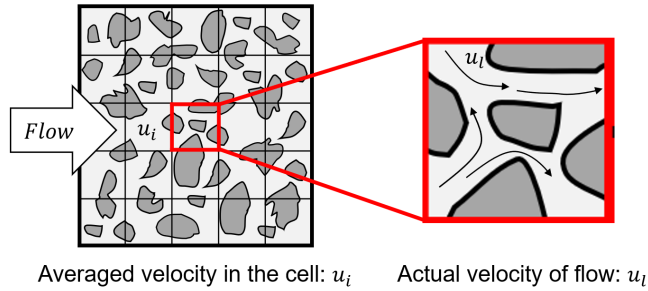


Figure 3.19: Impact of porous medium on the local velocity

It is possible to find the average actual velocity in the cell u_l from the cell averaged velocity from:

$$u_i = \phi u_l \begin{cases} u_l & T \geq T_l \\ \phi u_l & T_l > T \geq T_s \\ 0 & T < T_s \end{cases}$$

The change in density in the liquid region is relatively small, so that the Boussinesq approximation can be used to model the effect of natural convection. The Boussinesq approximation makes it possible to assume that the density is constant in all terms except the body force term, so that the equations are solved by assuming the flow is incompressible. This considerably simplifies the problem.

The source term S_b is used to account for the body force term:

$$S_b = -\frac{\rho_{ref}g\beta(h - h_{ref})}{c_p}y \quad (3.29)$$

Where g is the gravitational acceleration, β is the thermal expansion coefficient and h_{ref} is a reference value of the sensible heat.

3.5. Summary

This chapter summarized the state-of-the-art of PCM research and LHTES unit development. Many PCM are available covering a wide range of material properties, such that a suitable material needs to be carefully chosen based on specific applications. For this thesis, a commercial organic paraffin based PCM was selected due to its compatibility with metallic enclosures and its low propensity for supercooling, which enables better control over heat exchanger performance.

The following section provided a review of LHTES units in the literature. The first part summarized the various applications of PCM, before looking at the effect of geometry and operating conditions of common heat exchanger geometries. Most studies in the literature focus on operating conditions that differ significantly from those in aerospace applications, where minimizing melting time and reducing system mass and volume are critical. Consequently, the geometries explored in the literature, predominantly shell-and-tube and platular heat exchangers, are not directly applicable to aerospace requirements. This thesis focuses on plate-fin heat exchangers that are widely used in aircraft, where the PCM will be encapsulated on one side of the heat exchanger. A poor understanding of the melting uniformity based on fluid flow rate was identified, which will be a focus of the work in this thesis. Due to the low thermal conductivity of PCM, the geometry of the enclosure plays a key role in improving the thermal performance of PCM heat exchangers. Literature strategies to address this include incorporating materials with higher thermal conductivity, modifying fluid channel geometries to maximize surface area, and reducing diffusion length through fins, heat pipes, or smaller PCM cavities. Plate-fin

heat exchangers leverage the latter two strategies, as fins improve the heat transfer coefficient on the fluid side and reduce diffusion length on the PCM side.

The final section addressed modelling approaches for predicting the performance of PCM heat exchanger, by using various analytical and numeric models to qualify the heat transfer and phase change in PCM heat exchanger. Analytical models are often limited to simple phase-change scenarios, such that various numerical models were developed in the last century that are capable of modelling more complex heat transfer and fluid flow within the PCM. The enthalpy-porosity method, which integrates an enthalpy-based latent heat approach with a Darcy source term to stabilize velocity in the mushy region, offers a robust and accurate framework for modelling constrained PCM melting. The model will be validated against experimental results for a shell and tube heat exchanger geometry from the literature in Chapter 4 and will then be used throughout Chapters 5-6 to simulate phase change within PCM plate-fin heat exchangers. While CFD provides accurate results, it is associated with a large computational cost, such that optimizing the geometry or studying a wide range of parameters is complicated. The $\epsilon - NTU$ model provides a practical alternative. Various applications were found in the literature highlighting its ability to predict the performance of various PCM heat exchangers, e.g. shell and tube, platular and packed bed heat exchangers. A formulation for plate-fin heat exchangers is currently missing. This gap will be addressed in Chapter 7, where the model will be applied to determine the size of appropriate PCM plate-fin heat exchangers.

Modelling of a PCM Shell and Tube Heat Exchanger

Contents

4.1	Introduction	77
4.2	Problem Description	77
4.3	Solution Independence Study on 2D Geometry	79
4.3.1	Problem Setup	79
4.3.2	Mesh Independence and Time Step Independence Study .	81
4.3.3	Description of melting process	84
4.3.4	Effect of discretization	85
4.3.5	Effect of material properties definition	86
4.3.6	Effect of software packages	87
4.3.7	Comparison with literature results	89
4.4	3D Conjugate Heat Transfer Simulation	91
4.4.1	Problem Setup	91
4.4.2	Mesh Independence	92
4.4.3	Comparison with literature results	92
4.4.4	Effect of temperature drop over length of heat exchanger	94
4.5	Detailed 3D PCM section analysis	96
4.5.1	Problem Setup	96
4.5.2	Mesh Independence Study	96
4.5.3	Comparison with 2D Simulation	97
4.5.4	Melting Regimes	98
4.5.5	Movement of the phase change front	103
4.6	Summary	106

4.1. Introduction

This chapter assesses the impact of different modelling assumptions in an effort to develop a modelling framework that accurately evaluates the thermal performance of a PCM heat exchanger during the melting cycle. For this, the melting within an unfinned shell and tube heat exchanger geometry is considered based on a popular study from [Hosseini et al. \(2012\)](#), as described in Section 4.2.

Different modelling assumptions and software packages are assessed on a simplified 2D geometry of the problem in Section 4.3, before the results are validated by using a 3D conjugate heat transfer model in Section 4.4. Finally, a detailed study of the melting within a section of the shell is given in Section 4.5.

4.2. Problem Description

A shell and tube heat exchanger geometry without fins proposed by [Hosseini et al. \(2012, 2014\)](#) was chosen for an initial study to better understand the melting behaviour of PCM and validate the use of the enthalpy-porosity model for subsequent analysis.

The geometry of the problem is shown in Figure 4.1. The shell and tube heat exchanger consists of a shell filled with PCM and a concentric tube through which the heat transfer fluid flows. The shell is 1000 mm long with an inner diameter of 85 mm, while the tube has an internal diameter of 22 mm. Water is chosen as the heat transfer fluid with a flow rate of 1 L/min (equivalent to an inlet velocity of $\bar{u} = 4.4\text{cm/s}$). This corresponds to a Reynolds number $\text{Re} = 963$. For the PCM, a commercial RT50 PCM from Rubitherm that melts at around 50°C (equivalent to 323K) is used.

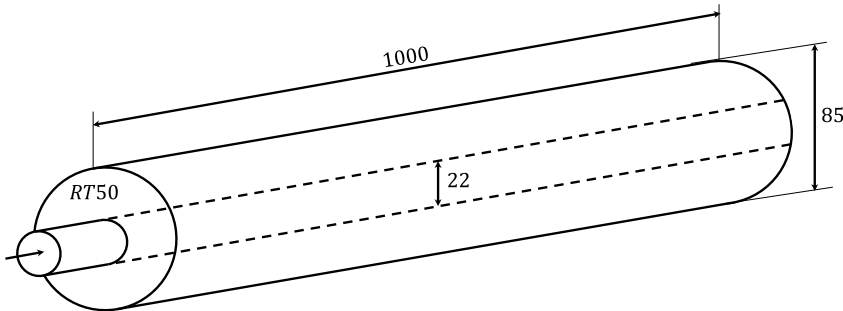


Figure 4.1: Geometry of the problem studied in [Hosseini et al. \(2012\)](#) (not to scale, dimensions given in *mm*).

At the initial condition, the heat exchanger is kept at a constant temperature of $T_0 = 298.15\text{K}$. The water at the inlet is maintained at $T_{in} = 343.15\text{K}$, such that heat will transfer from the tube into the shell, where it will heat up the PCM and lead to its melting. The outer shell of the heat exchanger is insulated to limit the heat that passes through the shell during the experiment. The melting cycle is characterised by strong buoyancy-driven convection, leading to the recirculation of warm PCM melt to the top section of the cylinder, such that the top part of the shell is melted first as visualized in Figure 4.2.

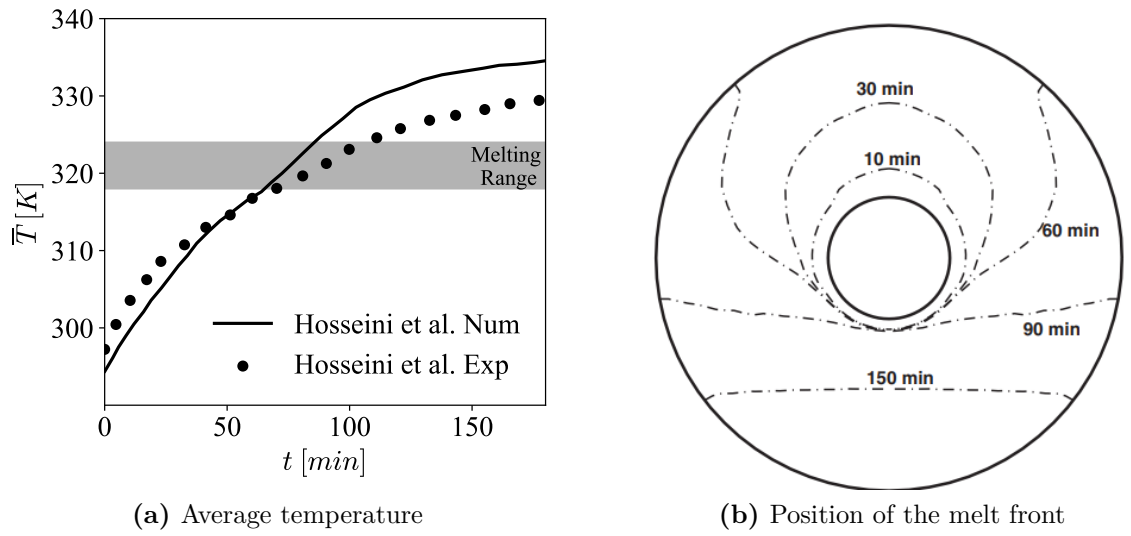


Figure 4.2: Experimental and numerical results for shell and tube heat exchanger study from Hosseini et al. (2014) for $T_{in} = 343.15\text{ K}$.

Reprinted from International Communications in Heat and Mass Transfer, Volume 50, M.J. Hosseini, M. Rahimi, R. Bahrampoury, Experimental and computational evolution of a shell and tube heat exchanger as a PCM thermal storage system, p.135, Copyright (2025), with permission from Elsevier.

Table 4.1: Simplified thermophysical material properties (based on Hosseini et al. (2012) and as implemented in Ansys FLUENT)

Properties	RT50	Water	Copper
ρ [kg/m ³]	780	998.2	8978
μ [Pa.s]	5.04×10^{-3}	0.001003	
c_p [J/kgK]	2000	4182	381
k [W/mK]	0.2	0.6	387.6
T_s [K]	318.15		
T_l [K]	324.15		
L [J/kg]	168,000		
β [1/K]	0.0006		

The material properties used in the paper were taken from the manufacturer Rubitherm and are provided in Table 4.1. In [Hosseini et al. \(2014\)](#), the authors conducted a thorough study to further characterise the variation in the properties of the RT50 PCM at a range of temperatures, including dynamic viscosity and thermal conductivity. This found that the viscosity of the liquid PCM (shown in Figure 4.3a) followed a temperature-dependent relationship defined by $\mu(T) = 0.819e^{-1.546 \times 10^{-2}T}$. The thermal conductivity of the PCM was also studied, and the measurements are shown in Figure 4.3b. This shows that there is a small jump in the phase change temperature range where the thermal conductivity increases before decreasing as the material becomes fully liquid.

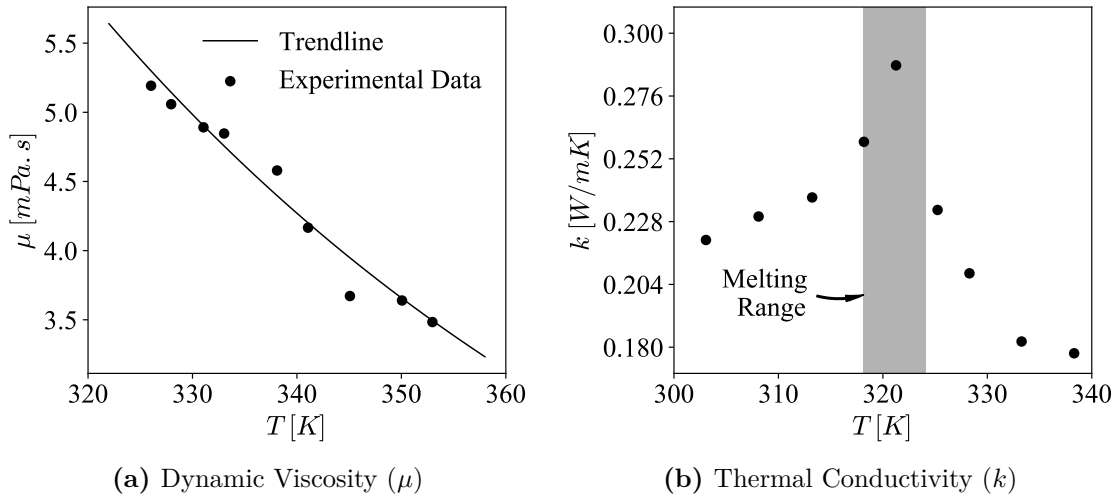


Figure 4.3: Experimental data for Dynamic Viscosity and Thermal Conductivity extracted from [Hosseini et al. \(2014\)](#).

4.3. Solution Independence Study on 2D Geometry

4.3.1. Problem Setup

The geometry was simplified to 2D by isolating the PCM shell from the rest of the heat exchanger and by taking a slice from the shell as shown in Figure 4.4a. To reduce the computational power of the simulation, only half of the shell was considered by using a symmetry boundary condition along the y-axis. The velocity boundary condition at the walls was set to no-slip, the wall temperature in contact with the tube was set to the fluid inlet temperature ($T_{in} = 343.15\text{K}$) and the outer wall was set as adiabatic.

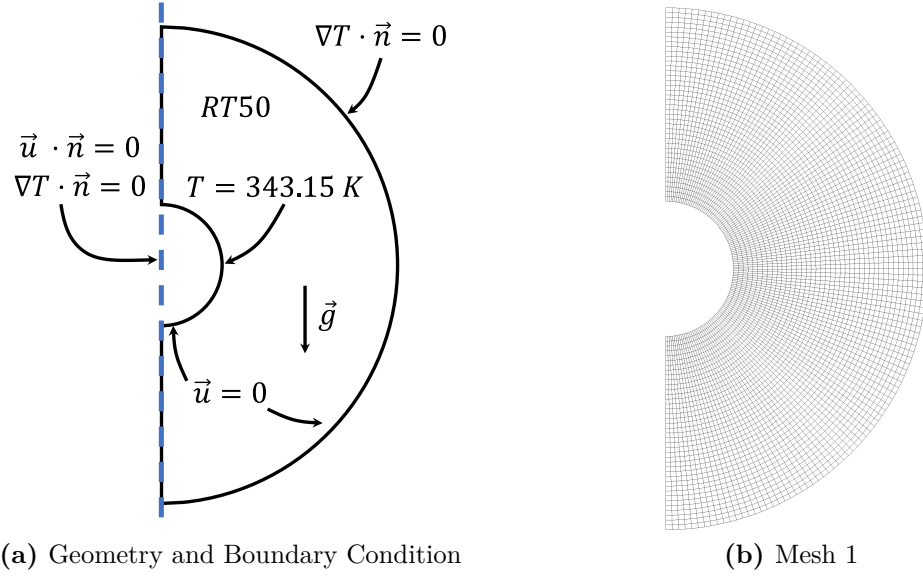


Figure 4.4: Geometry, boundary condition and example mesh (Mesh 1) used in 2D setup.

Natural convection in the liquid PCM was considered by using the Boussinesq approximation, where the reference density was set to $\rho_{\text{ref}} = 780 \text{ kg/m}^3$, based on a reference temperature $T_{\text{ref}} = 324.15 \text{ K}$, corresponding to the liquidus temperature and the acceleration due to gravity $g = -9.81 \text{ m/s}^2$. The flow in the PCM was assumed to be laminar and incompressible. Effects of radiation were neglected in this study.

The melting cycle was started from an initial temperature of $T_0 = 298.15 \text{ K}$, following the initial condition from the paper. The boundary conditions are summarized in Figure 4.4a and the material properties used in the main simulation work are shown in Table 4.1, where material properties given by Rubitherm were used to qualify the PCM.

A commercial finite volume software Ansys FLUENT was used for the solution independence study. The second order bounded implicit scheme was used for time discretization, second order schemes for momentum and the energy terms and least squares cells based was used for the gradient scheme. The PRESTO! scheme was used for pressure as this is the recommended choice for high Rayleigh number flow. The PISO pressure-velocity coupling was used.

Table 4.2: Size of different Meshes

Mesh	Number of Elements
Coarse	5000
Medium	7700
Fine	10800

4.3.2. Mesh Independence and Time Step Independence Study

Three different structured hexahedral meshes were created to investigate appropriate dimensions (see Table 4.2 for corresponding mesh sizes and Figure 4.4b for a representation of the coarsest mesh). For the purpose of the mesh independence, a time step size of 0.1 s was chosen. For this, volume-average properties for the temperature and liquid fraction of the PCM were tracked over time and the position of the melt front was captured at intervals in Figure 4.5.

The volume-average of a quantity ϕ of Volume V was computed in the following way:

$$\bar{\phi} = \frac{\sum_{i=1}^n |V_i| \phi_i}{V} \quad (4.1)$$

Figure 4.5.a shows the advancing melt front for the different meshes, where a distinct position of the melt front was defined as the position of the isosurface, where $\phi = 0.5$. This shows that the biggest difference in the melt front between the different meshes is observed at earlier time steps, when the melting front is predominantly advancing upwards. From this, good agreement between the medium and fine meshes can be seen, while a greater difference is observed for the coarse mesh, where the shape has a higher curvature and is pointing upwards at an angle of approximately 10 degrees. At later time steps no difference is observed for the different meshes that were considered here. Additionally, average properties over time were captured in Figure 4.5.b-c. Little difference is observed for the average properties over time between the different meshes, where the curves overlay each other. For the remaining analysis the fine mesh was selected, as it was observed that the computational time did not increase significantly, as the finer mesh was more stable and converged in less iterations per time step than the medium mesh.

A time independence study was conducted for $\Delta t = [0.5; 0.2; 0.1; 0.05]$ s. The effect of time step on the position of the melt front and on the average temperature and

liquid fraction over time is shown in Figure 4.6. The time independence shows a large discrepancy between results for $\Delta t = 0.5$ s, but very small changes for both the melt front at $t=30$ min and averaged values over time for $\Delta t \leq 0.2$ s. This suggests that the solution is time independent from this value onward, such that a time step of $\Delta t = 0.2$ s was chosen for the remainder of this section.

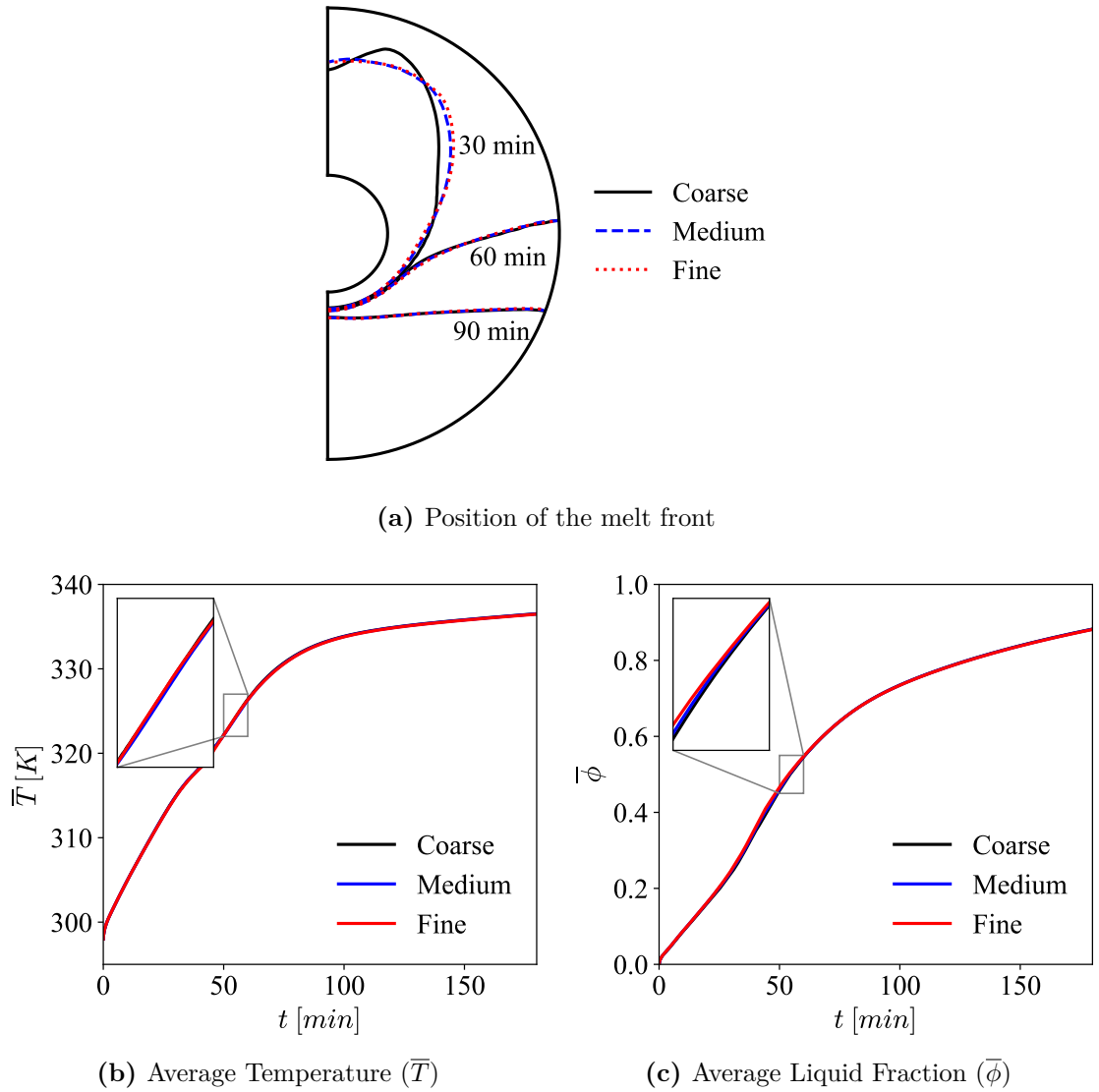


Figure 4.5: Mesh independence: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).

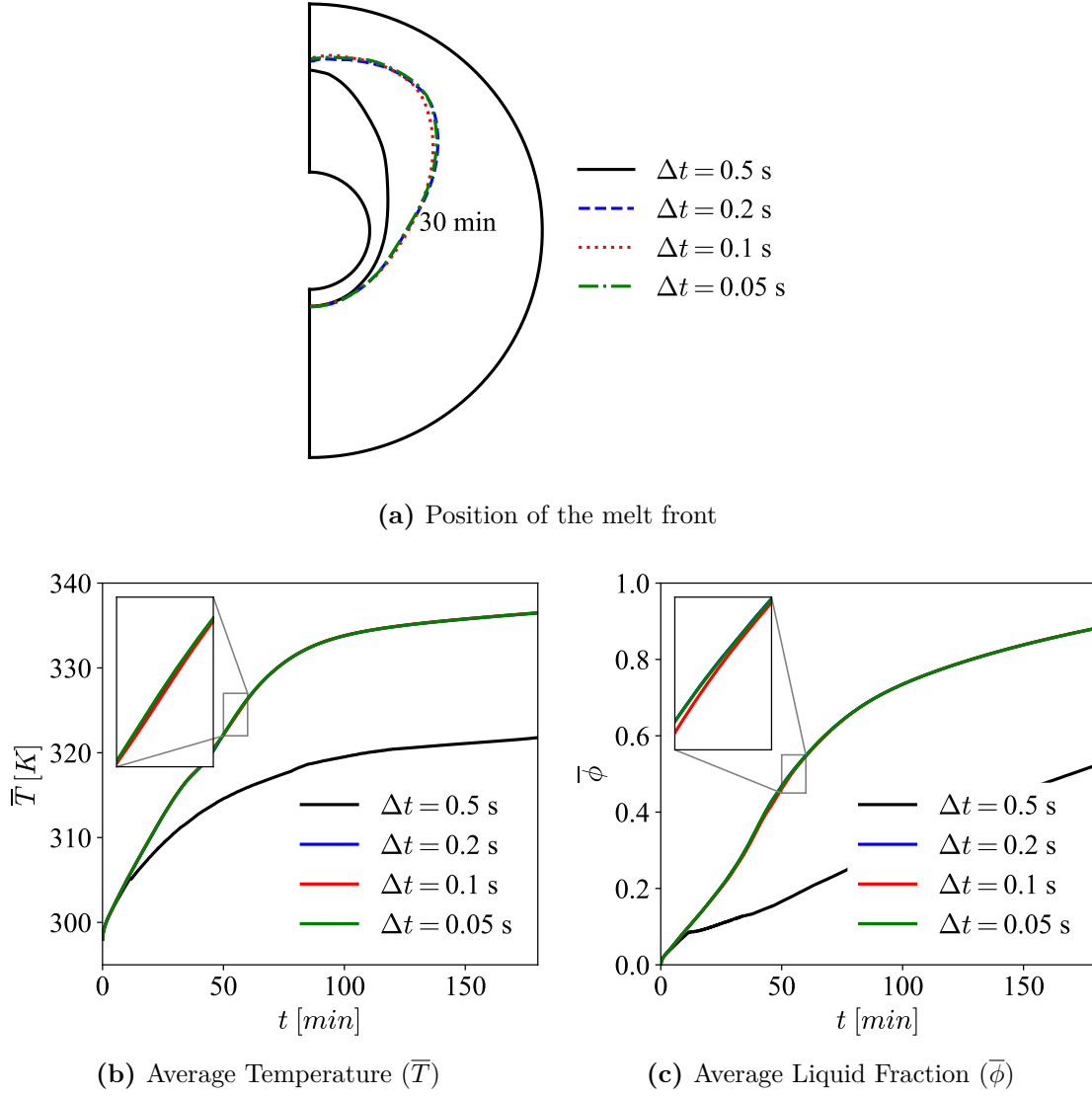


Figure 4.6: Time independence: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).

4.3.3. Description of melting process

The melting process is visualised in Figure 4.7 based on snapshots of temperature contours and streamlines taken between 30 and 180 minutes in time intervals of 30 minutes. Multiple recirculation regions are observed to form within the narrow melting region, arising from the temperature difference between the heated tube wall and colder solid/liquid interface. These recirculation regions convect hot PCM melt towards the upper section of the cylinder at a slight angle, such that this part of the cylinder is melted more significantly than the lower part. The recirculation regions grow and merge as more of the top half of the cylinder is melted between

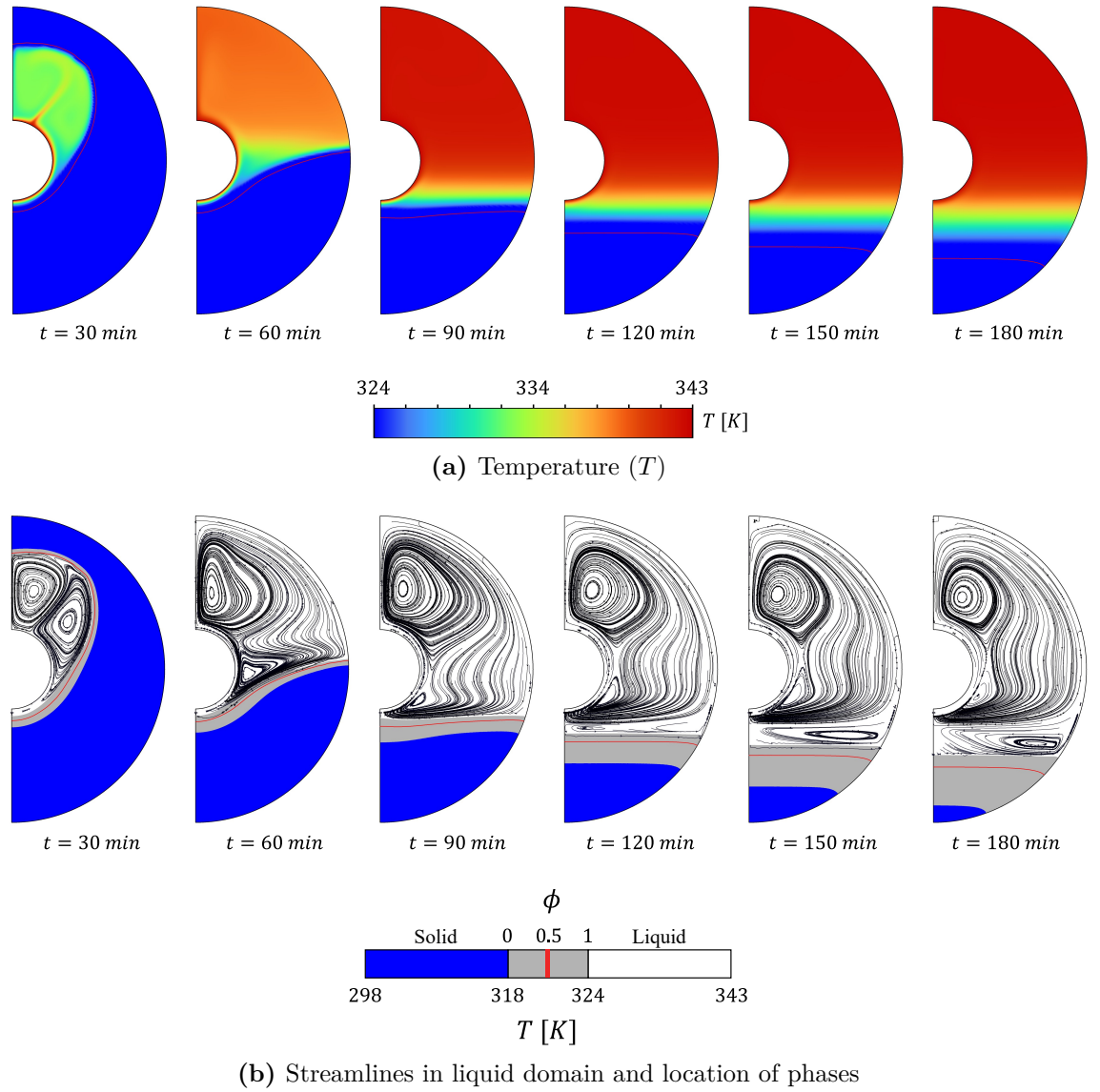


Figure 4.7: Description of the melting process in 2D domain over time: (a) Temperature Contours, (b) Streamlines overlaid on the different phases of PCM.

$t = 30 - 90$ min, leading to an increased melting rate. Following this, the melting rate decreases significantly and the emergence of a third vortex can be observed at the lower part of the cylinder during the solid PCM shrinking phase.

4.3.4. Effect of discretization

The effect of discretization on the results was compared by considering different discretization schemes. For this, first and second order accurate schemes were compared. On top of this, a third case was considered, where the second order pressure scheme was compared with the PRESTO! scheme. A summary of the setup for all three cases is given in Table 4.3.

Table 4.3: Summary of considered schemes

Scheme	First order	Second order	PRESTO!
Pressure	Linear	2nd order	PRESTO!
Momentum	1st order upwind	2nd order upwind	2nd order upwind
Energy	1st order upwind	2nd order upwind	2nd order upwind
Time	1st order implicit	Bounded 2nd order implicit	Bounded 2nd order implicit

Figure 4.8.a shows the melt front at $t = 30$ min. A large difference is observed between using first order schemes and second order schemes. The difference is further visible in Figure 4.8.b-c, where the first order scheme overpredicts the average temperature and liquid fraction when compared to the second order schemes. No measurable difference in results between using the PRESTO! and the Second order scheme for the pressure discretization is observed. The PRESTO! scheme was observed to be more stable, where the solution of coarser meshes was shown to diverge towards an unrealistic solution when using the second order scheme. This justifies using the PRESTO! scheme for the cases where the natural convection of PCM is dominant.

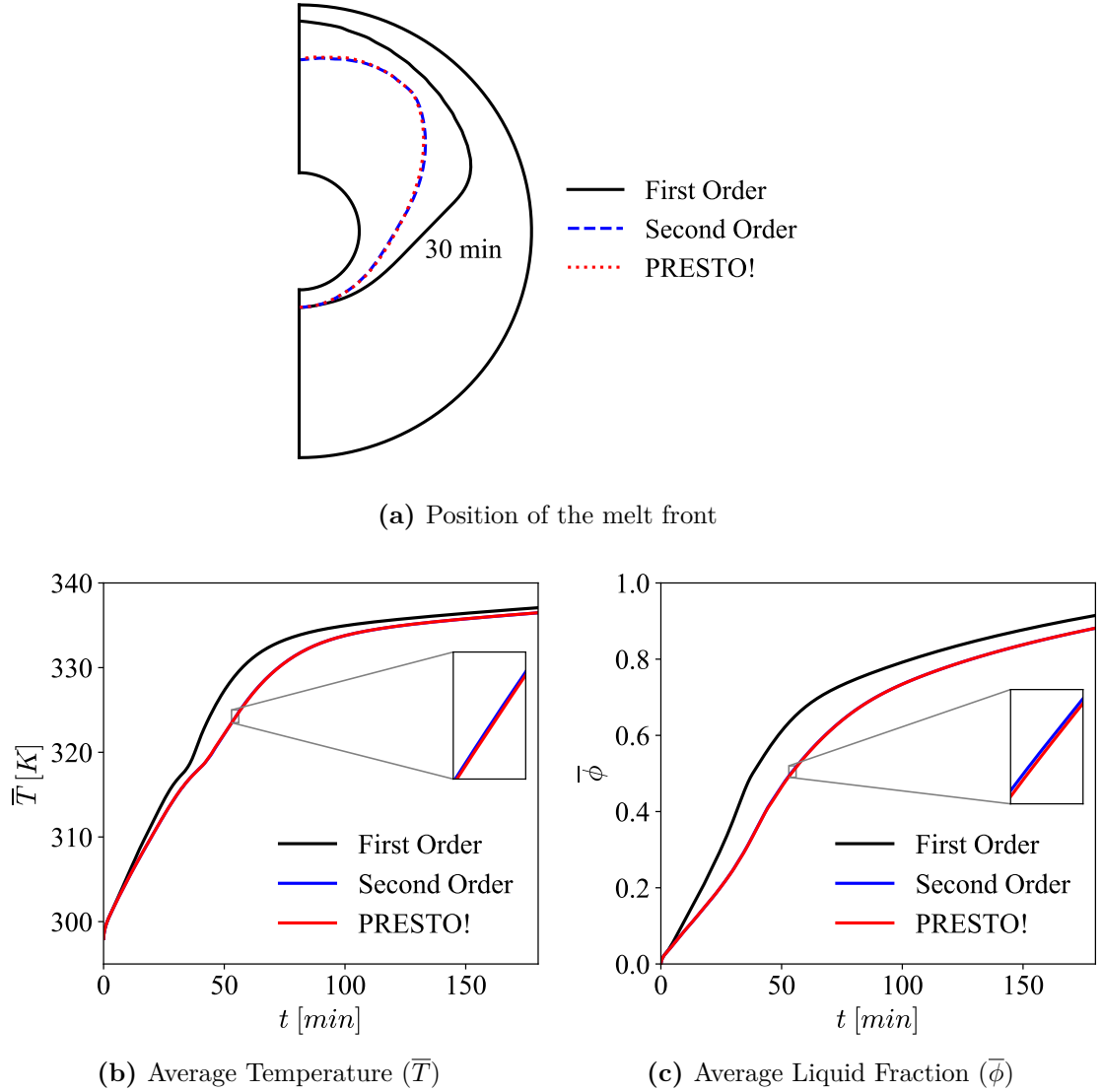


Figure 4.8: Discretization: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).

4.3.5. Effect of material properties definition

In Hosseini et al. (2014), the experiment included determining temperature dependent material properties for the viscosity (μ) and the thermal conductivity (k) as shown in Figure 4.3a and 4.3b. Using FLUENT, variable material properties following these experimental results were set up and the results were compared with a constant definition. A comparison of constant and variable material property definition is given in Figure 4.9. Little difference in average properties and the position of the melt front were observed, suggesting that constant material properties provide a sufficient amount of detail for these kind of problems.

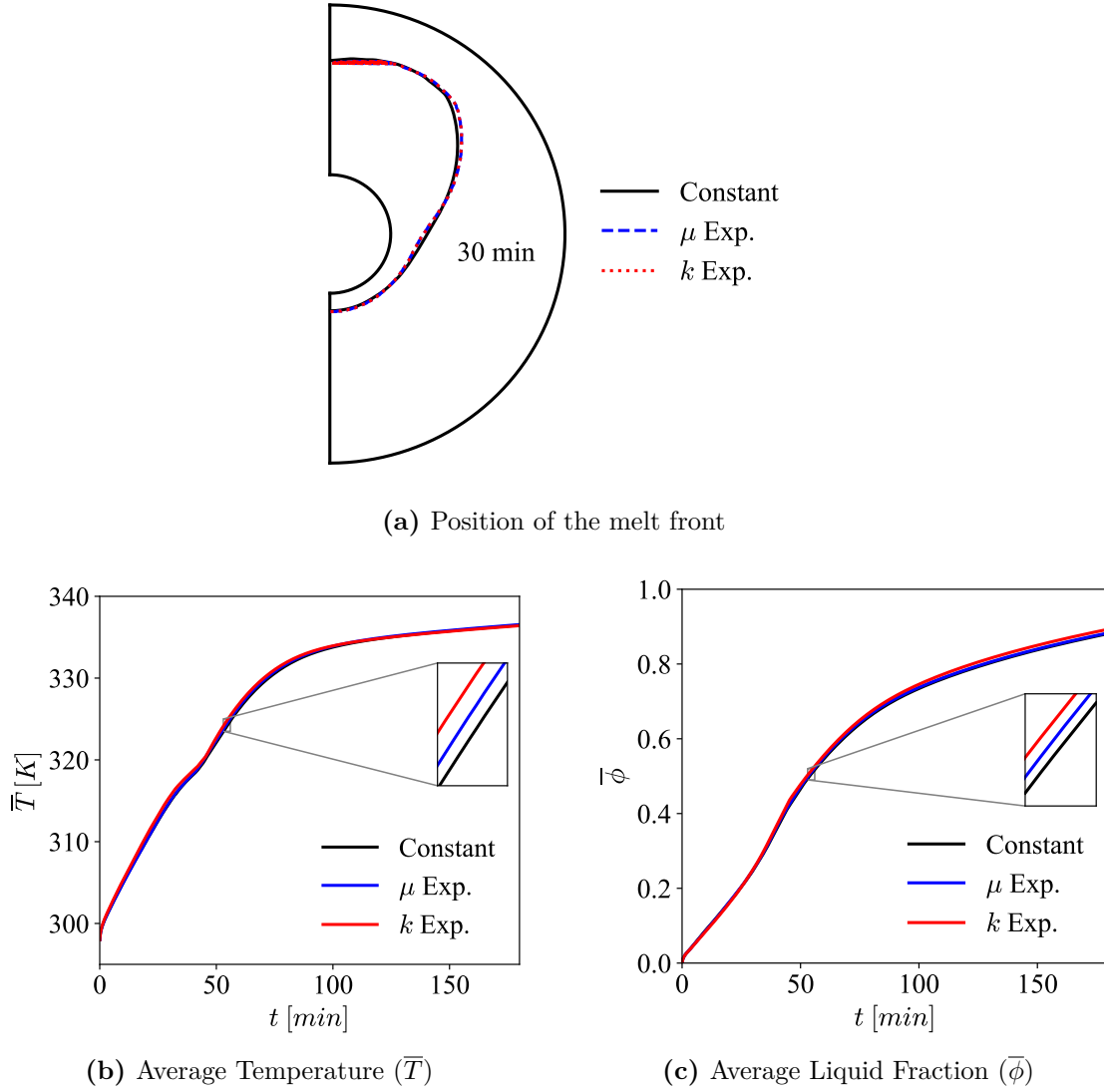


Figure 4.9: Effect of constant and variable material properties - Viscosity (μ) and Thermal Conductivity (k): (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).

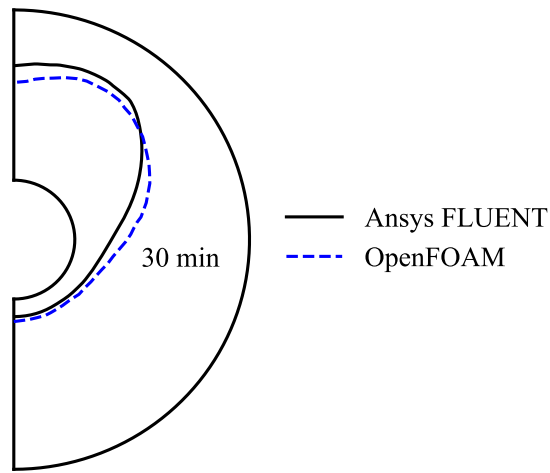
4.3.6. Effect of software packages

A similar simulation was set up in OpenFOAM by using the enthalpy-porosity model that is implemented in the software under the form of the `solidificationMeltingSource` library. Second order precise time schemes were chosen by using the Crank Nicolson scheme with a blending factor of 0.9, to ensure a stable and accurate solution. The second order accurate linear scheme was chosen for divergence, gradient and laplacian terms. The momentum and continuity equation are solved using the PIMPLE algorithm, a combination between the PISO and SIMPLE algorithm. In the PIMPLE algorithm, an additional outer corrector loop is added to the

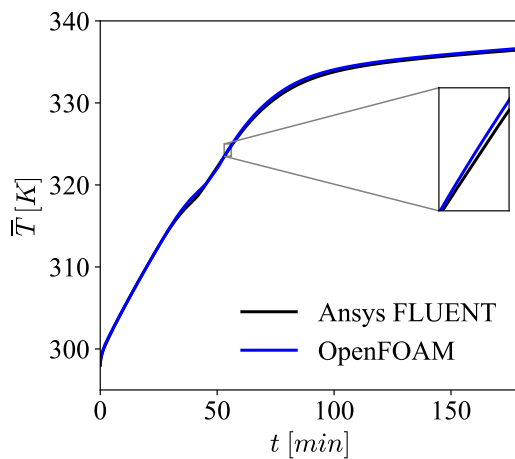
PISO algorithm, where the solution is converged for each timestep. This leads to better stability, so that larger time steps can be used when compared to the PISO algorithm.

Figure 4.10.a compares the position of the melt front for both software packages. The results are in good agreement, with a small shift of the position being observed. Similarly, the average quantities observed in Figure 4.10.b-c are in good agreement, building confidence on the ability of both software packages to produce agreeable results for modeling the melting of PCM over time.

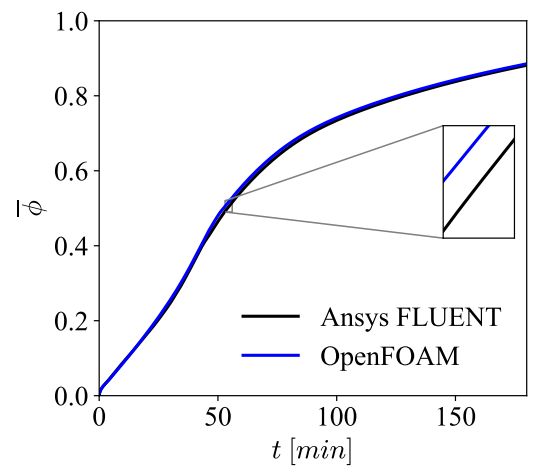
By providing similar computational resources for both simulations (2 cores on ARC3



(a) Position of the melt front



(b) Average Temperature ($\bar{T}$)



(c) Average Liquid Fraction ($\bar{\phi}$)

Figure 4.10: Comparison with OpenFOAM: (a) Position of the melt front; (b) Average temperature ($\bar{T}$) and (c) average liquid fraction ($\bar{\phi}$) over time (t).

HPC facility at the University of Leeds), it was found that OpenFOAM solved the problem significantly faster by using less memory than Ansys FLUENT, based on the statistics provided in Table 4.4. While OpenFOAM seems to be using the computational resources more efficiently, it is worth noting that using FLUENT comes with a lot of advantages. For example, the software is very user friendly, where it is easier to define complex material properties, boundary conditions and meshes in FLUENT. While it is possible to achieve similar outcomes in OpenFOAM, this would have to be coded into the library, increasing the time spent on the task.

Table 4.4: Computational resources used for the 2nd order accurate simulations of a 2D cylinder

	Ansys FLUENT	OpenFOAM
Wallclock Time	45:18:45	04:11:28
Maximum Memory	397 MB	208 MB

This suggests that time that is saved during the computation of the flow and thermal field can be recuperated from a more efficient pre-processing strategy when using FLUENT for more complex geometries than the shell and tube heat exchanger that is considered in this chapter. As the results for the 2D geometry were found to be comparable, both software packages will be used interchangeably for the remaining work, depending on whichever is most suited to a particular problem.

4.3.7. Comparison with literature results

The average temperature over time was compared against experimental results from [Hosseini et al. \(2012\)](#) and further numerical results from [Seddegh et al. \(2016\)](#) in Figure 4.11a. A large difference in the current simulation results was observed when compared to the experimental and numerical results obtained in the original paper, where the average temperature is higher for the current results when compared to the experimental results. Similar observations can be made for the liquid fraction plot shown in Figure 4.11b, where a higher melting rate is observed in the current simulations. [Seddegh et al. \(2016\)](#) recreated the same study in their paper, where they used a similar 2D model that was used in this section, showing a comparable average temperature and liquid fraction profile over time to the current numerical results.

The local position of the melt front was plotted in Figure 4.11c for the current Ansys FLUENT simulation and the numerical results obtained in [Hosseini et al. \(2014\)](#) (at

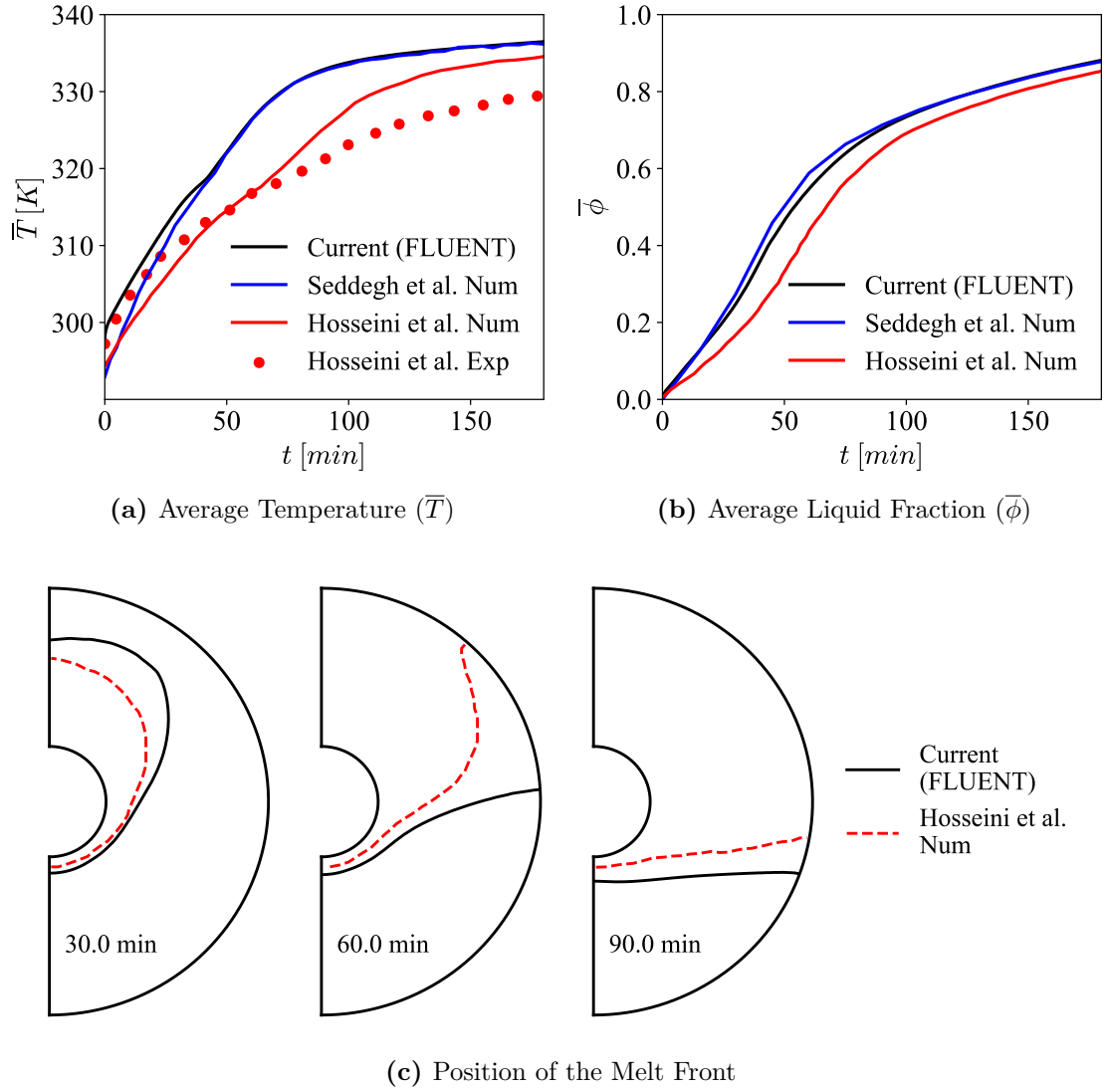


Figure 4.11: Comparison of current numerical obtained from Ansys FLUENT, and results from literature (2D numerical by Seddegh et al. (2016), experimental and numerical by Hosseini et al. (2012)): (a) Average Temperature ($\bar{T}$); (b) Liquid Fraction ($\bar{\phi}$) and (c) Position of the melt front at 30 min intervals.

500 mm from the inlet). This shows, that the position is consistently more advanced in the current 2D simulation, where the biggest difference is observed at 60 min.

The 2D results show a larger average temperature and larger quantity of melted PCM than the results from Hosseini et al. (2012), indicating that more energy has been absorbed by the PCM in the 2D simulation. It is worth noting that in Hosseini et al. (2012), the inner boundary of the shell is not maintained at a constant temperature as is assumed in the 2D simulation. Instead, water circulates inside the tube, which enters the system with a temperature of $T_{in} = 343.15 \text{ K}$. As heat is absorbed by

the PCM over the length of the heat exchanger, the bulk temperature of the water decreases. To take this into account, the full shell and tube system needs to be modelled in 3D by using a conjugate heat transfer approach. This will be covered in the following section.

4.4. 3D Conjugate Heat Transfer Simulation

4.4.1. Problem Setup

A 3D model of the shell and tube heat exchanger was created based on the geometry and boundary conditions shown in Figure 4.12. For this, the `chtMultiRegionFoam` library in OpenFOAM was used, as OpenFOAM was shown to have a lower computational cost when compared to FLUENT, and the shell and tube heat exchanger geometry is relatively simple. A conjugate heat transfer coupling between the water channel, the copper wall and the PCM shell filled with RT50 was created. In this model, the HTF enters the tube by using a constant inlet velocity of 4.4 cm/s at a temperature $T_{in} = 343.15\text{K}$ and leaves the tube through a pressure outlet maintained at $P = 0$ Pa. The water flow is assumed to be laminar due to the low Reynolds number ($\text{Re} = 963$).

All walls were assumed to be no-slip, and an adiabatic condition was used on the outer walls of the shell. The inner boundaries between the different cell zones were modelled by using coupled walls that enforce continuity of heat flux and temperature at the boundaries. Similar to the 2D model, only half of the heat exchanger was modelled by using a symmetry condition and the simulation was solved in a fashion

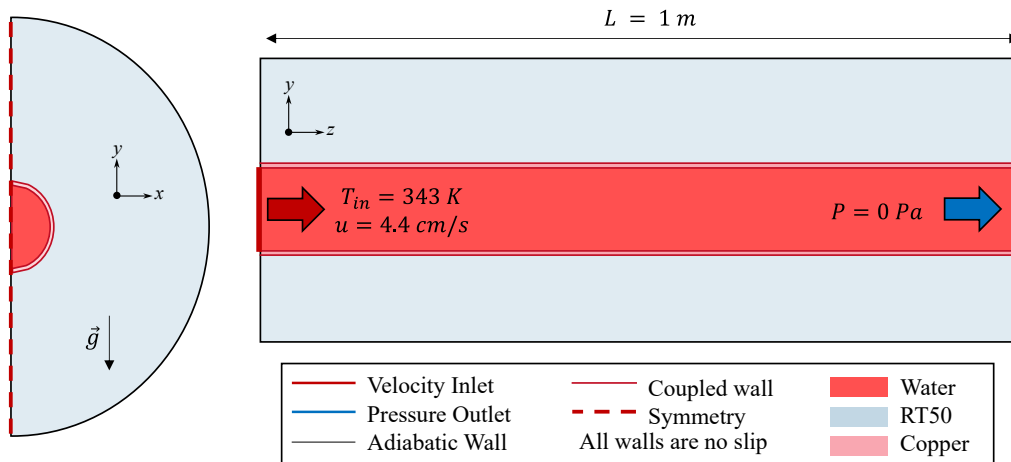


Figure 4.12: Geometry and Boundary Condition of the full 3D domain.

as previously described.

4.4.2. Mesh Independence

A mesh independence was conducted over the first 60 min of the melting process. For this the time step size was reduced to 0.02 s to maintain a CFL condition of approximately 1 for the different meshes (Coarse: 332k, Medium: 538k, Fine: 1.04M cells). Results for bulk properties are shown in Figure 4.13 where a good agreement is seen for the early melting of the PCM where the medium mesh converges towards the fine mesh results. As a result, this mesh was selected for the remaining simulation.

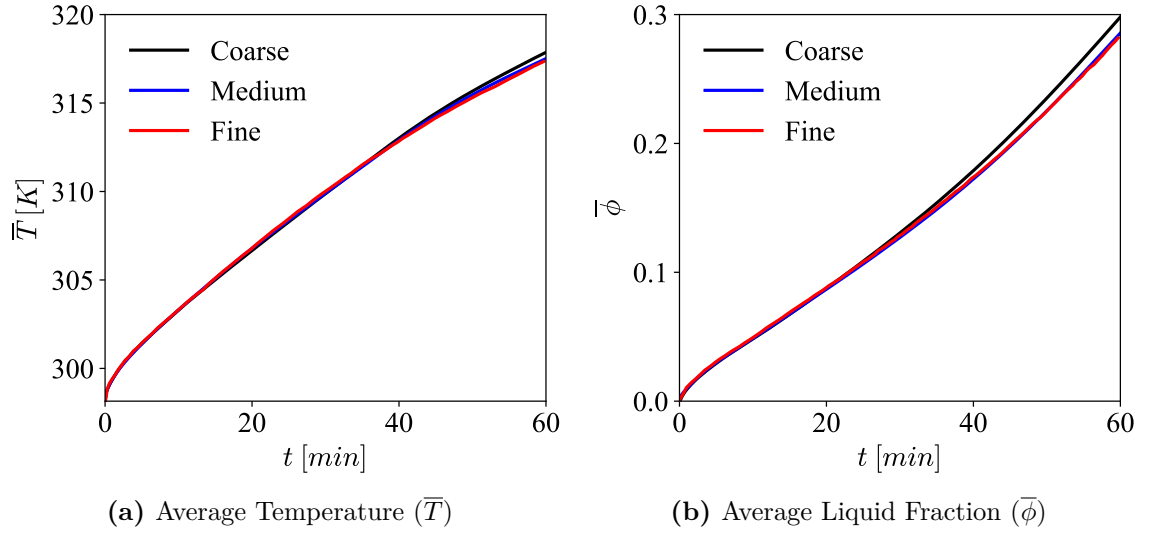


Figure 4.13: Mesh independence (3D long): Average temperature and average liquid fraction over time

4.4.3. Comparison with literature results

Figure 4.14.(a-b) compares bulk properties from 3D simulations from OpenFOAM and Ansys FLUENT with results from Hosseini et al. (2012). The simulations in Ansys FLUENT were generated by using the coarse mesh to save on computational cost. A closer agreement in the prediction of average temperature ($\bar{T}$) can be observed when compared to previous 2D simulations, where the curve lies between the experimental and numerical results from Hosseini et al. (2012). The small difference between experimental and current numerical results could be explained by considering the low number of thermocouples that were used in the experimental study and potential thermal losses through the shell. A greater difference is observed for the average liquid fraction ($\bar{\phi}$), where the current simulations predict slower melting

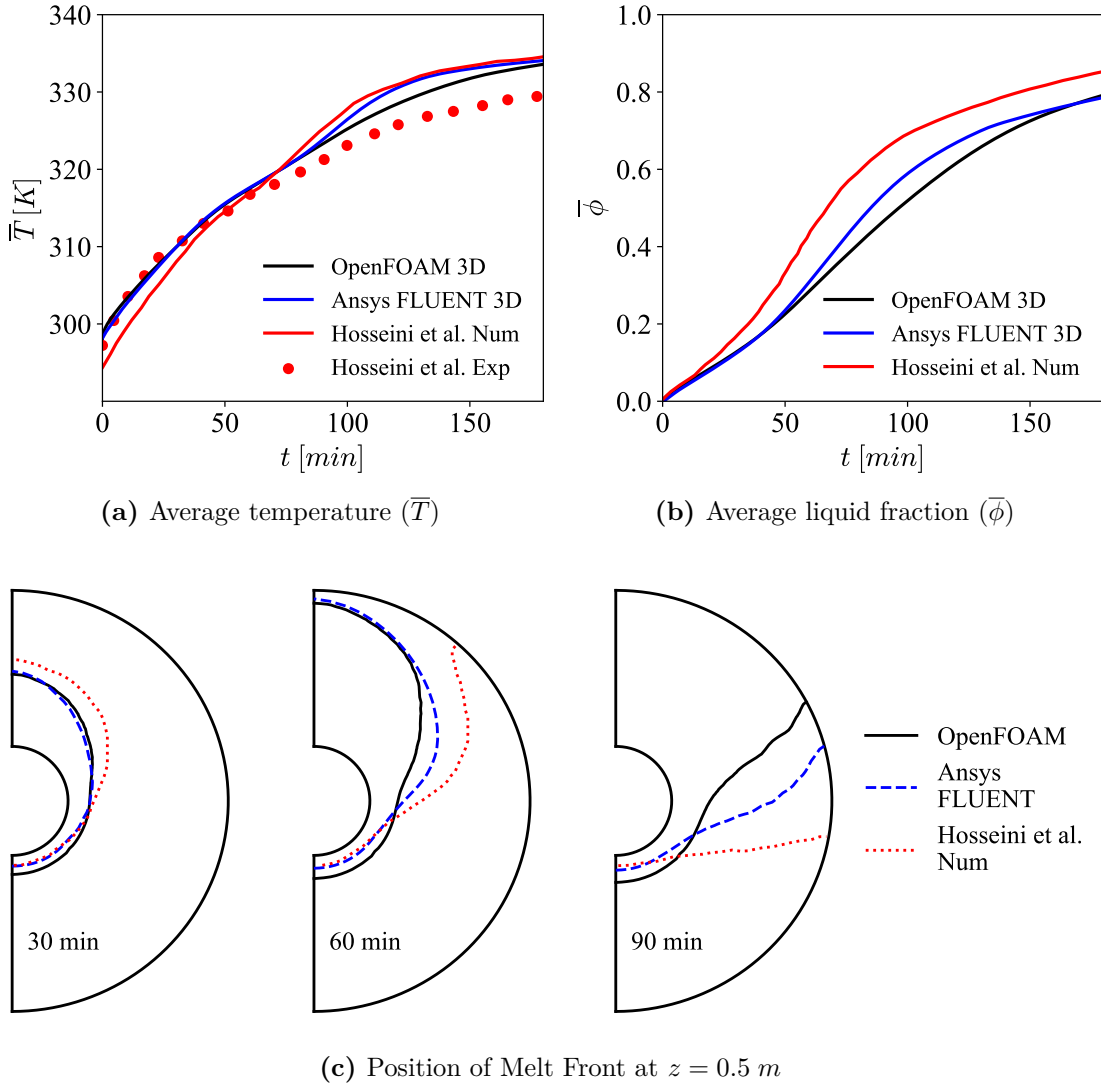


Figure 4.14: Comparison of current 3D simulations and experimental and numerical results from Hosseini et al. (2014).

of the PCM. Figure 4.14.c compares the position of the melt front over time in the centre of the heat exchanger. A greater discrepancy is observed between the current numerical results and the results from the paper. This difference could be explained by the small mesh size used in Hosseini et al. (2012) and limited information on certain boundary conditions, such that the setup could be slightly different.

The results from Ansys FLUENT and OpenFOAM show relatively good agreement over the length of the simulation, although a small difference is observed after $t = 50$ min for the average liquid fraction and $t = 80$ min for the average temperature. The divergence in results is likely to be linked to the difference in meshes between FLUENT and OpenFOAM. It is interesting to note that the FLUENT results seem

to converge towards the numerical results from [Hosseini et al. \(2014\)](#), where a smaller mesh size was also used.

4.4.4. Effect of temperature drop over length of heat exchanger

Figure 4.15 shows the average liquid fraction and average temperature of the fluid for various cross-sections over the length of the heat exchanger over time. From this, it is possible to observe a drop in the liquid fraction over the length in Figure 4.15.a. This change over the length is more significant at early time steps and becomes less significant later in time, such that at 180 min, almost uniform melting over the length of the heat exchanger can be observed. Figure 4.15.b shows the corresponding temperature drop in the water channel. This indicates that up to 90 min, the temperature drop is relatively uniform over time at $\Delta T \approx 4.5 \text{ K}$. Past this point, the top half of the shell is melted and the solid shrinking phase is entered. Here, the temperature drop decreases over time leading to a more uniform temperature of the fluid over the length of the heat exchanger. This could explain why the melting front becomes more uniform over the length of the heat exchanger as time progresses.

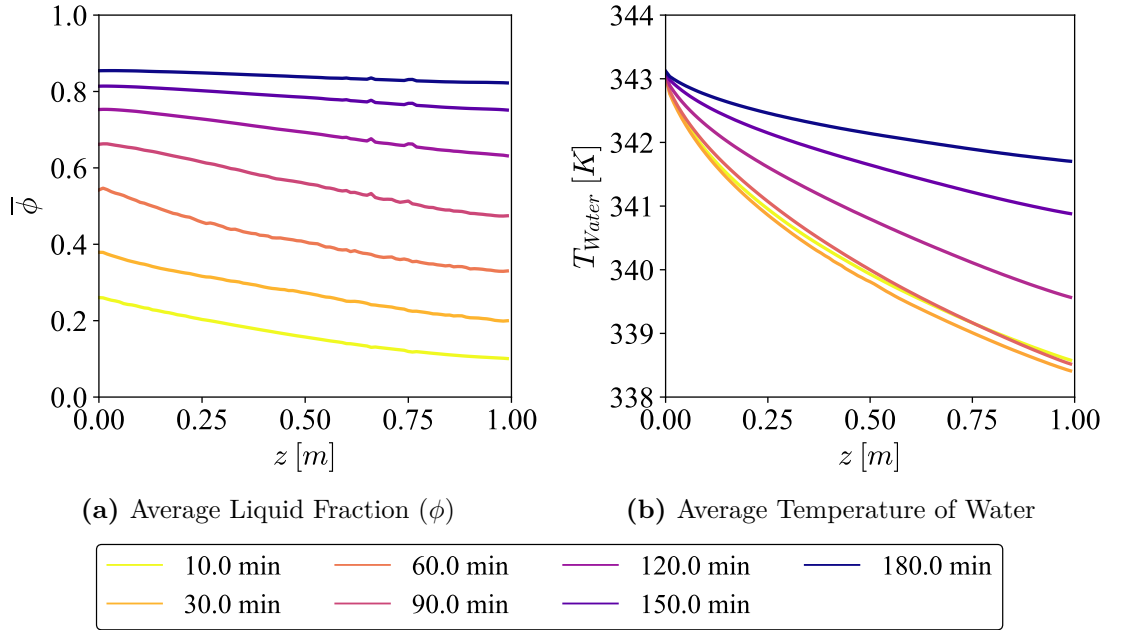


Figure 4.15: Variation in the average liquid fraction and average temperature over the length of the heat exchanger at different time steps.

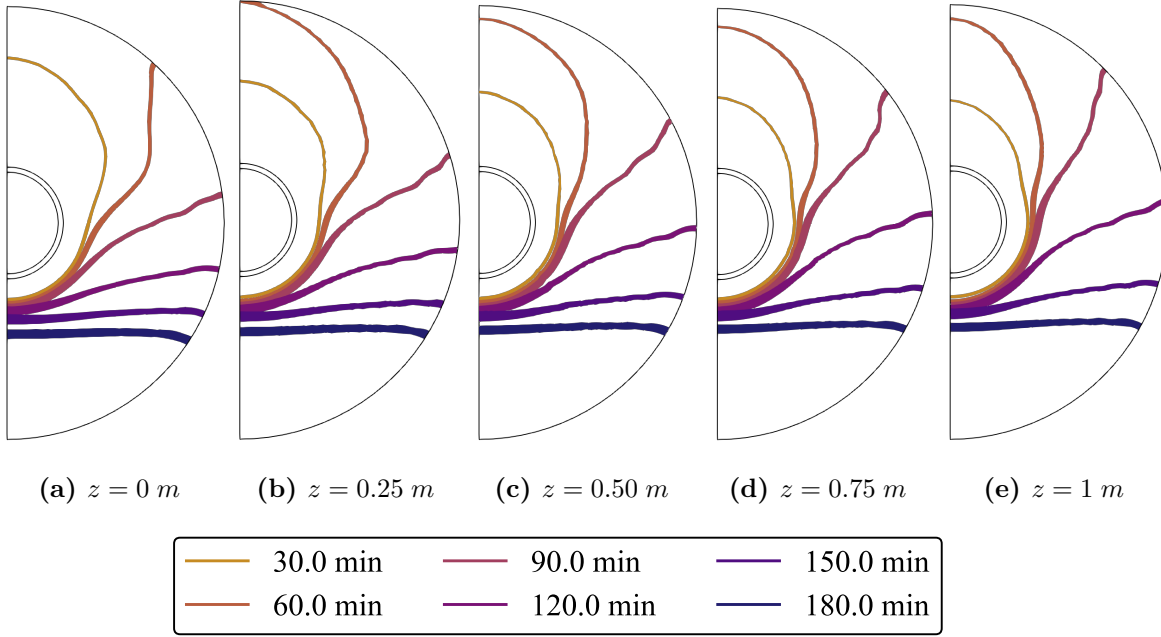


Figure 4.16: Isocontours of $\phi \approx 0.5$ taken at intervals of $t = 30 \text{ min}$ showing the position of the melting front at different locations along the length of the heat exchanger.

Figure 4.16 shows the position of the melt front over time at different positions over the length of the heat exchanger. A drop in the amount of melted PCM over the length of the heat exchanger is observed, with the difference being more pronounced at early time steps, in line with what could be observed in Figure 4.15. The difference is mostly visible in the first half of the length of the heat exchanger and this decreases with distance from the inlet. This is where the temperature drop of the fluid is more important as shown in Figure 4.15.b. The PCM absorbs some of the heat when the phase change happens, so that a temperature drop occurs over the length of the heat exchanger. This leads to a smaller heat flux through the wall separating the PCM and the water, so that less PCM is melted further along the heat exchanger at a given time. As the top half of the shell is completely melted, the melting rate decreases, such that a smaller temperature drop is seen at later time steps. This in turn creates a more uniform distribution of melted PCM along the heat exchanger, than what is initially observed.

In Figure 4.16, the melting interface at $t=30 \text{ min}$ is consistently pointing upwards, while in the 2D model, the melting was propagating at an angle. To gain a better understanding of this phenomenon, a simplified model of the shell will be considered in the next section.

4.5. Detailed 3D PCM section analysis

4.5.1. Problem Setup

A 3D model of a section of the shell (1/10th of the original length) was created based on the diagram shown in Figure 4.17. Only the shell is modelled where a constant temperature of $T = 343.15 \text{ K}$ is applied on the inner wall of the shell, creating a similar boundary condition to the 2D model that was previously studied. The shell is bounded by no slip, adiabatic walls and a symmetry condition along the height of the shell is applied. Simulations were carried out in OpenFOAM using the same time step size of $\Delta t = 0.02 \text{ s}$ as was used for the previous 3D simulation.

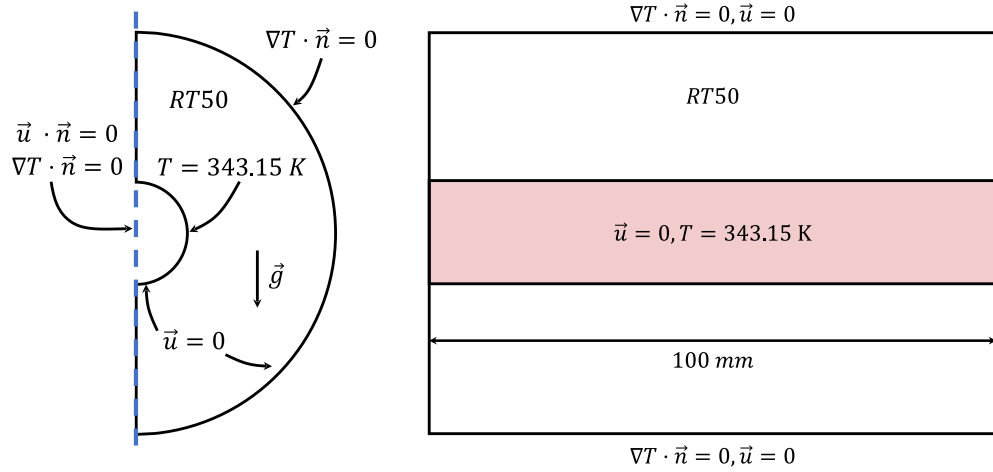


Figure 4.17: Geometry and Boundary Condition for Shell only model.

4.5.2. Mesh Independence Study

A mesh independence study was carried out, where three different meshes (Coarse: 720k, Medium: 1.35M, Fine: 2.27M cells) were compared over the first 20 min in Figure 4.18. Close agreement is observed for the location of the melt front for the three meshes, although the curvature of each plume in the Z direction is not fully resolved for the different meshes. This was deemed to be an acceptable refinement as a greater refinement was too computationally expensive, so that the medium mesh was selected for further analysis.

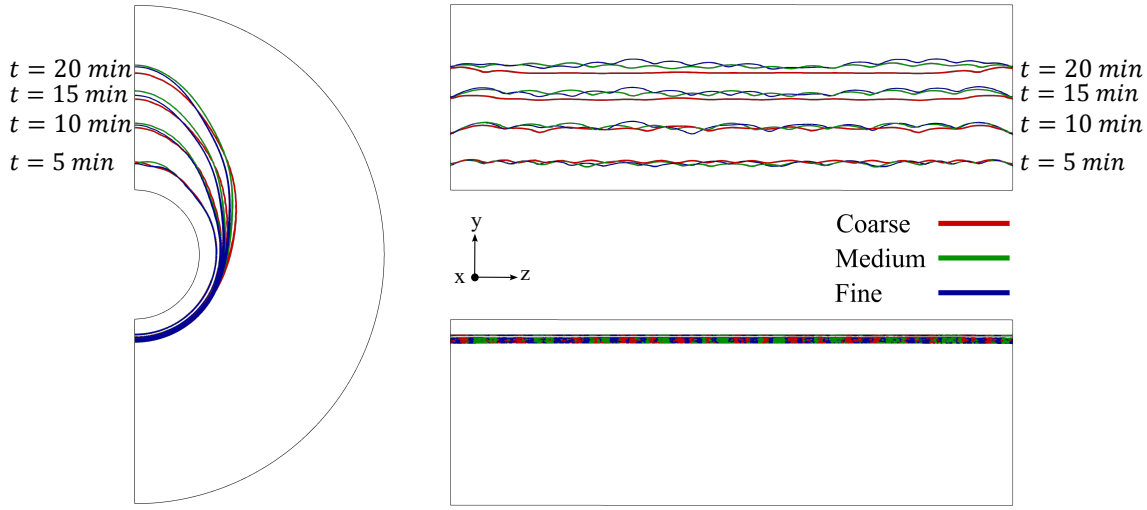
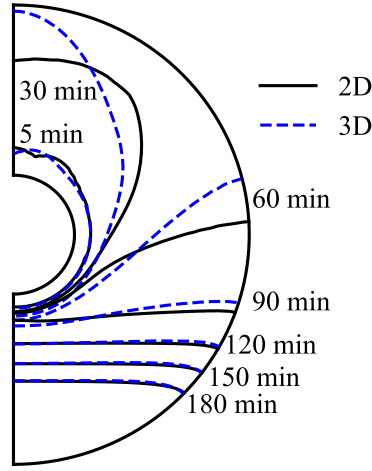


Figure 4.18: Mesh Independence: Position of Melt Front at on XY plane at $Z = 50 \text{ mm}$ (left) and on symmetry plane (right)

4.5.3. Comparison with 2D Simulation

The position of the melting front was extracted over time in Figure 4.19.a at the XY plane in the middle of the domain for the 2D and 3D simulation. In the first half of the cylinder, differences in the melting front are observed between the 2D and 3D results. At $t = 5 \text{ min}$, the position of the melting front is offset at an angle for the 3D case, while the 2D simulation two plumes curvatures are predicted. Later on at $t = 30 \text{ min}$, the two plumes merge into a larger plume for the 2D case, which is slightly angled. The plume of the 3D case is centred again. Figure 4.19 compares the average temperature and liquid fraction over time for the 2D results computed in Section 4.3 and current 3D simulation. This shows that the bulk properties are comparable for the 2D and 3D simulations, indicating that neglecting the flow and heat transfer in the axial direction does not have a major impact on determining the overall heat transfer performance of the shell and tube heat exchanger.



(a) Position of melt front

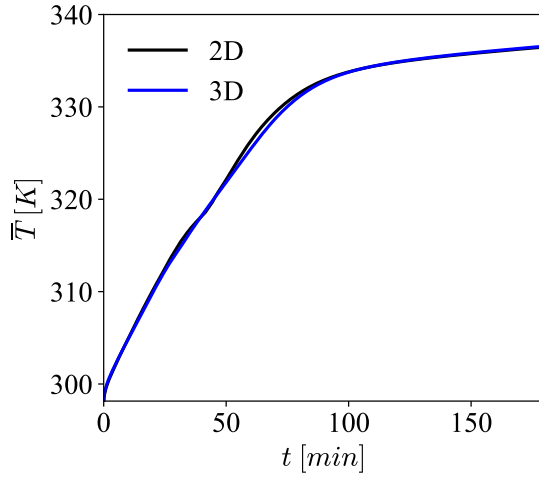
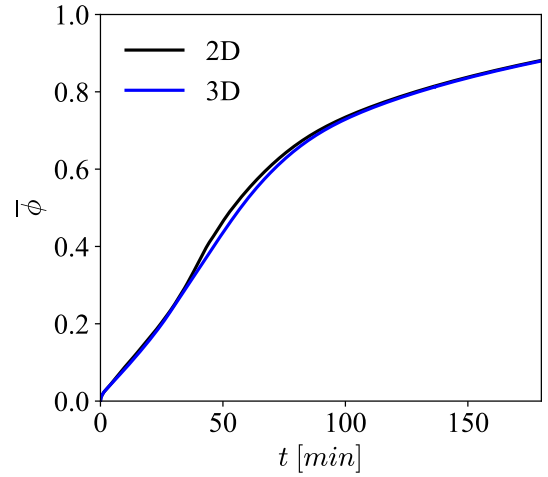
(b) Average Temperature ($\bar{T}$)(c) Average Liquid Fraction ($\bar{\phi}$)

Figure 4.19: Comparison of 2D and 3D simulation: (a) Position of melt front; (b) Average Temperature ($\bar{T}$); (c) Average liquid fraction ($\bar{\phi}$).

4.5.4. Melting Regimes

The following analysis will focus on describing the heat transfer and fluid flow occurring at the early stage of melting in the 3D shell.

4.5.4.1 Conductive regime and onset of convection

Initially, the melting interface grows radially outward, normal to the shell wall as shown in Figure 4.20. As the thickness of PCM increases, this regime becomes unstable, and the flow starts to convect at around 200 s (see Figure 4.21). This first happens close to the side walls of the shell, before the instability travels towards the

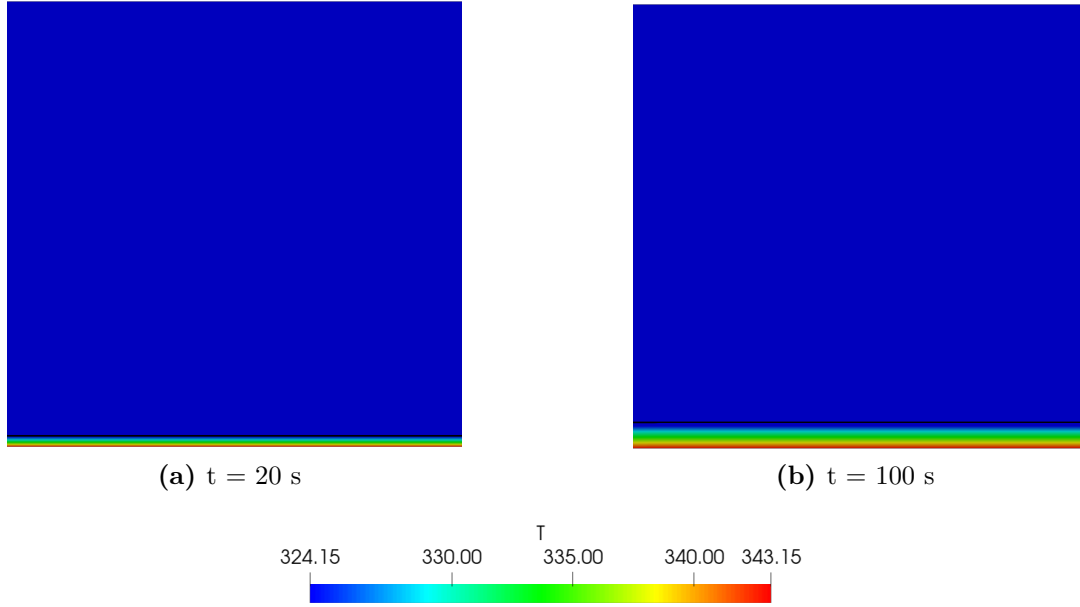


Figure 4.20: Conductive Regime: Contours of temperature with in plane streamlines in the X normal direction. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

center of the cylinder, so that an array of counter-rotating cells is gradually created along the length of the shell. The fact that the instability does not appear uniformly along the length of the heat exchanger is linked to the no slip boundary condition at the cylinder wall, where the additional resistance at the wall triggers the instability in this location earlier than in the rest of the domain.

The plumes convect fluid flow from the heated wall upwards to the melting interface where it is cooled as it travels along the solid/liquid interface. As this happens, the solid/liquid interface takes the shape of the plume leading to a curved front as shown in Figure 4.21.c.

4.5.4.2 Coarsening regime

As the PCM melts further, the plumes become larger and the convection in the plumes becomes more significant in both the lateral and axial direction. The plumes are restricted within the domain in the lateral direction by the walls on either side leading to a competition for lateral space during the expansion of the plumes. This leads to a faster growth rate for some of the plumes as shown in Figure 4.22, where these plumes experience more significant convection, as shown by the large amount of warm fluid that is moved to the interface by the dominating plumes. As the larger

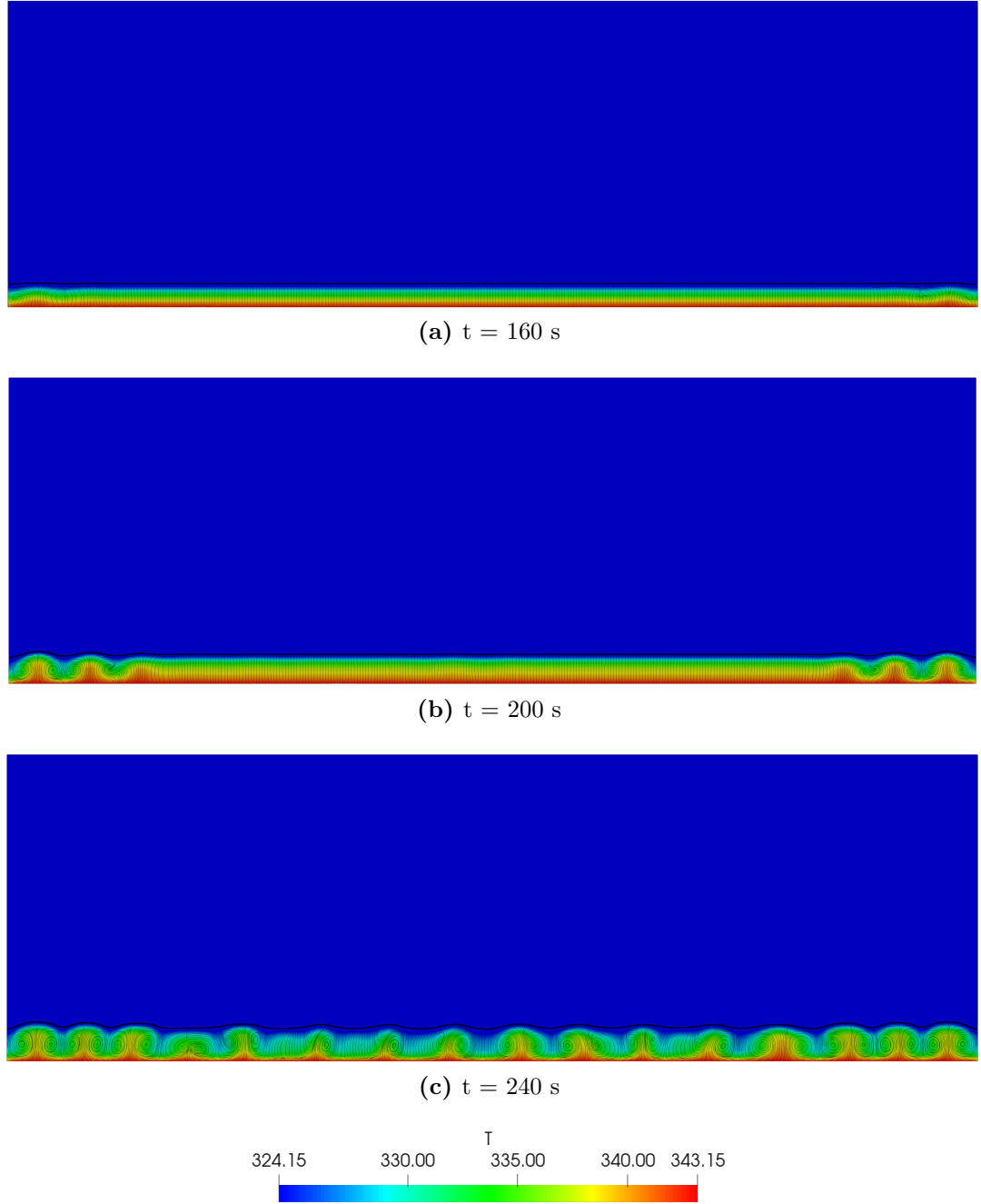


Figure 4.21: Onset of convection: Contours of temperature with in plane streamlines in the X normal direction. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

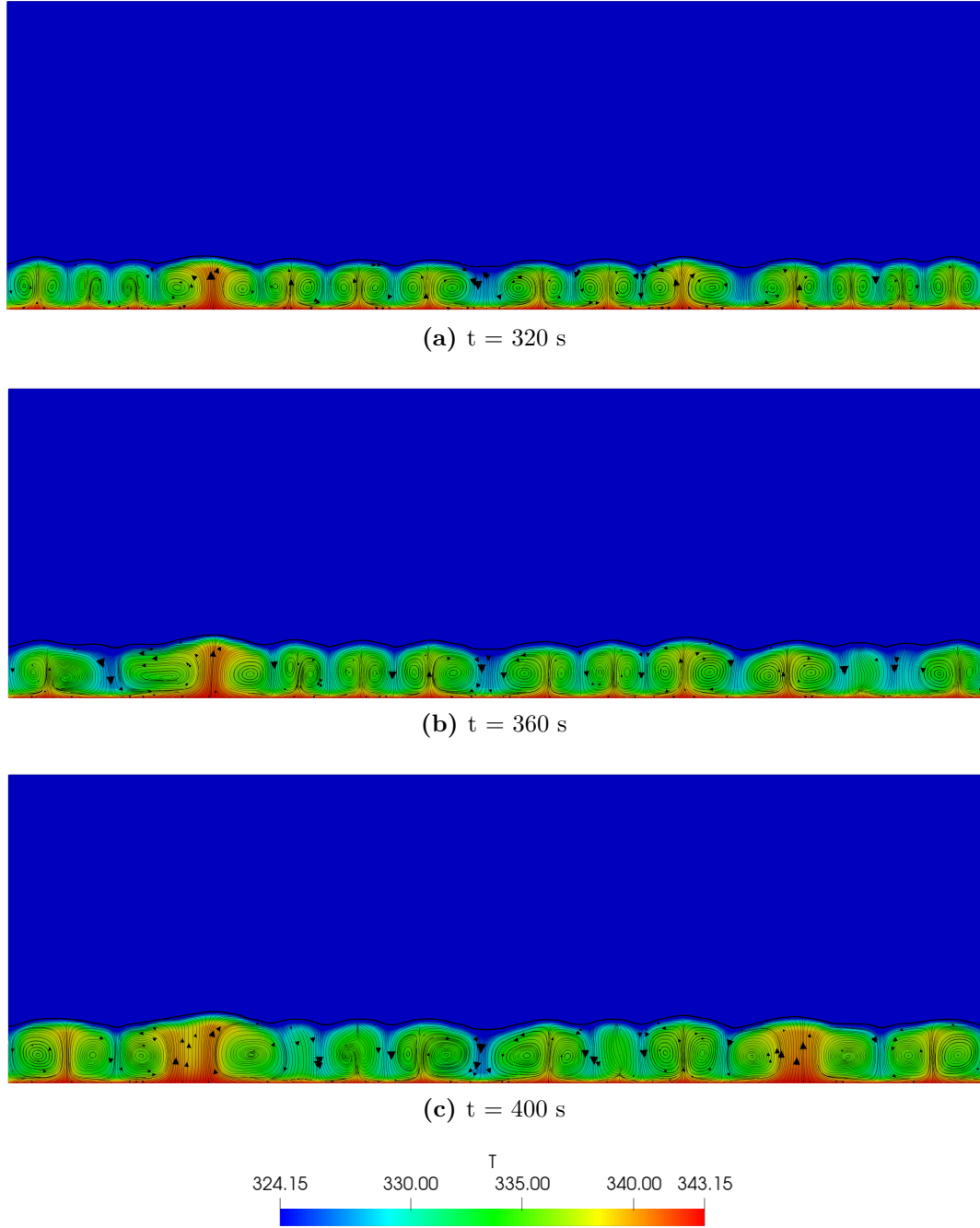


Figure 4.22: Coarsening Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

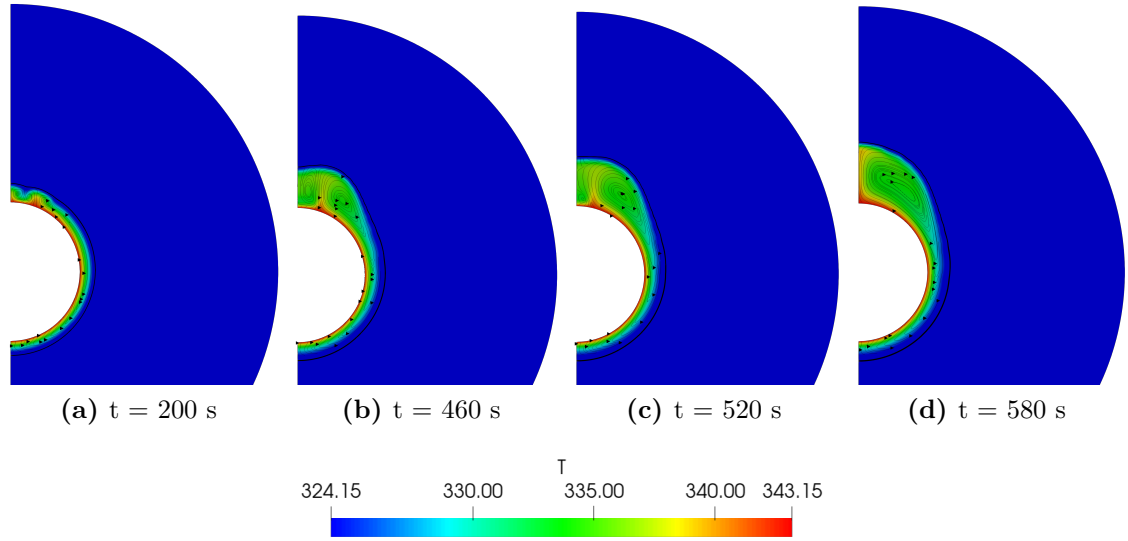


Figure 4.23: Early convection in cylinder: Contours of temperature with in plane streamlines in the Z normal direction ($z=50$ mm). This corresponds to a location in the cylinder where the peak in the solid-liquid interface is initially off-centred. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

plumes expand, neighbouring plumes can be absorbed by the dominating plume, so that a new pair of convective cells with a larger wavelength is created. As this happens, the shape of the interface changes to accommodate the new convective field.

Figure 4.23 shows contours of temperature with superimposed streamlines in the Z normal direction at an early point in the simulation, close to the onset of convection. This shows that initially, there are multiple small plumes that have developed in the small cavity. These rapidly merge into a larger plume as more of the material is melted. Figure 4.23.b-c shows the shifting of the plume from the off-centred position towards the symmetry axis, which is likely to be influenced by the movement of the plumes in the axial direction.

4.5.4.3 Mixed axial-lateral and lateral dominant flow regime

The newly created pairs of convective cells in the axial position are relatively unstable, where the plumes can re-separate as can be seen in between Figure 4.22.b and 4.22.c. This leads to two or more closely packed plumes where each plume has only one convective cell and the plumes are locked in place by the shape of the interface. This contributes to a new flow regime, where there is a gradual shift from

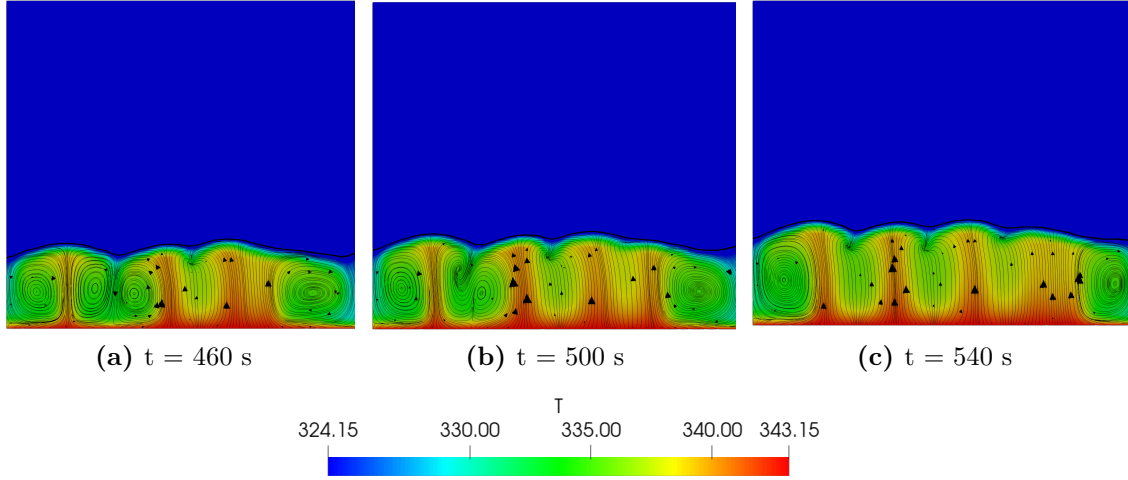


Figure 4.24: Mixed Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

axial-lateral to lateral dominant flow for these plumes (see Figure 4.24).

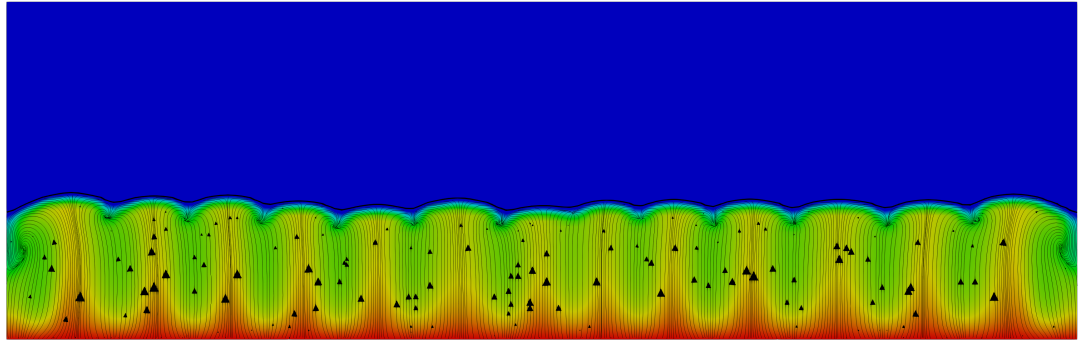
In this regime, the flow in the convective cell transports the plume away from its neighbour, until a more even distribution of the plumes is achieved. As this happens, the plumes in the axial direction gradually disappear from the symmetry plane, and the shape of the melting interface becomes more regular in the axial direction.

4.5.4.4 Lateral flow regime

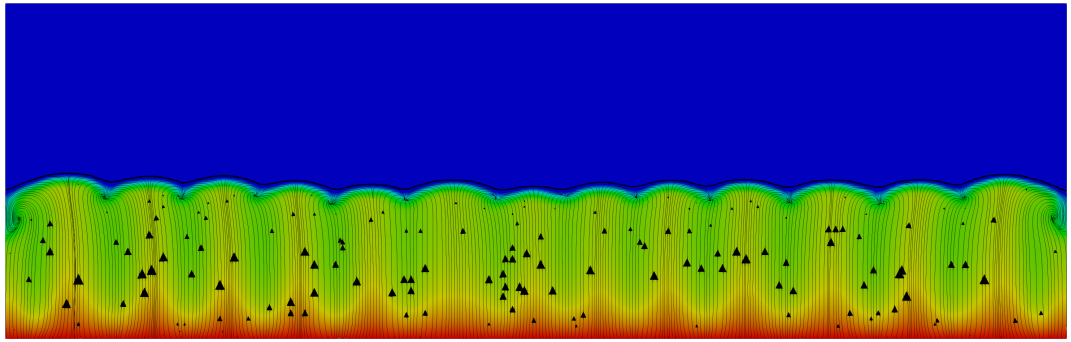
When all convective cells disappear in favour of lateral flow plumes, the plumes are locked in place in the axial direction as shown in Figure 4.25. At this point, the shape of the melting interface does no longer change until the outer wall of the shell is reached.

4.5.5. Movement of the phase change front

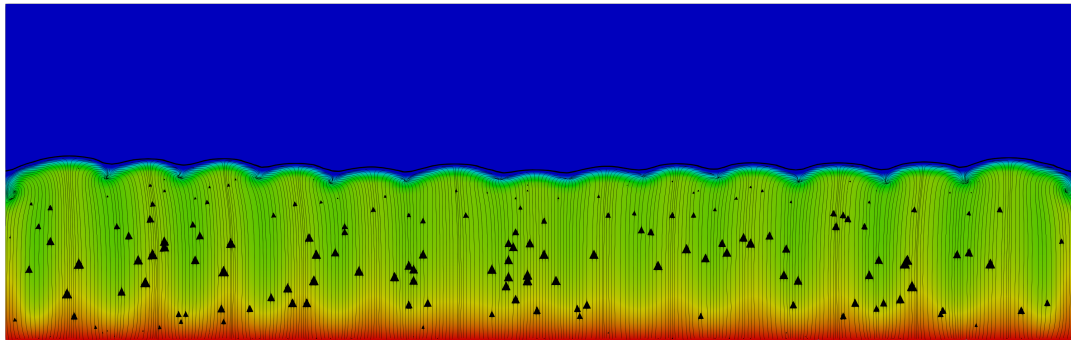
To get a better understanding of the change of axial-lateral to lateral dominated flow inside of the plumes, the liquid phase of the PCM was plotted by using iso-volumes of temperature of the liquid phase in Figure 4.26. To differentiate between axial and lateral plumes, the iso-volume is coloured based on an axial normalized velocity, where the velocity is sized based on the maximum velocity magnitude at that point in time:



(a) $t = 700$ s



(b) $t = 800$ s



(c) $t = 900$ s

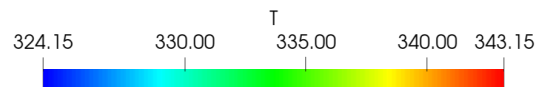
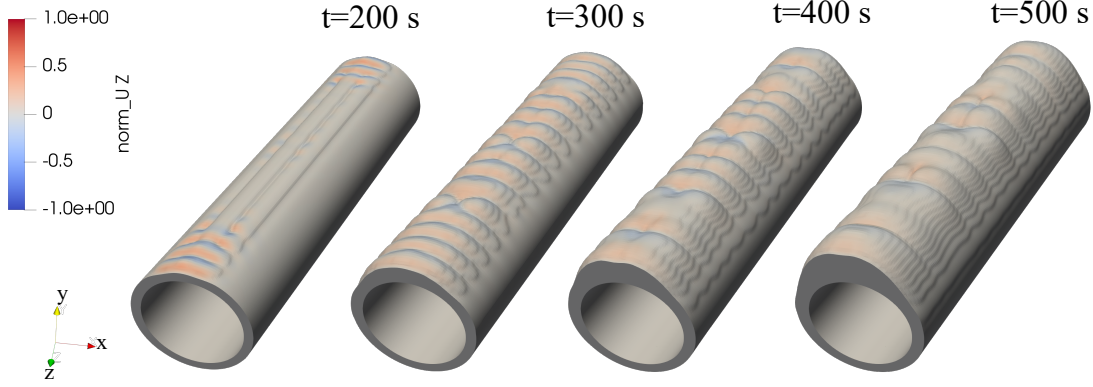


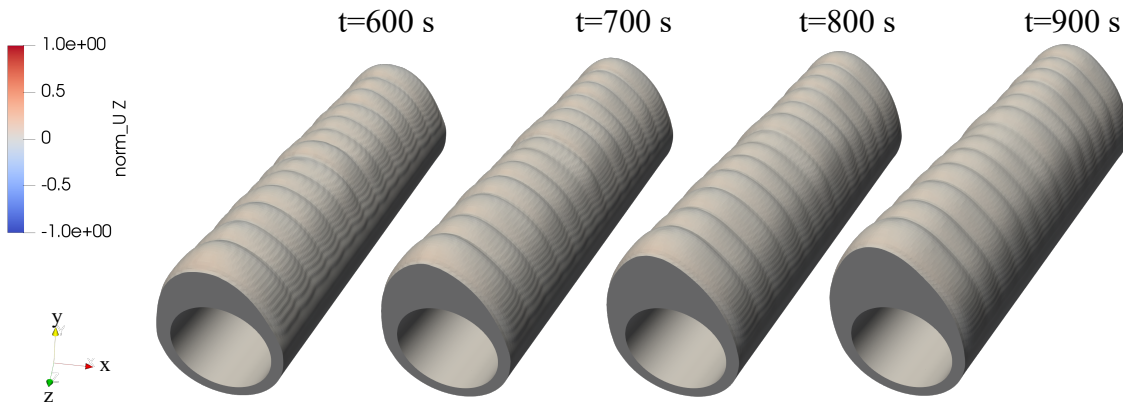
Figure 4.25: Lateral Regime: Contours of temperature with in plane streamlines along the symmetry plane. Temperature contours were limited to the range 324.15-343.15 K, corresponding to the range of temperature where the PCM is fully liquid.

$$\vec{u}_n = \frac{\vec{u}}{|\vec{u}|_{\max, t}} \quad (4.2)$$

At $t = 200$ s, it is possible to see that there are initially multiple small plumes above the cylinder, similar to what could be observed in Figure 4.23.a., where little movement in the axial direction is observable. Close to the walls, a single plume is visible in which the axial flow is relatively important. At $t = 300$ s, the plumes have increased in size and bifurcations in the shape of the melting interface in the lateral are observed where there is either a centred single or a pair of off-centred plumes. A shift in the direction of the flow then occurs, so that a strong axial flow is visible. As a result, this induces the merging of the plumes in the axial direction as shown for $t = 400$ and $t = 500$ s. To accommodate this, the shape of the solid/liquid



(a) $t = 200 - 500$ s: Displacement of melting interface in axial direction due to axial component leads to merging of plumes.



(b) $t = 600 - 900$ s: Low axial velocity component locks plumes in position so that plumes grow upwards

Figure 4.26: Shape of the liquid PCM volume obtained from the iso-volume of temperature in range $T = 324.15 - 343.15$ K coloured based on normalized axial velocity.

interface evolves in the lateral and axial direction with a varying proportion of single or double plumes.

Past $t = 600$ s, the melting boundary is locked into place in the axial direction and only grows in the radial direction (see Figure 4.26.b). This happens as the size of each peak is uniform along the axial direction, indicating that a stable state has been reached. At this point, the flow of PCM follows the centred plume and curves along the melting front normal to the YZ plane.

4.6. Summary

In this chapter, a shell and tube heat exchanger was used as an example to gain a better understanding of the melting behaviour of PCM in heat exchanger and the modelling assumptions needed to capture this behaviour. Different numerical modelling assumptions were tested and compared on a 2D geometry. This showed that second order accurate schemes were required to fully capture the melting of the PCM, with the PRESTO! pressure discretization scheme leading to a higher stability. Using constant material properties was found to be sufficient. the `solidificationMeltingSource` library implemented in OpenFOAM was found to give similar results to Ansys FLUENT on the melting of the PCM, but was found to be more computationally efficient where the simulation time was reduced by a factor of 10 with a lower memory usage. However, the lower computational cost could be offset by the user friendliness of Ansys FLUENT for more complex geometries.

The modelling framework was validated by comparing the average temperature of the PCM over time to experimental and numerical results from [Hosseini et al. \(2012\)](#). For this, capturing the complex coupling between the temperature drop of the heat exchanger and the melting was necessary, as the simulation of the shell led to an overestimation of the melting rate. This could be achieved by implementing a conjugate heat transfer approach, where the fluid channel was modelled. The 3D model achieved close results to the experimental study.

Finally, the melting of PCM within the shell was studied in a detailed simulation of a 3D section of the shell. Similar bulk properties were captured to the 2D simulation, although the shape of the melt front in the early melting of the PCM was found to be different, where the 2D simulation led to angled plumes, while a single centred plume was observed in the 3D simulation. By looking at the melting in the cylinder it was observed that this was linked to the axial flow and movement of the plumes

at the beginning of the melting cycle, that gradually disappears as more of the PCM is melted.

Improving the melting of PCM with finned surfaces in fluid layer

Contents

5.1	Introduction	108
5.2	Aerospace demand profile	109
5.3	Melting in between two heated plates	110
5.3.1	Onset of convection	110
5.3.2	Melting in a plate heat exchanger	115
5.4	PCM in compact heat exchanger - plate fin geometry .	122
5.4.1	Validation of compact heat exchanger modeling	122
5.4.2	Melting of PCM in offset strip fin compact heat exchanger	131
5.4.3	Melting of PCM in continuous fins compact heat exchanger	141
5.5	Effect of different fluid channel geometries	146
5.5.1	Hydrothermal performance of finned and unfinned surface at initial condition	147
5.5.2	Energy Storage and Effectiveness during melting	149
5.6	Summary	152

5.1. Introduction

In the previous chapter, a shell and tube heat exchanger geometry from the literature was covered to validate the CFD modelling methodology. These geometries are commonly used for HVAC units, where a large phase change period is acceptable,

but pressure drop needs to be limited. This leads to relatively large units, with relatively simple fluid channel geometries. This is different to aerospace requirements, where high energy density is required to reduce the mass and volume of the system, while higher pressure drop is allowable. In aerospace, conventional heat exchangers incorporate finned surfaces to increase the compactness of the heat exchanger to achieve this higher energy density.

In this chapter, the use of different fluid channel geometries will be studied to better understand its effect on the melting of PCM over the length of the heat exchanger, by comparing different finned and unfinned plate heat exchangers. For this, a project partner has provided a loading profile for a typical PCM heat exchanger for aerospace applications, which will be presented in Section 5.2. In the following section, the loading profile is applied to an unfinned plate heat exchanger. In Section 5.4, a compact heat exchanger geometry with offset strip fins and continuous fins are studied, before the effect of different extended surfaces on the melting is compared in Section 5.5.

5.2. Aerospace demand profile

A heat load demand profile for a PCM heat exchanger for a military aircraft was provided by an aerospace project partner as part of the project as shown in Figure 5.1. This corresponds to an heat flow of around 5 kW and an equivalent of 500 kJ of stored or released energy from the fluid stream (water) per cycle. To fit this demand curve, RT18HC (see Table 5.1 for material properties) was chosen by the project partner as a suitable PCM with an average phase change temperature of 291.15 K and a latent heat of fusion of 260 kJ/kg. As a result, about 2 kg of PCM will need to be melted in the given cycle to obtain an average heat flow of 5 kW.

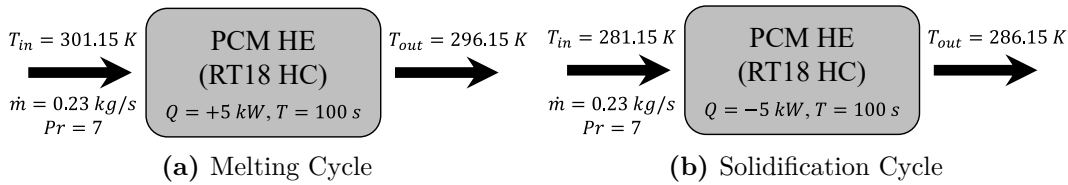


Figure 5.1: Demand profile for an aircraft PCM heat exchanger as provided by project partner, where $Pr = \frac{\mu c_p}{k}$

Note that the melting and solidification cycle are similar, with a similar melting time and a temperature difference of 10 K between the phase change temperature

and the inlet temperature of the fluid.

Table 5.1: Material Properties of RT18HC according to Rubitherm

Material Properties	RT18 HC
Heat Storage capacity [kJ/kg]	260
Melting Range [K]	290.15 - 292.15
Viscosity [mPa.s] at 40 deg C	2.31
Specific heat capacity [J/kgK]	2000
Heat conductivity [W/mK]	0.2
Density Solid [kg/m ³]	880
Density Liquid [kg/m ³]	770

This loading profile will be used for the remainder of this chapter, as well as the rest of the thesis, to design and study novel PCM heat exchanger geometries. As the geometries in this chapter will be getting more complex than the shell and tube heat exchanger considered in the previous chapter, Ansys FLUENT will be used for all simulations. This is linked to the ease in creating high quality meshes around complex geometries in Fluent Meshing.

5.3. Melting in between two heated plates

5.3.1. Onset of convection

In the previous chapter, different regimes were observed during the melting of PCM in a shell and tube heat exchanger. For a short period at the beginning of the cycle, the dominant heat transfer mode was through conduction before the layer of melted PCM started to convect. For aerospace applications, the PCM has to be melted in a significantly shorter period of time in the order of 100 s (see Figure 5.1). As a result, the PCM will be in the conductive regime for the majority of the melting cycle. In this section, the onset of convection will be studied for a cavity filled with PCM and bounded by walls.

5.3.1.1 Problem Setup

A geometry was created based on the domain shown in Figure 5.2 consisting of a $100\text{ mm} \times 5\text{ mm}$ cavity bounded by no slip walls. The cavity was assumed to be filled with RT18HC based on the material properties shown in table 5.1. All walls were assumed to be no slip and a constant temperature boundary condition of $T = 301.15\text{ K}$ was applied to the lower wall, while all other walls were assumed

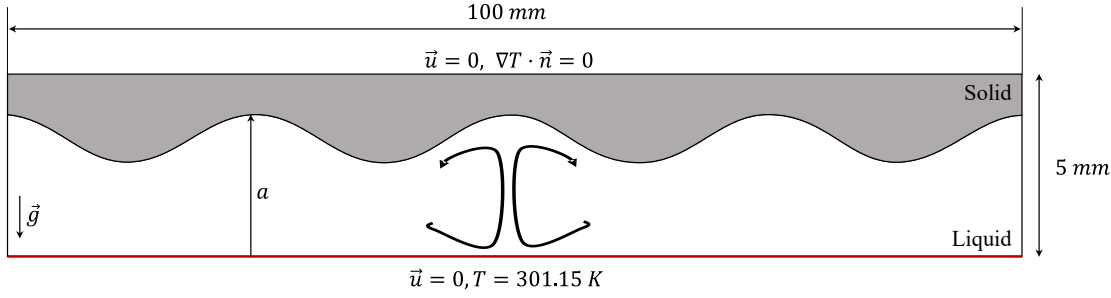


Figure 5.2: Diagram of the problem considered for measuring the onset of convection. Note: not to scale.

to be adiabatic. Effects of natural convection were taken into account by using the Boussinesq approximation and setting the gravity to $g = -9.81 \text{ m/s}^2$. As a thermal expansion coefficient for RT18HC could not be provided by Rubitherm, it was assumed that $\beta = 0.0006$, similar to the value used in the previous chapter for RT50. Similarly, no appropriate value of the mushy zone constant was provided such that $A_{\text{mush}} = 10^6 \text{ kg m}^{-3}\text{s}^{-1}$ was used as part of the enthalpy-porosity model.

At the initial time, the PCM temperature was set to 290.14K, corresponding to a temperature close to the solidus temperature such that the PCM is fully solid, but starts melting imminently after the start of the melting cycle. The PISO pressure-velocity coupling and 2nd order accurate numerical schemes (see Table 5.2 for more details) were used to solve for the melting of PCM, based on the study that was conducted in the previous chapter to increase stability and accuracy of the simulation. The simulation was run for a total time of 500 s, corresponding to 5 times the melting cycle, with a time step size $\Delta t = 0.1 \text{ s}$, which was found to be sufficient for simulating the melting of PCM in the previous chapter.

Table 5.2: Summary of numerical schemes used for the melting of PCM in a cavity. WB: With Buoyancy; NB: No Buoyancy.

	Discretization Scheme
Gradient	Least Squares Cell Based
Pressure	PRESTO! (WB) / Second Order (NB)
Momentum	Second Order Upwind
Energy	Second Order Upwind
Transient	Bounded Second Order Implicit

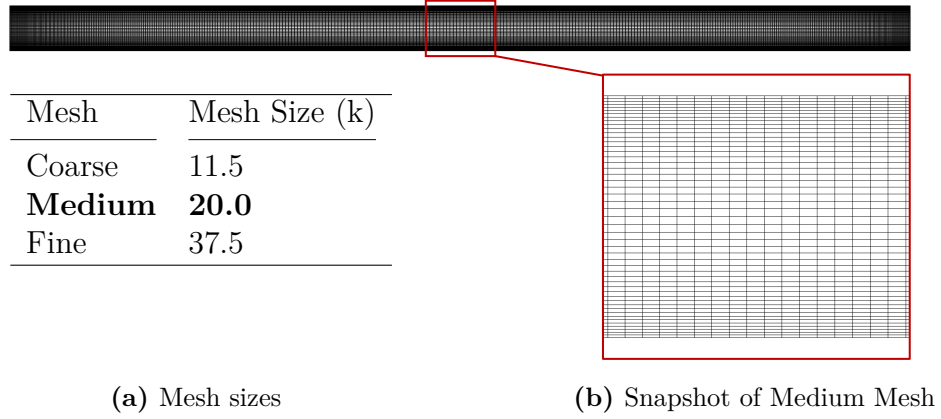


Figure 5.3: Mesh information for independence study for onset of convection study in a 2D bounded cavity

5.3.1.2 Mesh Independence Study

A mesh independence study was conducted for three different hex meshes as shown in Figure 5.3. For this, the location of the solid/liquid interface was tracked over time in Figure 5.4 based on the melting thickness a , defined as the distance of the interface from the heated wall. Based on this analysis, it is possible to see that the medium mesh provides a similar prediction of the position of the melting interface when compared to the fine mesh in both the conduction dominated and early convection regime. As a result, the medium mesh was selected for further analysis.

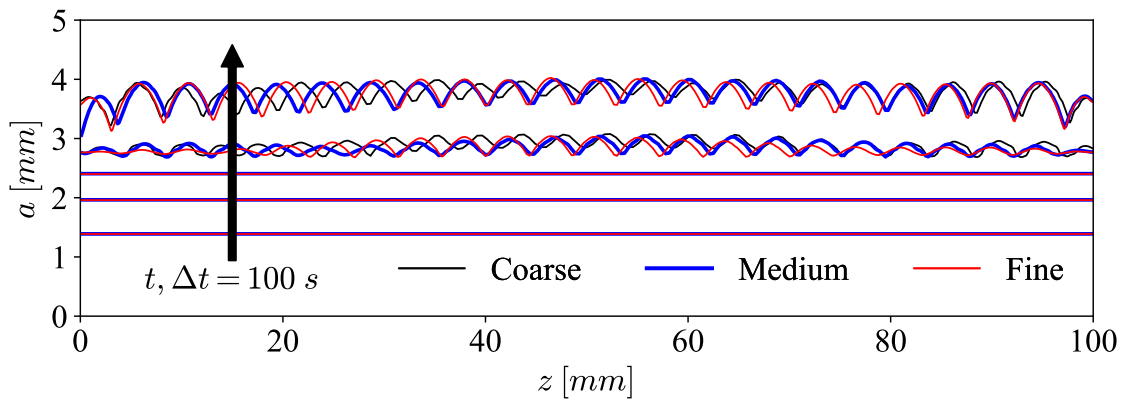


Figure 5.4: Mesh Independence Study: Comparison of the location of the solid liquid interface over time ($t = 100 - 500$ s) based on the melting thickness (a), obtained by creating an iso-surface of $\phi = 0.5$.

5.3.1.3 Effect of buoyancy on melting

To study the effect of buoyancy on the melting of PCM within the cavity, a case where the natural convection was suppressed was compared with the case with natural convection. This was achieved within the simulation by setting the gravitational acceleration to:

$$\begin{cases} g = -9.81 \text{ m/s}^2 & \text{With Buoyancy,} \\ g = 0 \text{ m/s}^2 & \text{No Buoyancy.} \end{cases} \quad (5.1)$$

For the case where the effect of natural convection is neglected, there is no flow in the PCM, so that the enthalpy-porosity model simplifies to an enthalpy model. In this model, the heat equation is simplified, where the convection is neglected and a single source term is used to add or remove heat from the system based on how quickly the material melts:

$$\frac{\partial \rho_{\text{PCM}} h}{\partial t} = \nabla \cdot (k_{\text{PCM}} \nabla T) - \underbrace{\frac{\partial \rho_{\text{PCM}} \Delta H}{\partial t}}_{\text{Source Term}}, \quad (5.2)$$

where ΔH is the latent heat enthalpy and is defined based on the liquid fraction ϕ and the latent heat storage capacity L :

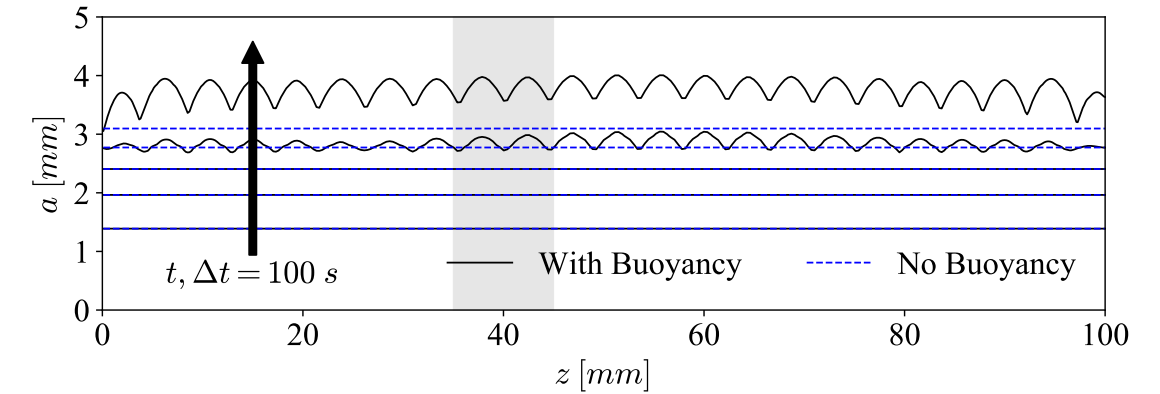
$$\Delta H = \phi L. \quad (5.3)$$

Figure 5.5 tracks the position of the melting interface for the two cases over time. This shows that in the conduction dominated regime up to $t \approx 300 \text{ s}$, the two simulations predict a similar amount of PCM to be melted and the shape of the melting front is linear and grows outward normal to the heated wall. The onset of convection occurs in between 300 and 400s, at which point the shape of melting front starts to be corrugated (see melting front with buoyancy). Close to the onset of convection at $t = 400 \text{ s}$, both models predict a similar amount of melted PCM, but the no buoyancy case does not predict the corrugation of the melting interface due to natural convection. At a later time, the thickness of melted PCM increases so that the convection in the melted pool and the velocity of the melting front becomes more significant. As a result, the conduction only case no longer predicts the amount of PCM accurately.

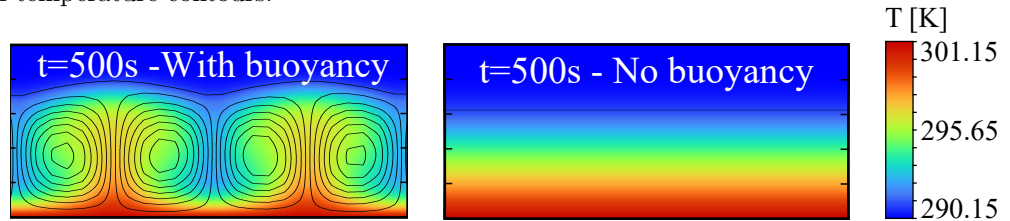
To get a better understanding of the point at which the PCM starts to convect, bulk properties are compared for the two cases in Figure 5.6. This indicates that

the onset of convection occurs around $t = 380$ s, at which point, the prediction of the stored sensible diverges for the two modelling approaches. The effect of buoyancy on the amount of melted PCM occurs a little later around $t = 400$ s, at which point a significant increase in the liquid fraction can be observed for the case where buoyancy is modelled.

Based on this analysis, it was shown that the onset of convection occurs after the end of the melting cycle for the loading profile. By only considering conduction, similar results were obtained for $t < 100$ s when compared to the simulations with convection. As such, it is possible to assume that conduction will be the dominating mode of heat transfer for the aerospace PCM heat exchangers in this chapter, so that natural convection will be neglected in the PCM. Due to the symmetry of the loading profile, the solidification cycle will behave similarly to the melting cycle, such that an equivalent amount of heat will be absorbed during the melting cycle to what is released during the solidification cycle. As such, only the melting cycle will be considered in this chapter.



(a) Position of the melting front (a) over time ($t = 100 - 500$ s). Shaded area indicates location of temperature contours.



(b) Temperature (T) contours at $t = 500$ s

Figure 5.5: Effect of buoyancy on melting based on two different modelling assumptions: with buoyancy and no buoyancy.

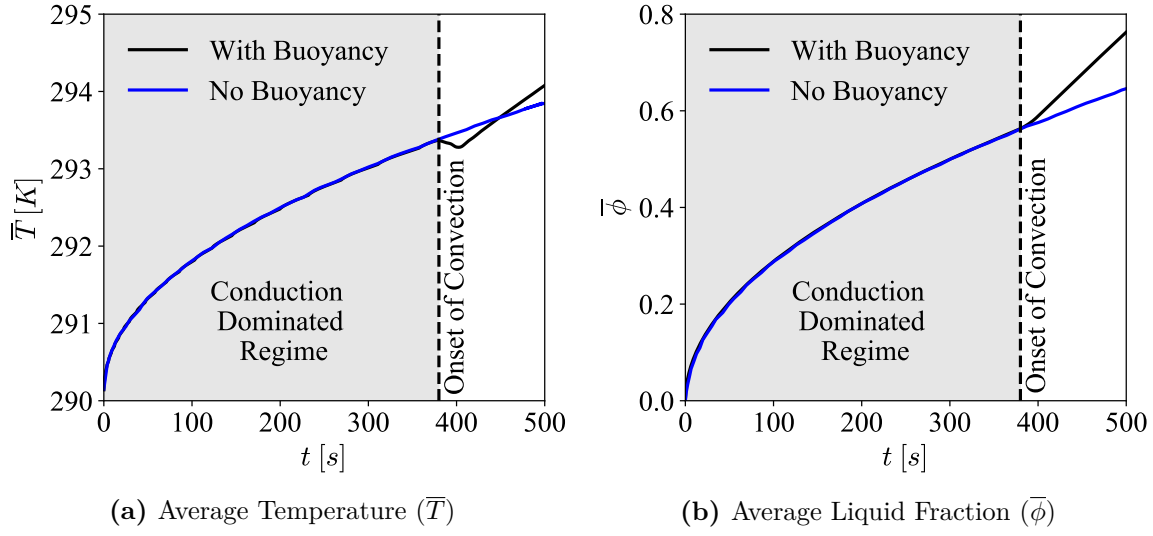


Figure 5.6: Effect of buoyancy on average properties.

5.3.2. Melting in a plate heat exchanger

Following this initial analysis, a melting cycle within a plate heat exchanger was studied based on the 2D geometry shown in Figure 5.7.

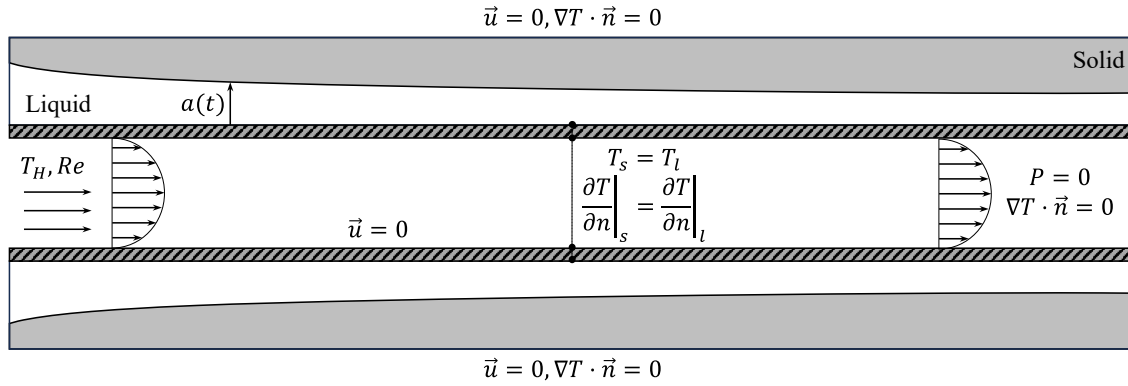


Figure 5.7: Geometry and Boundary conditions used for the plate heat exchanger. Note: not to scale.

5.3.2.1 Problem Setup

The geometry represents a 100 mm long layer of the heat exchanger where a 9.252 mm thick water layer is enveloped by two 2 mm thick layers of PCM (RT18 HC) separated by a 0.5 mm thick plate of aluminium. The thickness of the PCM was chosen so that more PCM was available in the channel than what could be melted in the melting cycle. The fluid enters normal to the inlet by assuming a hydraulically fully developed flow at $Re = 200$ with a constant temperature $T_H = 301.15$ K and the

Table 5.3: Material Properties used in compact heat exchanger simulations

Material Properties	RT18 HC	Water	Aluminium
Density (l) [kg/m ³]	770	998.2	2719
Viscosity [mPa.s]	-	1.003	-
Specific heat capacity [J/kgK]	2000	4182	871
Thermal conductivity [W/mK]	0.2	0.6	202.4
Heat Storage capacity [kJ/kg]	260	-	-
Melting Range [K]	290.15 - 292.15	-	-

flow leaves the heat exchanger through a pressure outlet. To define the hydraulically developed flow condition at the inlet, a velocity profile was obtained from a separate simulation with a larger plate length.

A summary of the material properties is given in Table 5.3. At the low Reynolds number ($Re < 2000$), the fluid flow is assumed to be laminar, while for $Re \geq 2000$, the $k-\omega$ *SST* model is used to capture the turbulence. The effect of the temperature drop is captured by using a conjugate heat transfer approach ensuring the continuity of heat flux and temperature at the walls separating the different zones. All walls are assumed to be no-slip and outer walls of the heat exchanger are insulated.

The transient solution of the melting starts from a predefined flow and temperature field where it is assumed that the fluid in the heat exchanger is pre-heated and fully developed. This field is obtained by finding the steady solution of the flow in which the PCM is maintained at a constant temperature $T_0 = 290.14$ K. Once an initial solution is obtained, a melting cycle of 100 s is modelled using a time step size of $\Delta t = 0.1$ s.

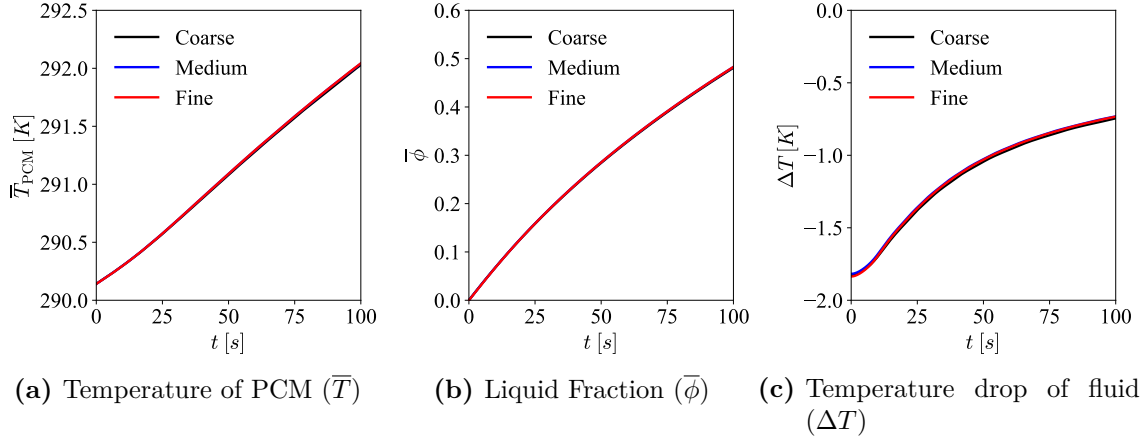
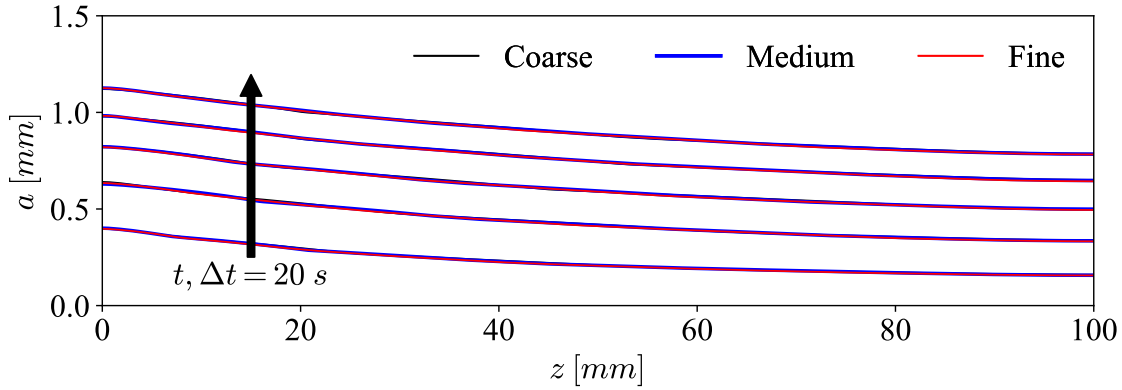
5.3.2.2 Mesh Independence

A mesh independence was conducted for the pre-heated fully developed case at a Reynolds number $Re = 200$. For the initial solution, bulk properties were tracked such as the maximum velocity (u_{max}), the temperature drop in the channel (ΔT) and the pressure drop (ΔP) in Table 5.4. This shows that mesh independence has been reached for the medium mesh.

Using the same mesh sizes, a mesh independence study was also conducted on the melting of the PCM. Figure 5.8 shows the variation of bulk properties over time for the different meshes and Figure 5.9 shows the change in the melting thickness over the length of the heat exchanger z over time. A good agreement is observed is

Table 5.4: Mesh Independence of the initial solution

Mesh	Size (k)	$u_{\max}$ [m/s]	% diff	ΔT [K]	% diff	ΔP [Pa]	% diff
Coarse	10.9	0.0156	-	-1.803	-	0.1380	-
Medium	16.2	0.0158	1.39%	-1.817	-0.81%	0.1385	0.30%
Fine	23.0	0.0157	1.20%	-1.836	-1.03%	0.1383	0.09%

**Figure 5.8:** Mesh Independence Study for plate heat exchanger: Average properties over time.**Figure 5.9:** Mesh Independence: melting thickness (a) over the length of the heat exchanger (z) over time.

observed for all meshes, with bulk properties and the position of the melting front over the length being accurately predicted.

Based on the results of the mesh independence study for the initial solution and the melting process, the medium mesh was selected for further consideration.

5.3.2.3 Description of the melting process

Figure 5.10 shows the melting thickness a and the normalized melting thickness a/a_{max} over the length of the heat exchanger z for different time steps at $Re = 200$. Here, a_{max} is given as the maximum melting thickness at a given time step, which in this case is located close to the inlet of the heat exchanger. A significant drop in the melting thickness over the length of the heat exchanger is shown, where at $t = 100\text{ s}$, the PCM melts around 25% less at the outlet than close to the inlet.

It can be seen that as time increases, the melting rate of the PCM decreases in line with what is observed for the bulk properties shown in Figure 5.8. This could be linked to the decrease in the temperature drop in the water channel ΔT observed in Figure 5.11, so that the temperature of the fluid is more uniform as time progresses. As the melting rate of PCM is highly dependent on the fluid temperature, this could lead to a higher melting rate further along the length of the heat exchanger at later time steps.

5.3.2.4 Effect of Reynolds number

In this section, the effect of increasing the Reynolds number on the melting behaviour of the PCM is studied for $200 \leq Re \leq 40,000$, where the Reynolds number was chosen based on an average inlet velocity ($\bar{u}$) and the hydraulic diameter (d_h):

$$Re = \frac{\rho \bar{u} d_h}{\mu} \quad (5.4)$$

The $k-\omega$ SST turbulence model was used to take effects of turbulence into account at higher Reynolds number $Re \geq 2000$. Care was taken to ensure that a $y^+ \approx 1$ was met in cases where the turbulence model was used.

Bulk properties are captured for different Reynolds number values in Figure 5.12, where the temperature drop in the fluid channel and the liquid fraction of the PCM is captured over time. This shows that as the Reynolds number increases, this leads to a significant improvement in the heat transfer performance of the heat exchanger, where the amount of heat stored in the PCM increases significantly. As such increasing the Reynolds number between 200 and 20,000 led to a 46% increase in the melt fraction. This is contrary to what was observed in multiple studies, where the flow rate was shown to have little impact on the melting rate of PCM. Figure 5.13 shows the position of the melt front for different melt fractions over time. The drastic increase in the liquid fraction at higher Reynolds number that

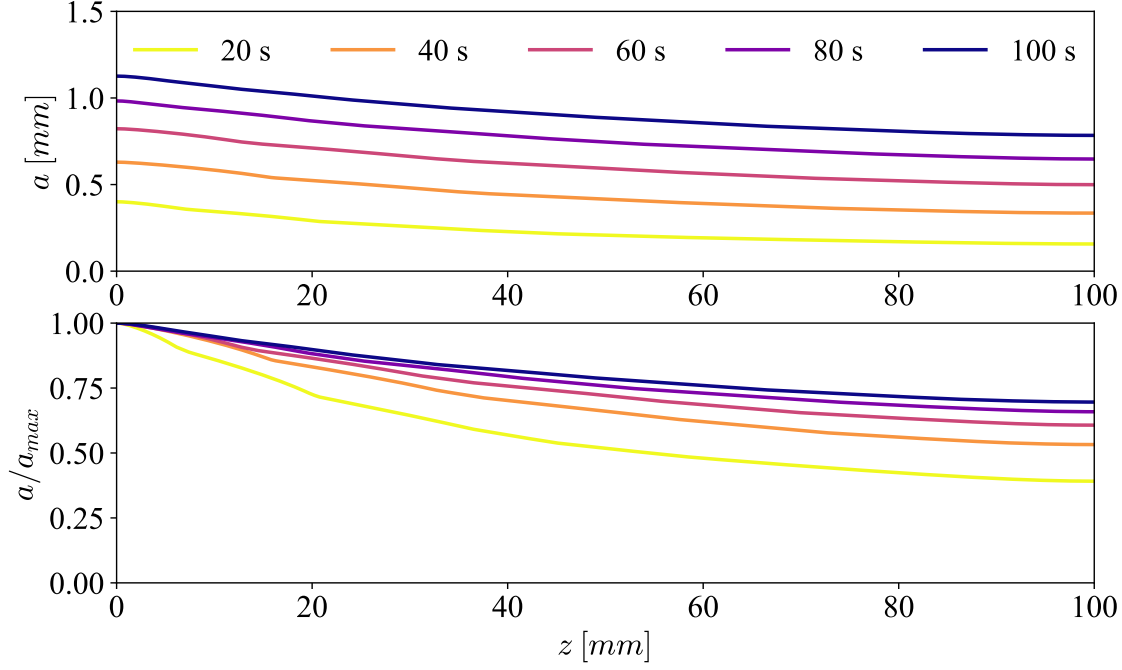


Figure 5.10: Melting thickness (a) and normalized melting thickness ($a/a_{\max}$) over the length of the heat exchanger (z) over time at $Re=200$.

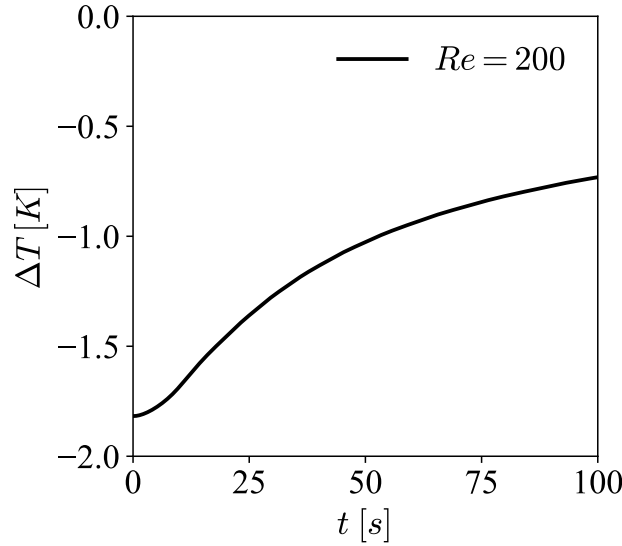


Figure 5.11: Temperature drop (ΔT) between the inlet and outlet of the water channel over time t at $Re=200$.

was previously observed is linked to the smaller drop in the amount of PCM that is melted over the length of the heat exchanger. Figure 5.12.a shows that as the Reynolds number increases, the temperature drop between the inlet and outlet is smaller, such that at $Re = 20000$, the temperature drop between the inlet and outlet is almost imperceptible. This leads to a more uniform temperature over the length of the heat exchanger, which helps in reducing the drop in the melting thickness.

Figure 5.14 summarizes the fraction of PCM that is melted at the end of the charging cycle over a wide range of Reynolds number. This shows a consistent increase in the amount of PCM that is melted over time, up to a Reynolds number in the order of 10^4 , at which point the amount of liquid PCM starts to slowly plateauing. This indicates that a relatively high Reynolds number is needed for optimally melting the PCM over the length of the heat exchanger. This is linked to the heat transfer between the fluid channel and the PCM increasing at higher Reynolds number. To gain a better understanding of the improvement of heat transfer from increasing the Reynolds number leading to an increased amount of PCM being melted at constant position, the thermal boundary layer thickness δ_t was captured in Figure 5.15, where the thermal boundary layer thickness was defined as the position where:

$$T(\delta_t, z) = 0.99T_B(z), \quad (5.5)$$

where the bulk temperature $T_B(z)$ was defined as the mass average temperature at each position z . This indicates a thinning of the thermal boundary layer as the

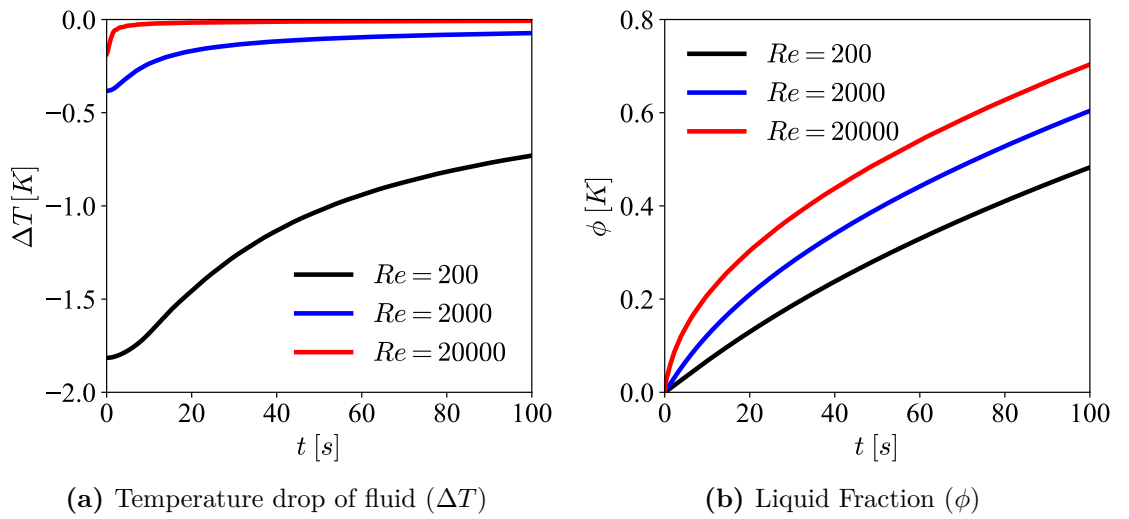


Figure 5.12: Effect of Reynolds number on bulk properties: (a) Temperature drop (ΔT) in fluid channel, (b) Average liquid fraction (ϕ) over time of PCM.

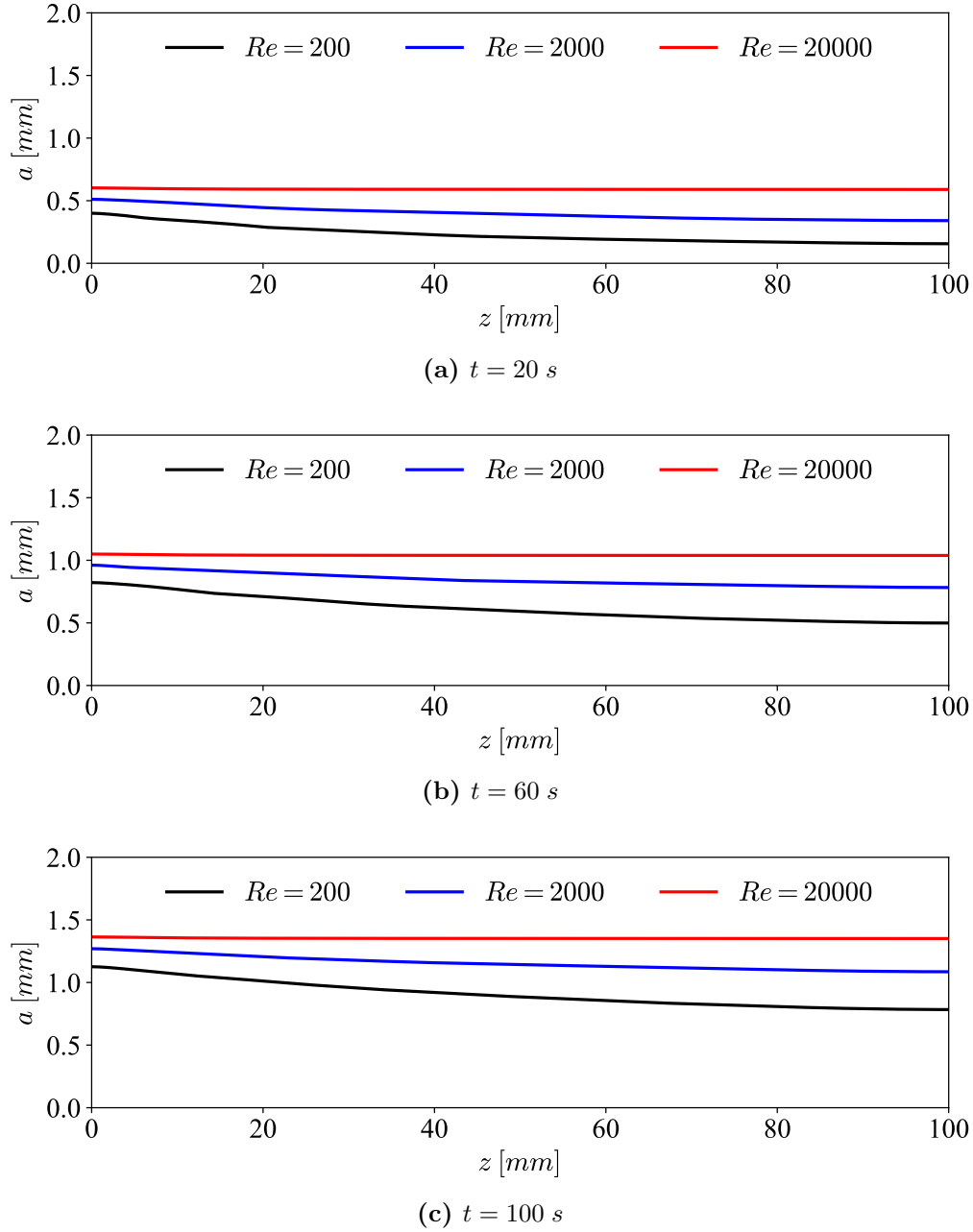


Figure 5.13: Position of the melt front (a) over the length of the heat exchanger (z) over time (t) for different Reynolds number values.

Reynolds number increases, leading to less resistance to the heat transfer to the PCM channel, such that the heat transfer between the fluid and the PCM is more efficient.

5.4. PCM in compact heat exchanger - plate fin geometry

The addition of fins can help in increasing the heat transfer in the heat exchanger by increasing the surface area in contact with the hot fluid layer. The addition of different fin geometries is studied in this section based on the dimensions of an offset strip fin geometry provided by [Kays & London \(1964\)](#). To ensure that all the physics are captured in the fluid layer, a single channel geometry model will be used to validate the fluid flow and heat transfer modelling methodology against experimental results from [Kays & London \(1964\)](#). Following this, the melting of PCM in a heat exchanger layer incorporating the offset strip fin and a continuous fin geometry will be modelled.

5.4.1. Validation of compact heat exchanger modeling

To validate the fluid flow and heat transfer modelling framework in complex geometries, CFD simulations of a single channel model were carried out by modeling

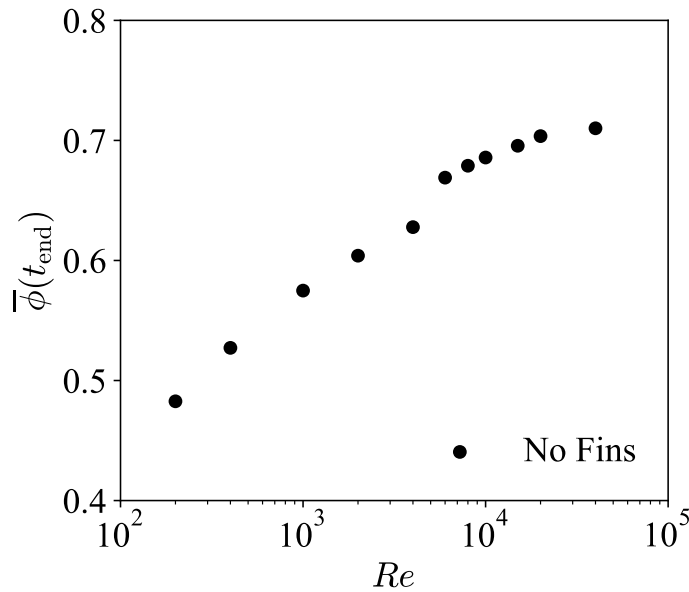


Figure 5.14: Total fraction of melted PCM ($\bar{\phi}$) at the end of the melting cycle (t_{end}).

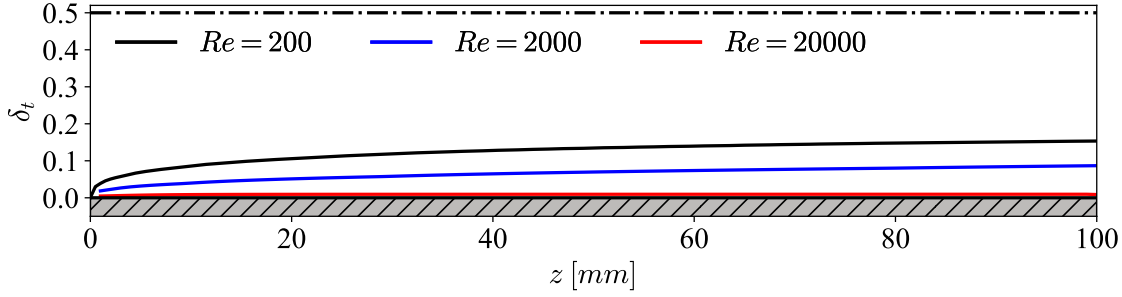


Figure 5.15: Thermal boundary layer thickness (δ_t) for different values of Reynolds value for $t=0$ s, normalized by the plate spacing.

Table 5.5: Fin properties of 1/8-13.95 offset strip fin geometry from [Kays & London \(1964\)](#)

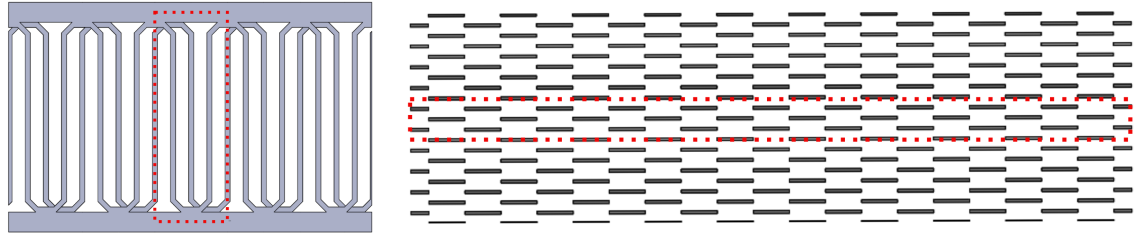
Fin properties	
Fin pitch [1/m]	549
Plate spacing, b [mm]	9.525
Fin length [mm]	3.175
Fin spacing [mm]	1.829
Hydraulic diameter [mm]	2.68
Fin metal thickness [mm]	0.254
Compactness β [m ² /m ³]	1250
Fin area/total area	0.84

the flow and heat transfer in a single channel geometry. This was validated against experimental results for an offset strip fin geometry by simulating the heat transfer and flow for Reynolds numbers between 400 and 4000.

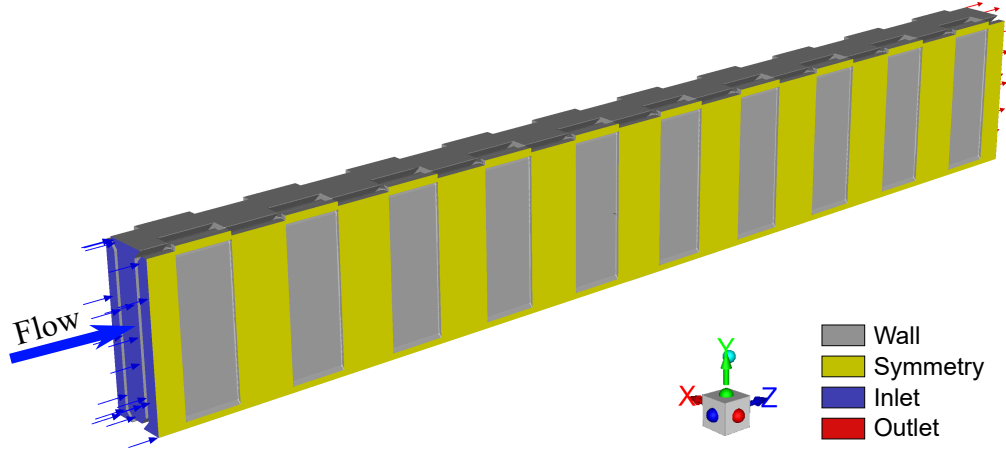
5.4.1.1 Single channel geometry

The strip fin 1/8-13.95 from [Kays & London \(1964\)](#) was chosen for this validation with dimensions summarized in Table 5.5. Figure 5.16 shows this geometry for 10 periodic channels, as well as the boundary condition used for this analysis. A single channel model was selected for the mesh independence and validation process, in which only one fluid side is modelled, by setting the walls to a constant temperature. For the purpose of validating the CFD, the finite channel consisted of 10 periodic lengths (≈ 60 mm) and water was used as the working fluid.

A temperature difference between the inlet and the walls of 10K was selected following the loading profile provided by the project partner, where the temperature at the walls was set to 291.15 K. The inlet was set to a uniform mass flow rate ($\dot{m}$) acting normal to the inlet, which was chosen based on the Reynolds number such



(a) Fin Geometry. Highlighted area shows the selected region for the channel flow domain.



(b) Single Channel flow domain with boundary conditions

Figure 5.16: Strip Fin geometry and flow domain for a single channel with 10 periodic flow lengths used for CFD analysis.

that:

$$Re = \frac{d_h \dot{m}}{A_c \mu}, \quad (5.6)$$

where A_c is the heat exchanger free-flow area, and d_h is the hydraulic diameter of the surface.

Coupled steady simulations with pseudo-transient time step approach was chosen for the process, with second order accurate schemes. The $k-\omega$ SST model was chosen for accounting for the turbulence over the range of Reynolds number that will be studied, as this was shown to give the best agreement with experimental results in Greiciunas (2019). Due to the choice of turbulence model, it was important to maintain a y^+ value close to 1. For the largest Reynolds number studied here, a y^+ value of 4.77 was measured for the medium mesh.

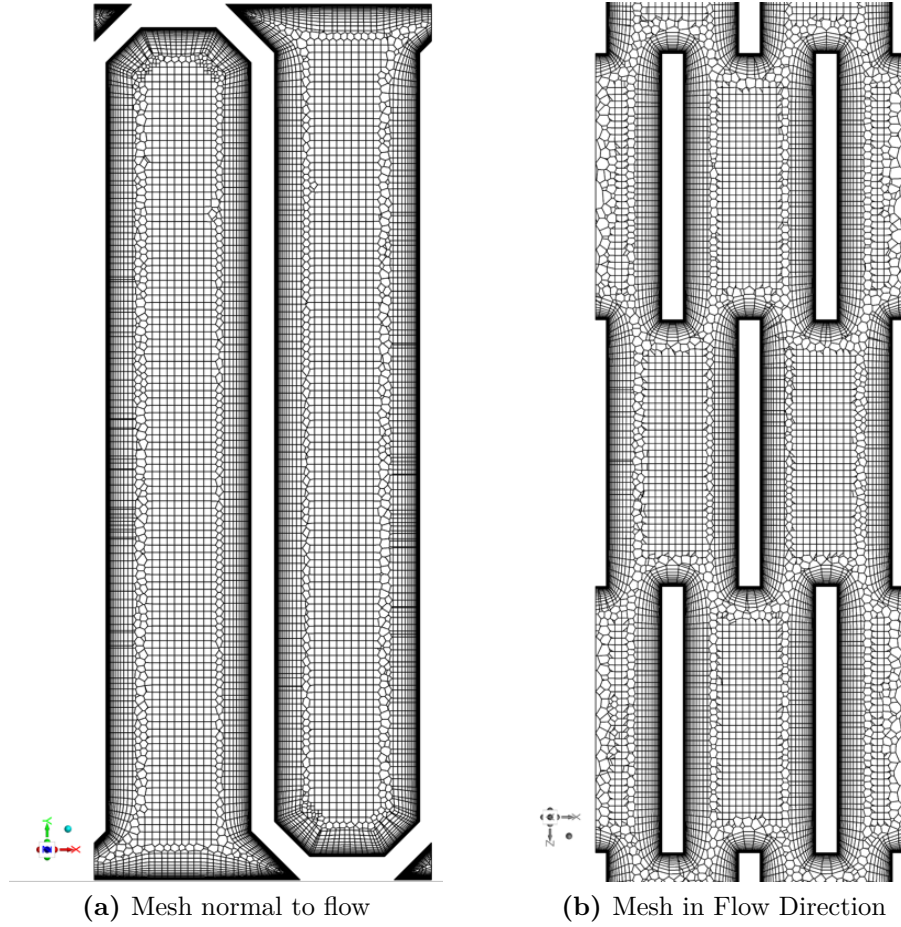


Figure 5.17: Picture of final mesh (Medium).

5.4.1.2 Mesh Independence Study

A mesh independence study was carried out at a Reynolds number of 4000 by measuring relevant bulk properties and by sampling the temperature and velocity magnitude along different lines in the domain. For this, four different poly-hexcore meshes were created in FLUENT Meshing, where the mesh resolution was consistently refined by a factor of 1.2, to ensure grid independence. On top of this, the value of the first layer thickness of the boundary layer was reduced to ensure that the viscous layer was fully resolved. A visualisation of the final medium mesh is shown in Figure 5.17.

Table 5.6 shows the temperature drop (ΔT) and the pressure drop (ΔP) over the length of the heat exchanger for the different meshes. The pressure drop was found to deviate more between the different meshes, while only small differences in the temperature drop were observed across the meshes. Negligible change in the temper-

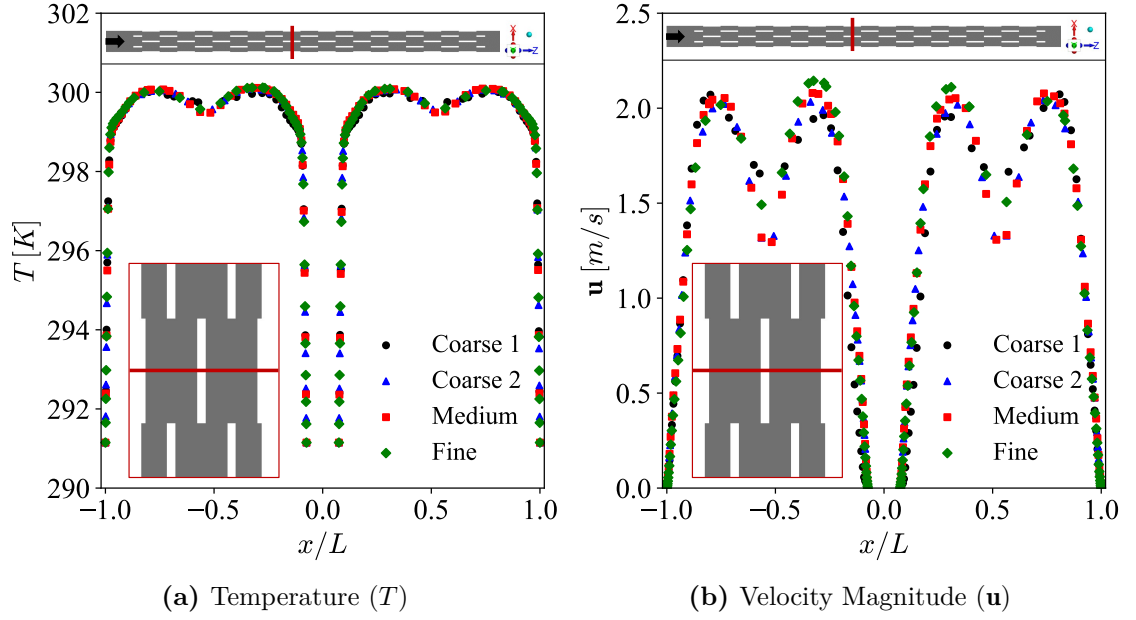


Figure 5.18: Comparison of Temperature (T) and Velocity Magnitude (u) along central line.

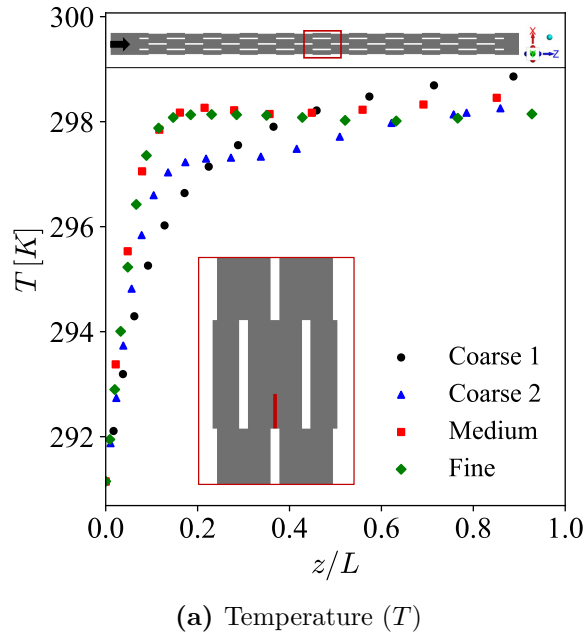


Figure 5.19: Boundary layer resolution in the wake of a fin in the middle of the domain

ature and pressure drop below 5% was observed between the Coarse 2 and Medium mesh, and the Medium and Fine mesh.

Furthermore, the temperature and velocity magnitude along a central line halfway along the channel was tracked for the different meshes (see Figure 5.18) and a further line was added in the vicinity of a fin in the flow direction in Figure 5.19, where z/L is the normalized distance from the fin in the $\hat{z}$ direction. Figure 5.18 shows little difference in the temperature and velocity magnitude along the line. However, when looking at Figure 5.19, it is possible to see that in the wake of the fin, a large difference in the temperature is visible for the coarse meshes when compared to the other meshes, so that only the medium mesh fully resolves the boundary layer in the wake of the flow. As a result, this mesh was chosen for the validation process.

5.4.1.3 Validation against experimental results

The simulation was run for different values of Re to compare against experimental results for the j and f factor from [Kays & London \(1964\)](#)'s database.

For this, the colburn j factor was evaluated from:

$$j = \frac{Nu}{RePr^{1/3}}, \quad (5.7)$$

where the Nusselt number Nu was calculated as:

$$Nu = \frac{\dot{m}c_p(T_o - T_i)D_h}{kA_h(LMTD)}, \quad (5.8)$$

with the logarithmic mean temperature difference ($LMTD$) being evaluated as:

$$LMTD = \frac{(T_w - T_o) - (T_w - T_i)}{\ln [(T_w - T_o)/(T_w - T_i)]}. \quad (5.9)$$

The fanning friction factor f was defined as:

Table 5.6: bulk properties measured as part of the mesh independence study: Temperature and Pressure Drop.

Mesh	Size (M cells)	1st layer [mm]	ΔT [K]	% diff.	ΔP [Pa]	% diff.
Coarse 1	5.03	4.0×10^{-4}	2.90	-	8395.4	-
Coarse 2	8.28	3.0×10^{-4}	2.88	0.69%	7486.6	11.44%
Medium	12.4	2.0×10^{-4}	2.91	1.04%	7145.4	4.66%
Fine	30.3	1.5×10^{-4}	2.91	0.07%	7127.8	0.25%

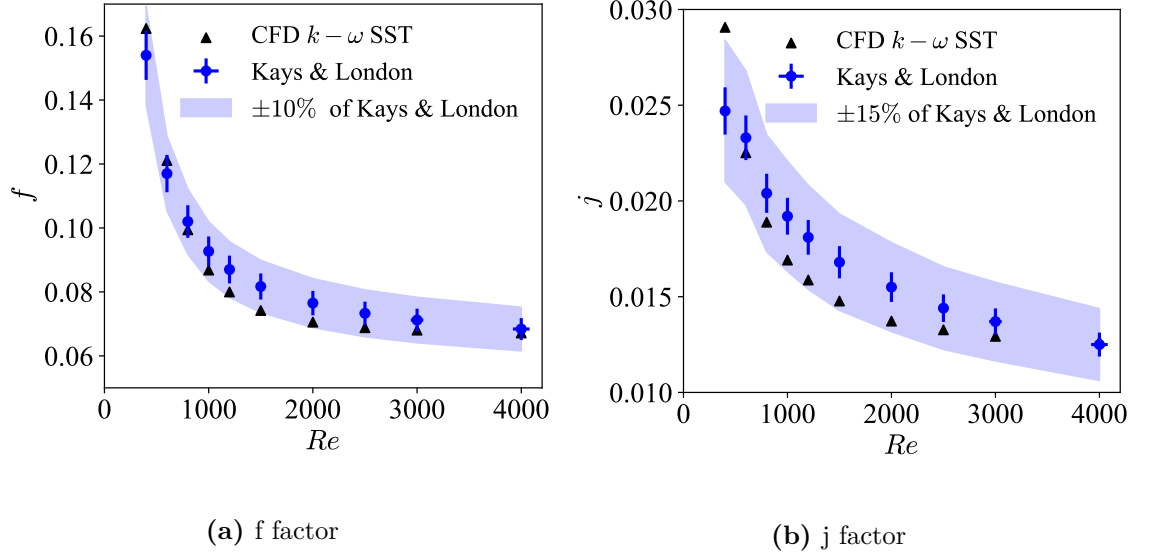


Figure 5.20: Comparison of j and f factor obtained using CFD with experimental results from Kays & London (1964).

$$f = \left(-\frac{dp}{dz} \right) \left(\frac{D_h}{2\rho u_m^2} \right) \approx \frac{\Delta P D_h}{2\rho u_m^2 \Delta z}, \quad (5.10)$$

where Δz is the length of the channel and ΔP is the pressure drop over the length.

The turbulent CFD results and experimental results are compared in Figure 5.20. Kays & London (1964) estimate that the experimental error is of the order of 5% for both the j and f factor and the error for the Reynolds number is around 2%, where this is visualized in the figure by using error bars. A good agreement between the CFD simulation results and the experimental results for the j and f factor is observed, where the CFD results tend to slightly underestimate the j and f factor predictions. For the f factor, CFD results were consistently within 10% of the experimental results. For the j factor, a greater deviation was observed, where the CFD results were within 15% of the simulation results, except for $Re = 400$, where a maximum deviation from the experimental results of 16.3% was measured. It is worth noting that the experimental point at $Re = 400$ does not seem to follow the general trend of the experimental results for the j factor and might have been mismeasured.

Some small differences in the geometry exist between the current CFD study and the experimental results from Kays & London (1964). In the current study, small simplifications were made to the fin geometry to reduce the computational cost of the simulation. Furthermore, the authors mention that there might be uncertainties related to the dimensions of the test core. Kays & London (1964) use steam-to-air

tests, which are susceptible to drop-off error of the j vs Re curve (see [Shah \(2003\)](#) for further information), which could explain the larger deviation of the first point at $Re = 400$. The material properties in the experiment will be temperature dependent, while in our case, water with a relatively small temperature difference between the inlet and outlet are used, where only a small variation in the material properties would be observed. Based on these considerations, the current methodology for modelling the flow and heat transfer in compact heat exchanger was deemed to be acceptable.

5.4.1.4 Turbulence Modelling

Similar simulations were carried out at low Re under 2000, by neglecting any turbulence effects (see red squares in Figure 5.22). This showed little difference up to $Re = 1000$ when compared to results which were run with the $k-\omega$ SST turbulence model enabled, suggesting that the flow is mostly laminar in the strip-fin channels at relatively low Reynolds number. Past $Re = 1000$, a larger deviation between laminar and turbulent results is visible, so that using a laminar approximation is no longer valid past this point.

Figure 5.21 compares contours of velocity magnitude and temperature over the length of the heat exchanger at $Re = 200$ for the laminar and $k-\omega$ SST models. This shows that the laminar model produces an almost identical flow and temperature field over the length of the heat exchanger at low Reynolds number, such that this simplification is valid for low Reynolds number flow.

5.4.1.5 Flow and Heat Transfer in stripped-fin compact heat exchanger

It is possible to see from Figure 5.21 that the fins create a symmetric flow and temperature field with repeating flow features at every periodic length. The entrance length is relatively short, such that the flow is almost fully developed past the second periodic length. This is further shown in Figure 5.23, where it is possible to see that past the first periodic length, the velocity profile does not change significantly.

Figure 5.24 compares the velocity field along the length of the heat exchanger at different values of the Reynolds number. This shows that at low Reynolds number, the flow separates into two jets at the tip of the fin and reattaches in its wake. For larger Reynolds number, the separation region is larger and two distinct jets are created on each side of the fin, where the jets stay separated in the wake of the fin.

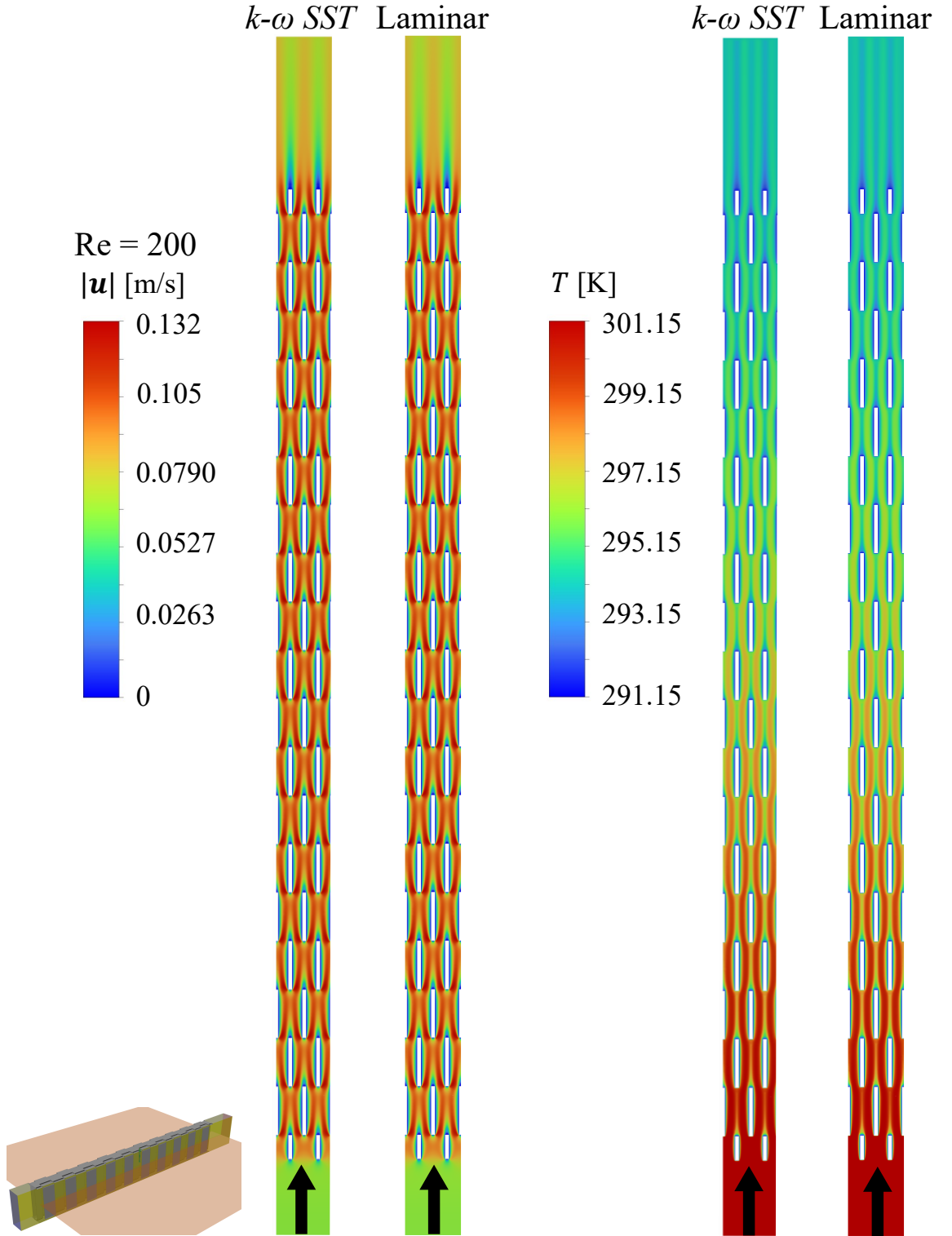


Figure 5.21: Contours of Velocity Magnitude (left) and Temperature (right) along the length of the water channel at $Re=200$ for laminar and $k-\omega$ SST models.

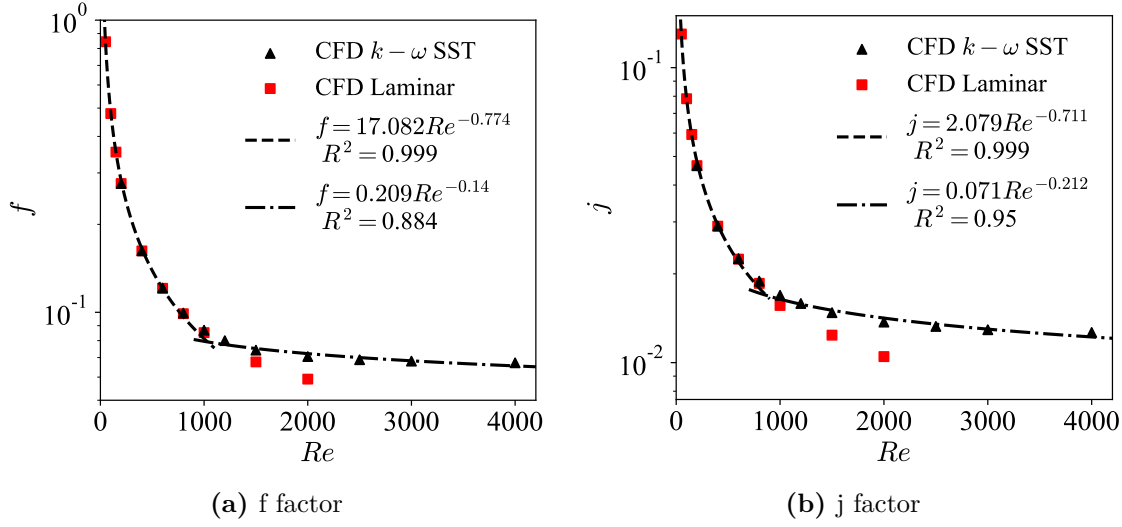


Figure 5.22: Effect of turbulence modelling on the j and f factor prediction.

5.4.2. Melting of PCM in offset strip fin compact heat exchanger

5.4.2.1 Problem Description

A PCM heat exchanger geometry with stripped fins was created from the finned fluid channel with 10 periodic lengths, that was validated in the previous section. For this the fluid channel was sandwiched in between two layers of RT18 HC PCM with a thickness of 2 mm, and two separating plates with a thickness of 0.5 mm. The geometry and boundary conditions used for this simulation are shown in Figure 5.25. The heat exchanger is subjected to a melting cycle of 100 s, where the inlet fluid temperature is set to $T_{in} = 301.15 \text{ K}$ to match the operating conditions from the loading profile.

A steady flow and temperature field was solved for the initial solution by maintaining the PCM temperature at 290.14 K. Figure 5.27 shows contours of velocity and temperature extracted from the initial condition. As can be seen from this Figure, the temperature drop over the length of the heat exchanger is lower ($\approx 4\text{K}$) than what was previously noted in the single fluid domain model used in the validation process. This is due to the effect of the walls, where the fin temperature changes over the length of the fin. The temperature is lower closer to the separating plate than at the bulk of the fluid, leading to a drop in efficiency in the heat transfer, commonly referred as fin efficiency (see [Kays & London \(1964\)](#) for more detail).

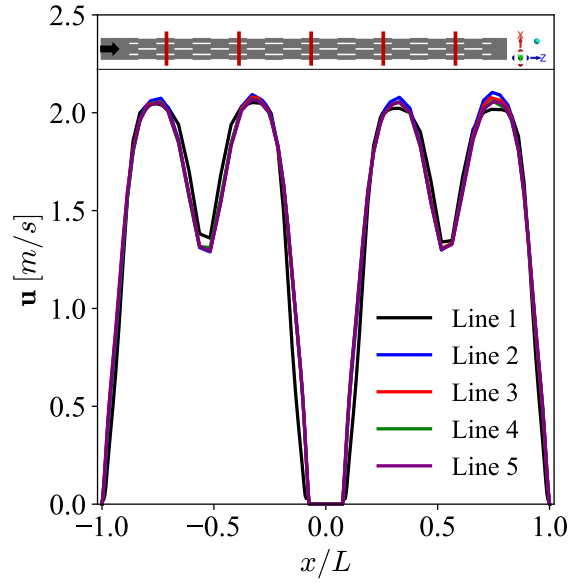


Figure 5.23: Velocity magnitude ($\mathbf{u}$) at different locations along the length of the strip-fin compact heat exchanger

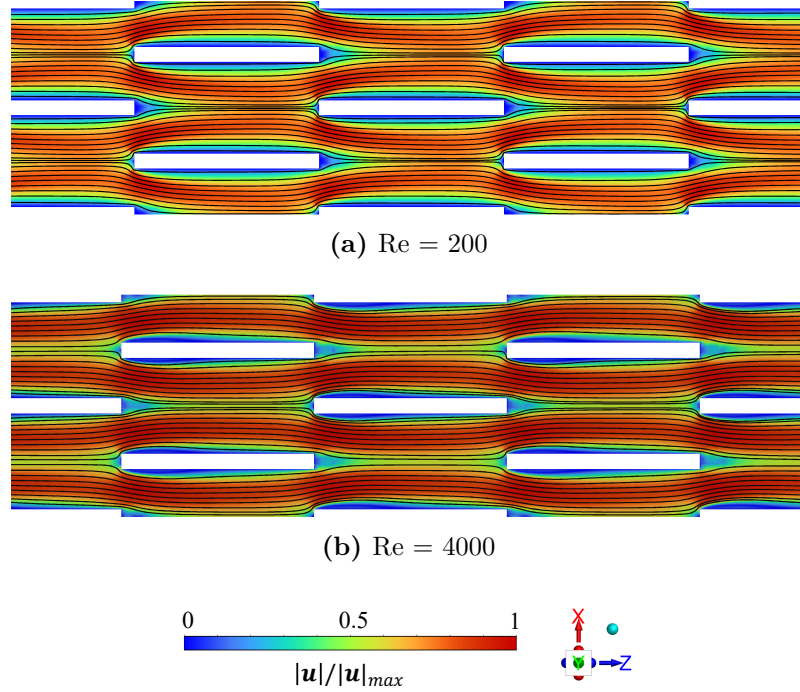
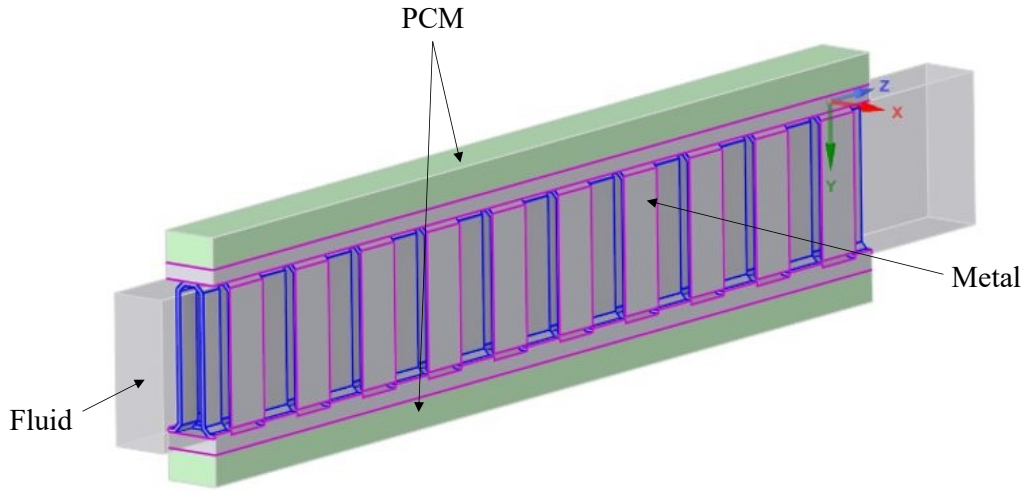
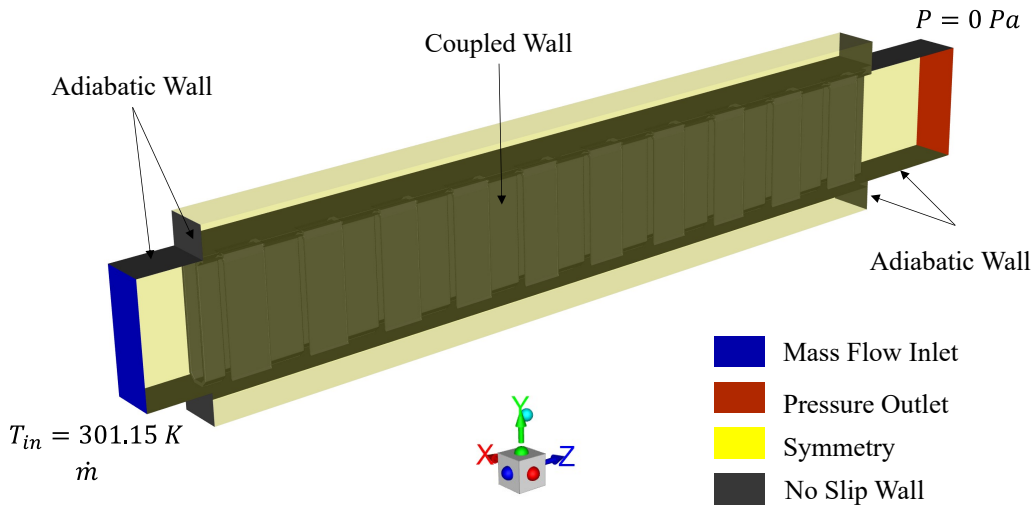


Figure 5.24: Contours of normalized velocity magnitude for different values of the Reynolds number for the strip-fin geometry. White line shows the average value of the velocity magnitude in the domain

Following the validation process, laminar flow was found to be sufficient to resolve the flow in the strip fin channel at low and moderate Reynolds number, such that this assumption was used for the remaining simulations. The effect of buoyancy on the melting of the PCM was neglected based on the findings in Section 5.3. As such, gravity was not modelled, making it possible to decouple the energy and flow equations, to reduce the computational cost of the simulation. The simulation uses similar schemes to previous simulations in this chapter.



(a) Geometry of PCM-Compact Heat Exchanger with stripped fins



(b) PCM-Compact Heat Exchanger domain with boundary conditions

Figure 5.25: Model of the PCM-Compact Heat Exchanger that will be used .

5.4.2.2 Mesh and Time Step Independence Study

A mesh independence analysis was conducted by considering 3 different meshes with a grid refinement ratio of 1.2. The initial mesh was based on the mesh used in the validation process, where the medium mesh was slightly coarsened in the boundary layer region, as smaller Reynolds number of 200 was considered (see Figure 5.26 for a picture of the final mesh). A time step of $\Delta t = 0.25s$ was selected for the mesh independence process, as this was found to be sufficient for the medium mesh size (see Figure 5.29). A good agreement is observed for the average temperature drop and liquid fraction over time, where little variation between the different meshes was observed for the average quantities in Figure 5.28.

Following this, the thickness of melted PCM was extracted at different time steps for the different meshes. This was achieved by creating an isosurface of the liquid fraction such that $\phi = 0.5$ on the $X = 0$ ZY plane. A general good agreement is

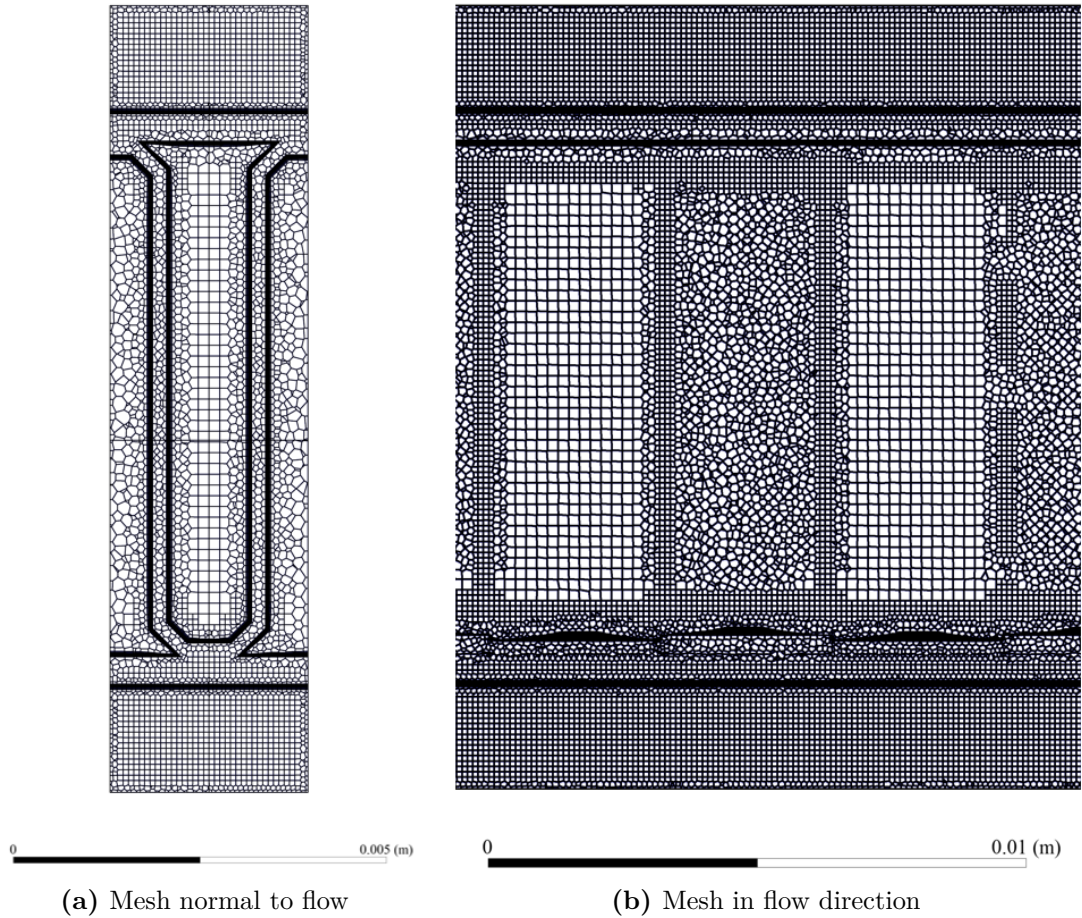


Figure 5.26: Medium mesh used for PCM-Compact Heat Exchanger simulations.

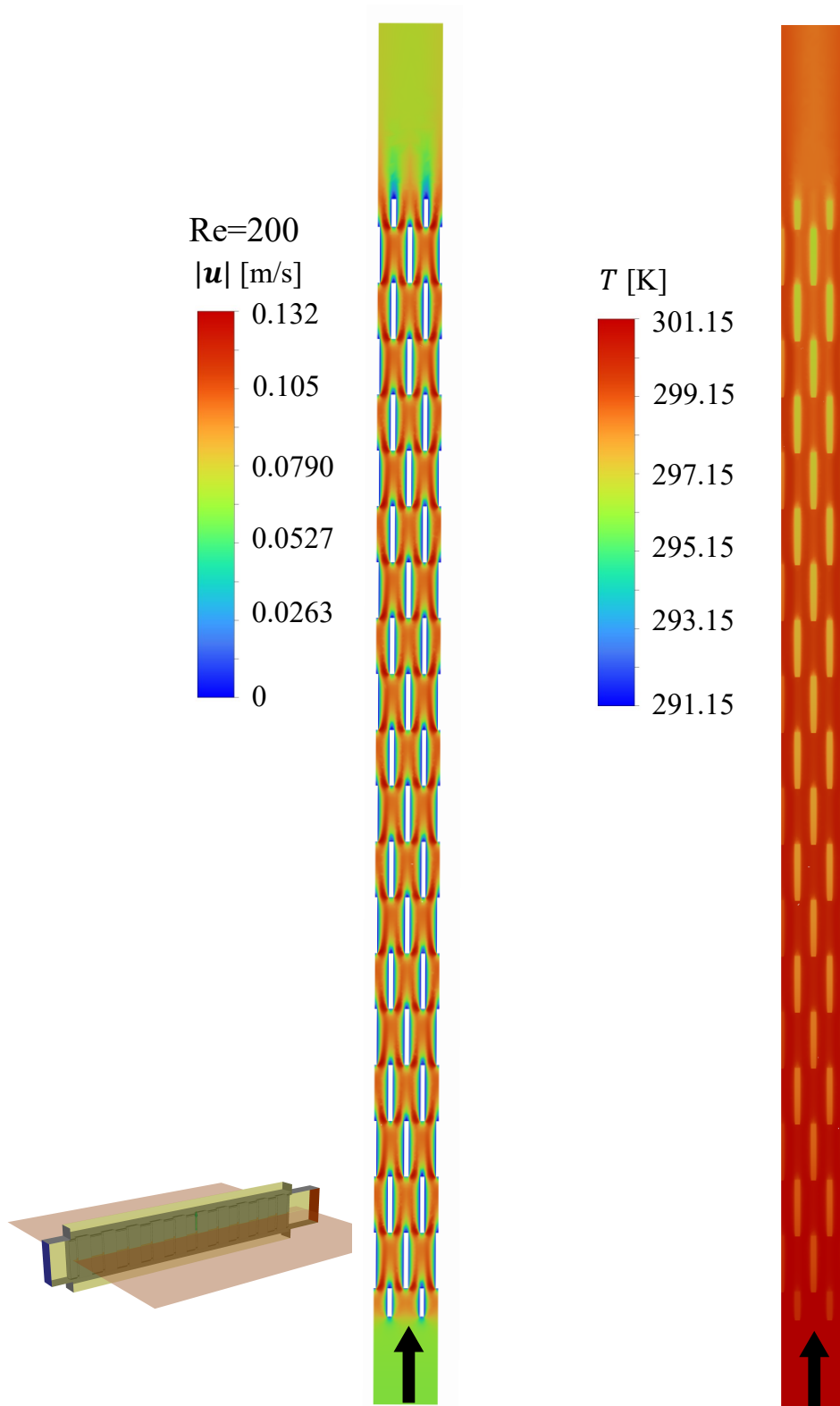


Figure 5.27: Initial flow and temperature profile in water channel: contours of velocity magnitude (left) and temperature (right) at $Re = 200$ for the strip fin geometry.

observed between the two meshes, but it is possible to see that the prediction of the melt front over time is closer for the medium and fine meshes. As a result, the medium mesh was selected.

5.4.2.3 Melting process

Figure 5.31 shows contours of temperature at the initial time (start of the melting cycle) and at $t = 100$ s at the interface between the metal wall and the PCM. This indicates that the temperature at the interface is more important directly above fins. As time progresses, as the solid/liquid interface moves further away from this wall, the heat diffuses more evenly so that smaller temperature gradients are observed at the wall.

Figure 5.32 shows the change in the thickness of melted PCM along the length z and the depth x of the heat exchanger at the end of the melting cycle. This shows that the the main direction in which the melting thickness varies is along the length of the heat exchanger, although this change is relatively small (≈ 0.05 mm). A larger decrease in the melting thickness over the length is observed close to the inlet than the outlet, linked to the more rapid drop in the fluid temperature at this location. Small variations in the melting thickness can also be observed directly above the locations of the fins where a higher plate temperature was recorded, although this is relatively negligible when compared to the total amount of melted PCM.

As the melting thickness mostly changes over the length of the heat exchanger,

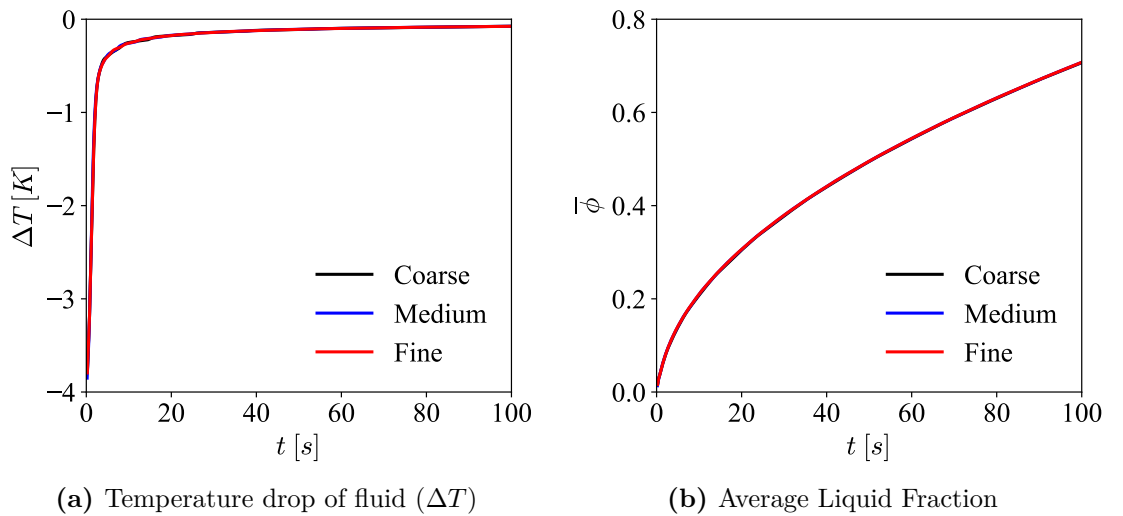


Figure 5.28: Mesh Independence Study : Variation of averaged properties over time for the strip-fin compact heat exchanger.

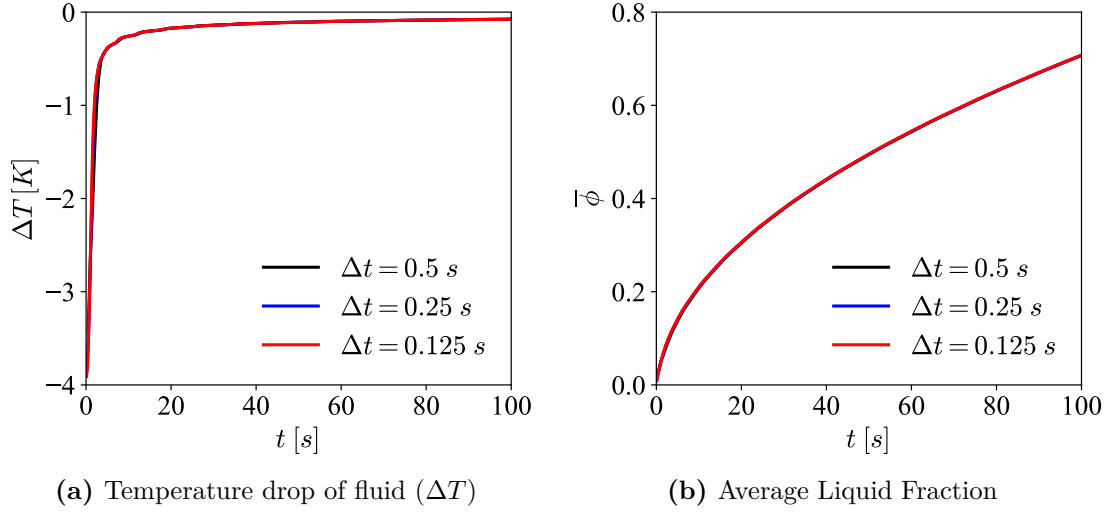


Figure 5.29: Time Independence Study: Variation of averaged properties over time for the strip-fin compact heat exchanger.

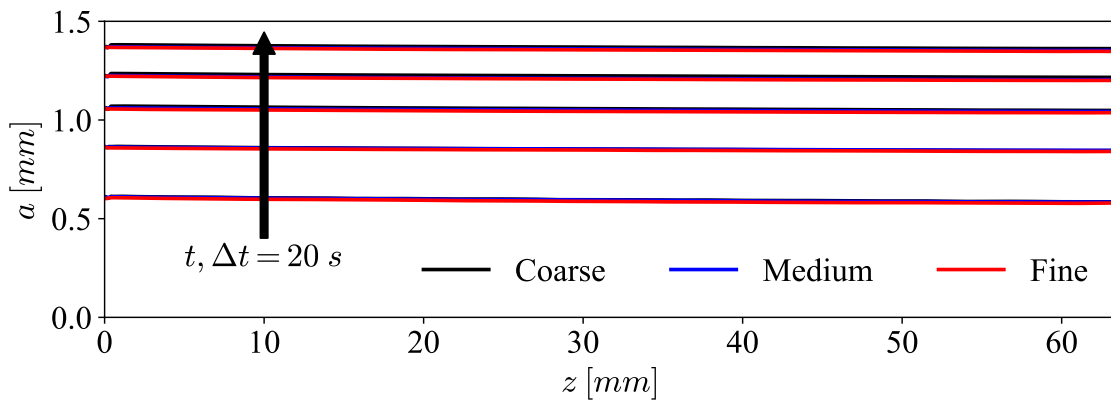


Figure 5.30: Mesh Independence Study: Variation in the melting thickness (a) over the length (z) of the strip-fin compact heat exchanger.

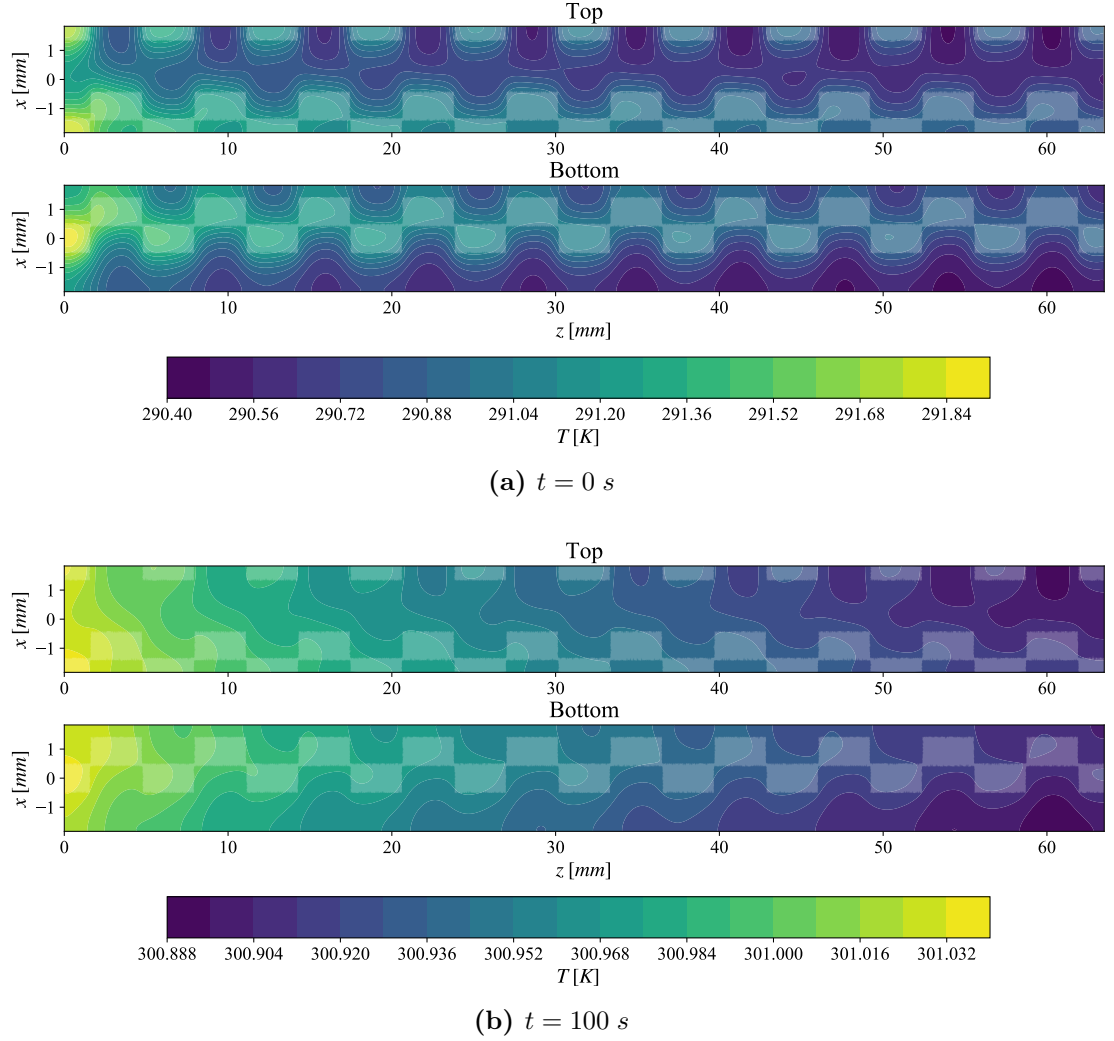


Figure 5.31: Change in temperature (T) along the length (z) and depth (x) of the strip-fin compact heat exchanger at the interface between PCM and metal plate. White shadow indicates the location of the fins underneath the PCM layer.

the melting thickness over the length at the $X = 0$ ZY plane was extracted over time in Figure 5.33. A very low drop in the melting thickness over the length of the heat exchanger is observed over time, even at relatively low Reynolds number ($Re = 200$). A sharper drop is observed at the early point in the melting cycle, while the melting becomes more uniform as time increases, where at the end of the melting cycle, $a_{\text{out}} = 0.99a_{\text{in}}$. This can be attributed to the sharp drop in the temperature difference between the inlet and outlet observed in the first 10 seconds of the melting cycle in Figure 5.28.a. This leads to a uniform bulk fluid temperature over the length of the heat exchanger for the majority of the melting cycle, such that the drop in the melting thickness becomes negligible.

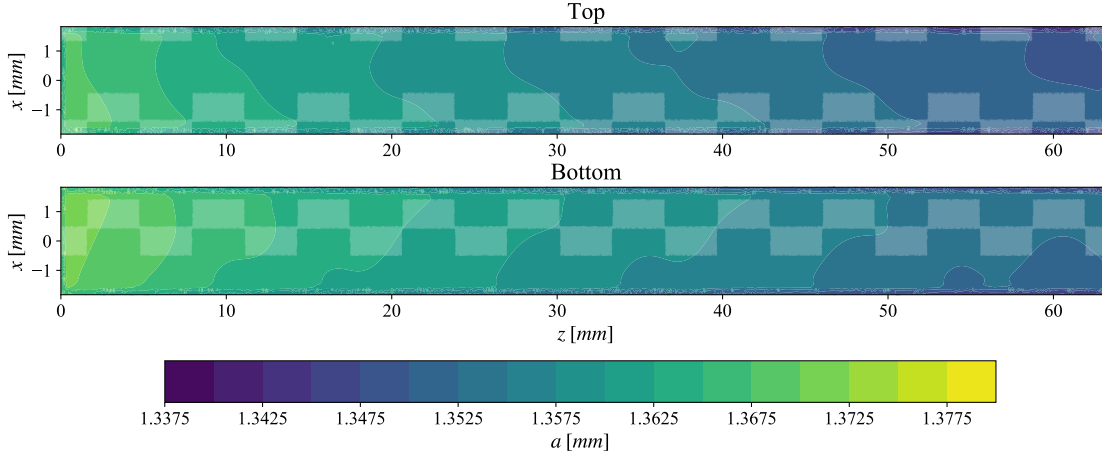


Figure 5.32: Change in the thickness of melted PCM along the length (z) and depth (x) of the strip-fin compact heat exchanger at $t = 100$ s. This was obtained by creating an isosurface of $\phi = 0.5$ and by extracting the position of this isosurface. White shadow indicates the location of the fins underneath the PCM layer.

The improvement in heat exchange is attributed to the addition of fins in the heat exchanger, that increase the heat transfer surface area and disturb the thermal

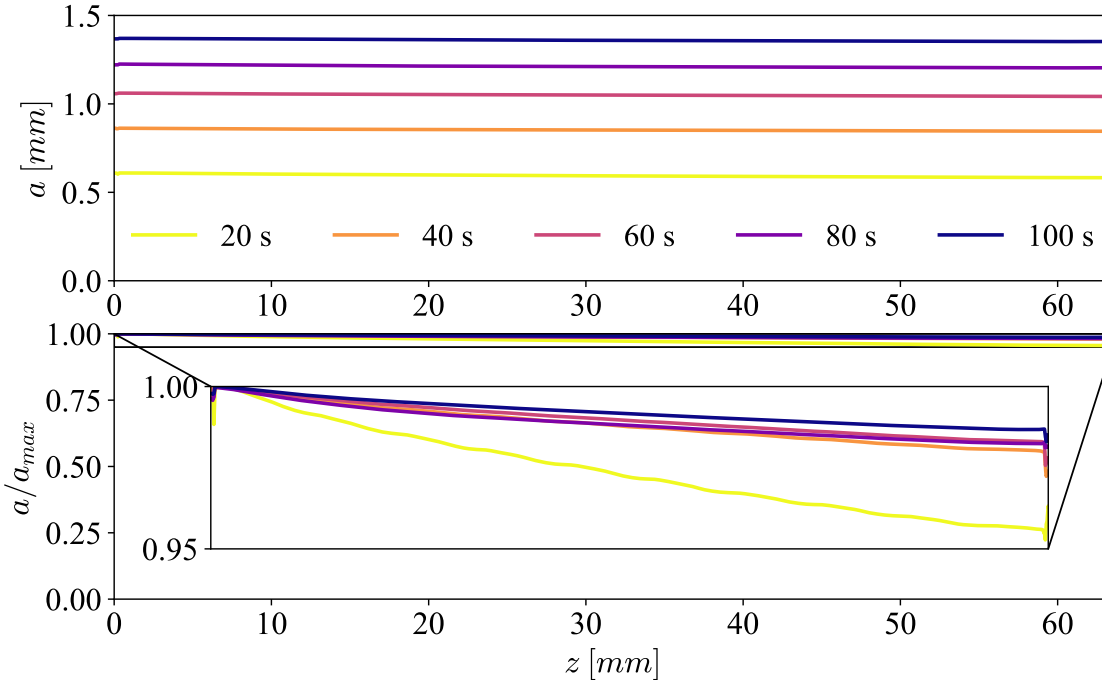


Figure 5.33: Variation in the melting thickness (a) and normalized melting thickness (a/a_{max}) over the length of strip-fin compact the heat exchanger (z) over time.

boundary layer as observed in Figure 5.27. As this happens, the thermal resistance of the PCM becomes more rapidly dominant, due to the low thermal conductivity of the material. To achieve similar results in the plain plate heat exchanger case considered in Section 5.3, a significant increase in the Reynolds number was necessary.

5.4.2.4 Effect of Reynolds number

Using the methodology described in the previous section, the Reynolds number was varied between 50 and 200 to observe the effect on the temperature drop and amount of energy stored in the PCM. The temperature drop in the water channel and liquid fraction have been plotted in Figure 5.34 for this range of Reynolds value. This shows that the initial temperature drop is more significant and the decrease in temperature drop in time is less pronounced at lower Reynolds number. It is also possible to observe that the effect of the increase in Reynolds number on the temperature drop rapidly decreases. Similar findings are observable for the liquid fraction, where an increase in the liquid fraction only produced a significant difference between $Re = 50$ and $Re = 100$. This indicates that as the Reynolds number increases, this does not lead to an increase in the amount of energy that is stored inside the PCM. This is in line with findings of [Shahsavari et al. \(2021\)](#), but counteracts what was found in Section 5.3, where a significant difference was observed for the plain plate fin heat exchanger.

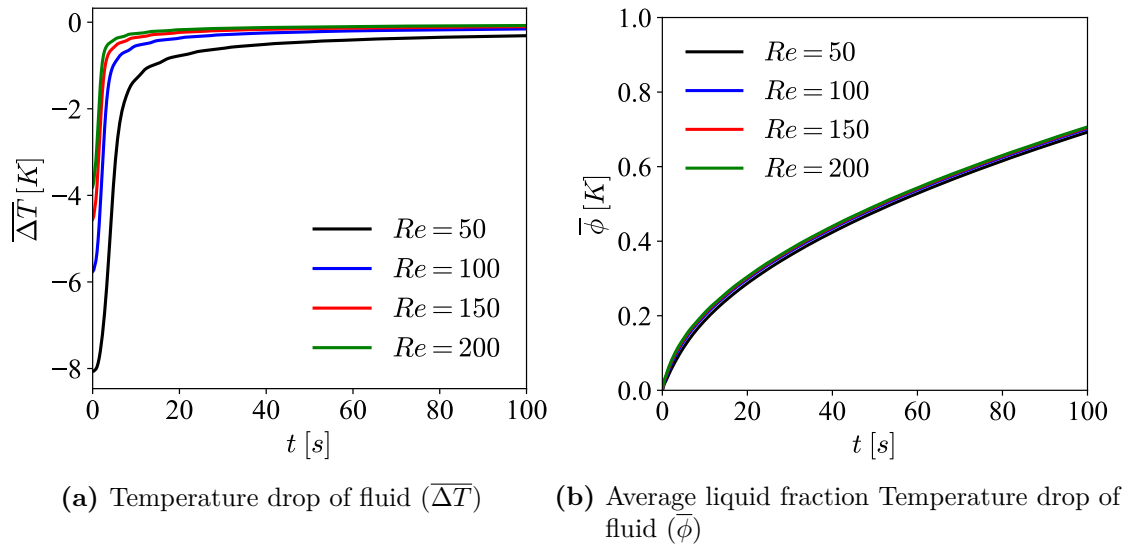


Figure 5.34: Variation of average properties over time for different values of the Reynolds value ($Re = [50, 100, 150, 200]$).

To inspect the effect of Reynolds number further, the change in melting thickness over the length of the heat exchanger has been plotted in Figure 5.35. Towards the inlet, the melting thickness is similar for all Reynolds number, although a slightly higher melting thickness is observed for higher Reynolds number, attributed to the decrease in the thermal resistance of the fluid layer at higher Reynolds number. Along the length of the heat exchanger, a lower Reynolds number leads to a more pronounced drop in the melting thickness, attributed to the smaller gradient of the temperature drop in the fluid channel at low Reynolds number. This suggests that differences in the melting of PCM for a range of Reynolds number will mostly be observed over the length of the heat exchanger for large heat exchangers, where the temperature drop will be more significant.

5.4.3. Melting of PCM in continuous fins compact heat exchanger

In the previous section, it was found that using stripped-fins for PCM-compact heat exchanger led to very uniform melting of the PCM over the length of the heat exchanger at very low Reynolds number. While the interruption of the fins leads to a well mixed flow, this also leads to an increased pressure drop when compared to continuous fins.

In this section a continuous fin geometry will be studied with similar dimensions to the offset strip fin studied in the previous section.

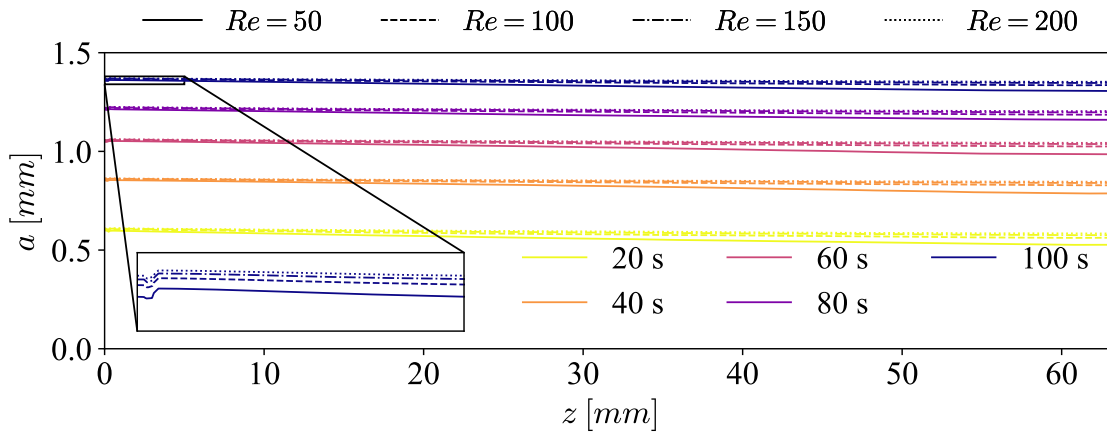


Figure 5.35: Variation in the melting thickness (a) over the length of the strip-fin compact heat exchanger (z) for different values of the Reynolds number ($Re = [50, 100, 150, 200]$) and time.

5.4.3.1 Problem Description

Figure 5.36 shows the geometry and boundary conditions used for modelling the melting of PCM. A mesh was generated based on the same settings used for the strip-fin PCM-compact heat exchanger, leading to a mesh size of 6.95M elements for the continuous fin heat exchanger.

A similar set up is used to what was previously described in this chapter. Similarly to the previous section, a steady state background flow and temperature profile was generated for the initial flow inside of the heat exchanger by maintaining the temperature of the PCM at a constant temperature of $T_0 = 290.14 \text{ K}$. An example of this is shown in Figure 5.37, where the contours of the velocity magnitude and temperature are represented at $\text{Re} = 200$. It is possible to see from the velocity

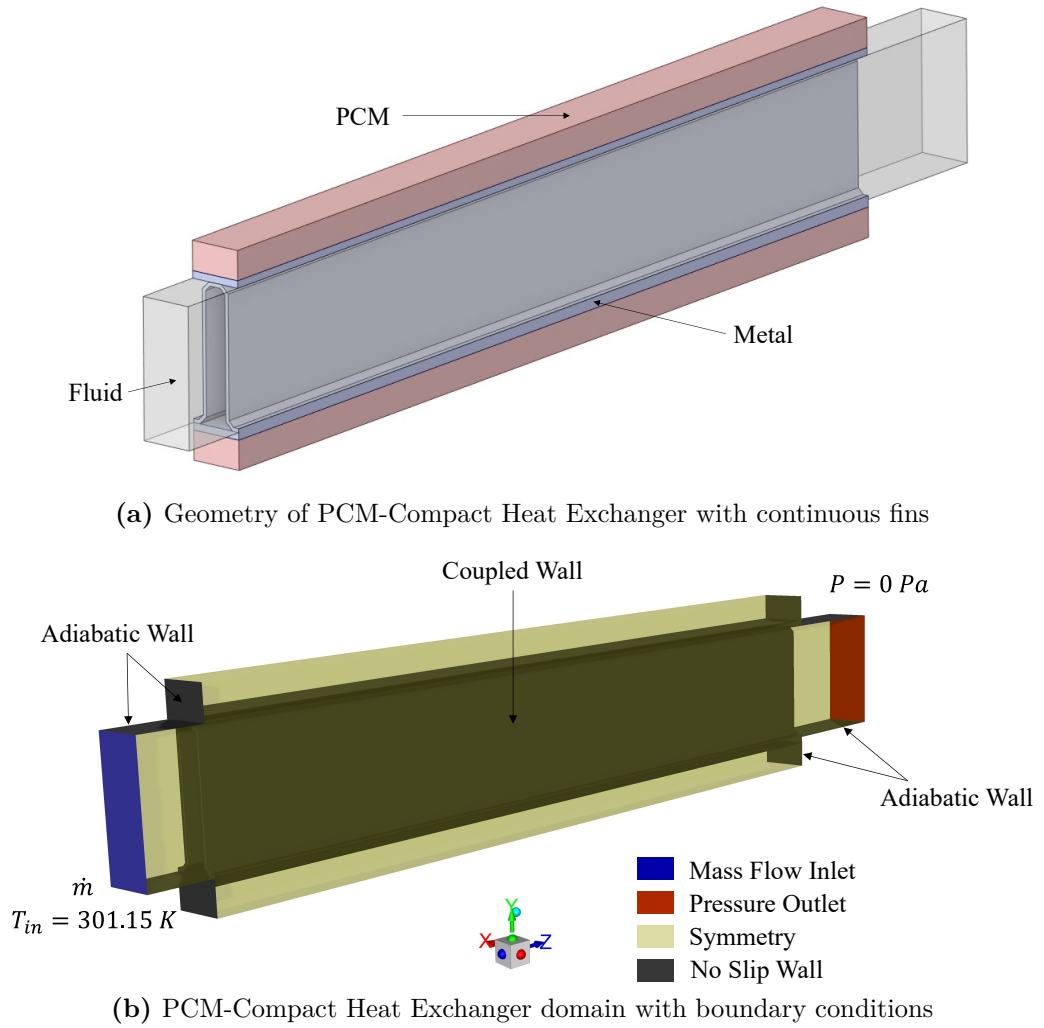


Figure 5.36: Model of the PCM-Compact Heat Exchanger with a continuous fin geometry.

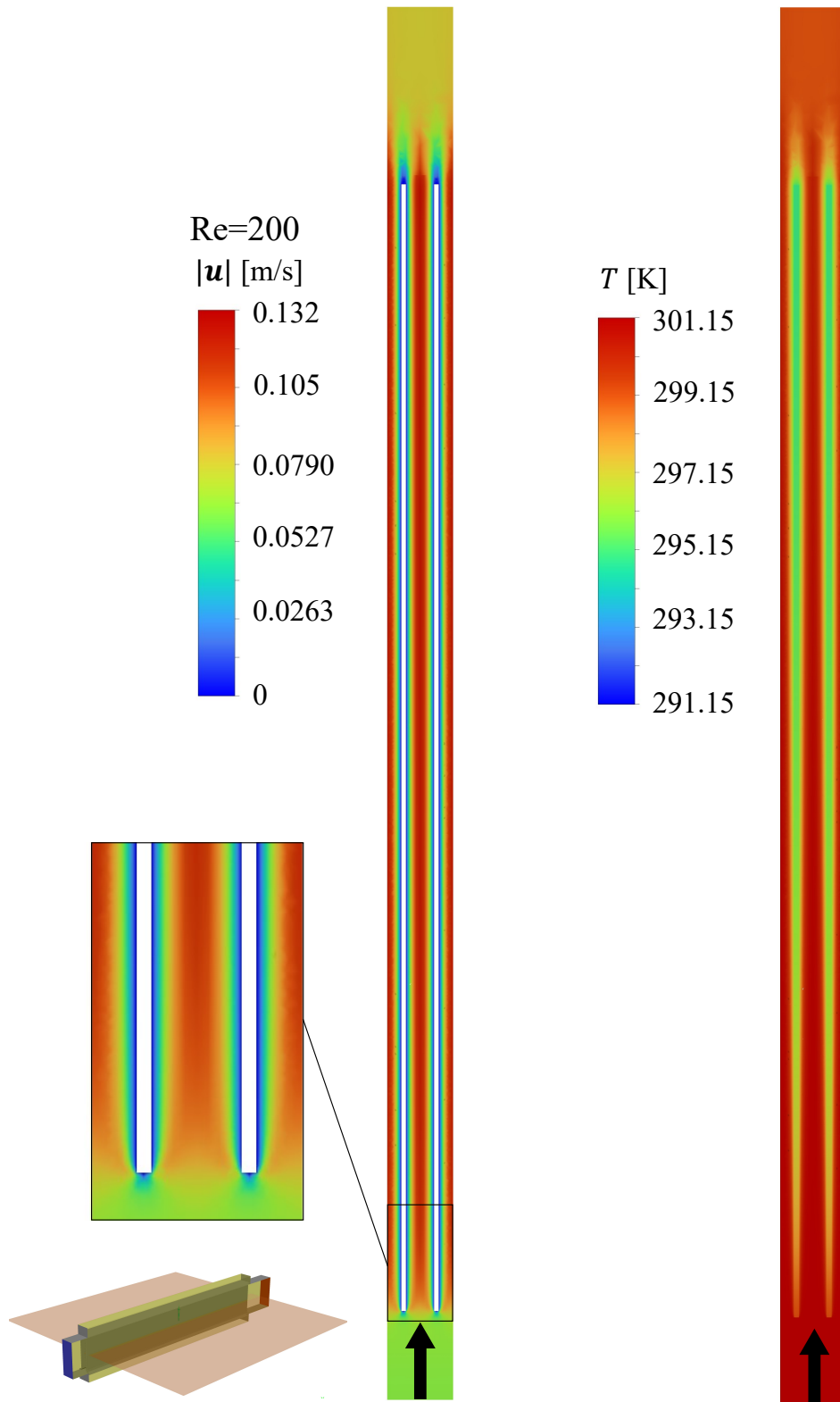


Figure 5.37: Initial flow and temperature profile in water channel: contours of velocity Magnitude (left) and Temperature (right) at $Re = 200$ for the continuous fin geometry.

contour that the flow develops over a relatively small length close to the inlet, creating a similar flow profile close to the fin walls to what can be observed for developing flow between two planes. An initial temperature drop of $\Delta T_0 = 2.8\text{ K}$ was measured between the inlet and outlet, lower than what was observed at similar Reynolds number for the stripped fin heat exchanger.

Following this, the melting of PCM is simulated by decoupling the energy and flow equations. The melt cycle is run for 100 s by using a time step $\Delta t = 0.25\text{ s}$, as was used for the stripped fin simulations.

5.4.3.2 Melting of PCM over length

Figure 5.39 represents the average wall temperature and the melting thickness a of the PCM over the length and depth of the heat exchanger at $t=100\text{ s}$. Similarly to what could be observed for the offset strip fin case, there are only small changes in the temperature and the melting thickness over the domain, so that the melting is relatively uniform. It can however be noticed that in places where the wall temperature is higher, i.e. where the metal fin is directly underneath the wall, this leads to a slightly higher melting thickness. This is attributed to the fact that the fin transmits

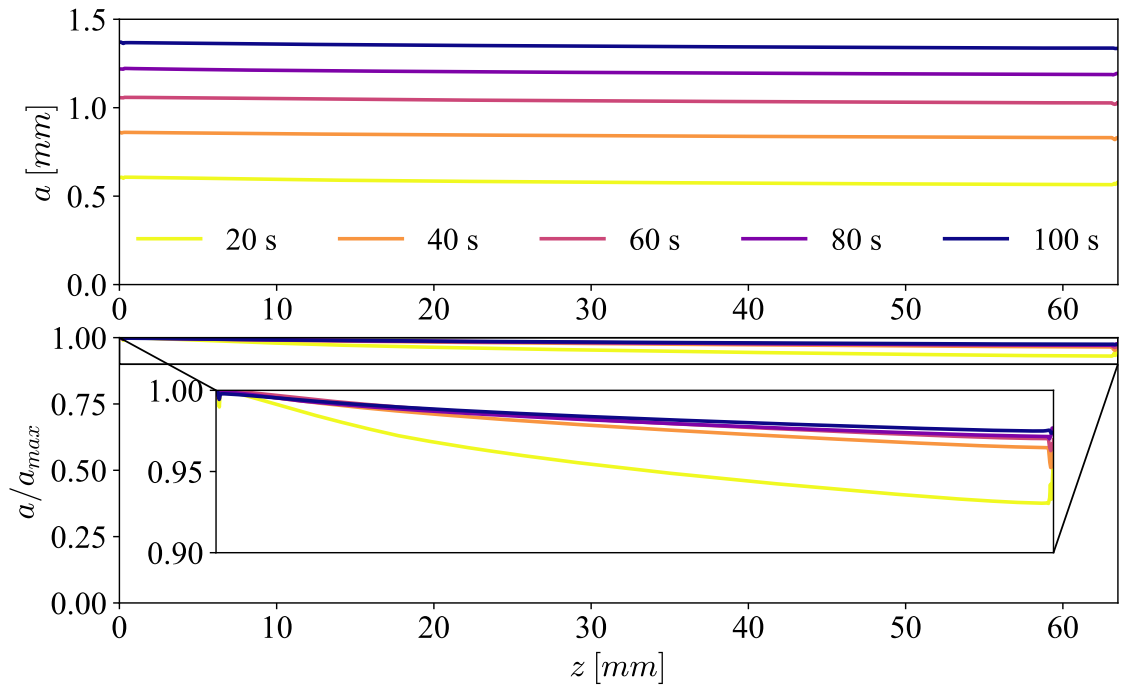


Figure 5.38: Variation in the melting thickness (a) and normalized melting thickness (a/a_{max}) over the length of the continuous fin compact heat exchanger (z) over time for continuous fin at $Re = 200$.

the heat from the fluid more effectively, due to a higher thermal diffusivity.

Figure 5.38 shows the drop in melting thickness over the length of the heat exchanger over time. The drop in melting thickness is relatively uniform through time, such that the melting thickness over the length of the heat exchanger is almost linear, similar to the strip fin case.

5.4.3.3 Effect of Reynolds number

Following this, the effect of Reynolds number was studied for the continuous fin heat exchanger. Averaged bulk properties are captured in Figure 5.40, while Figure 5.41 shows the effect of Reynolds number on the melting thickness over the length

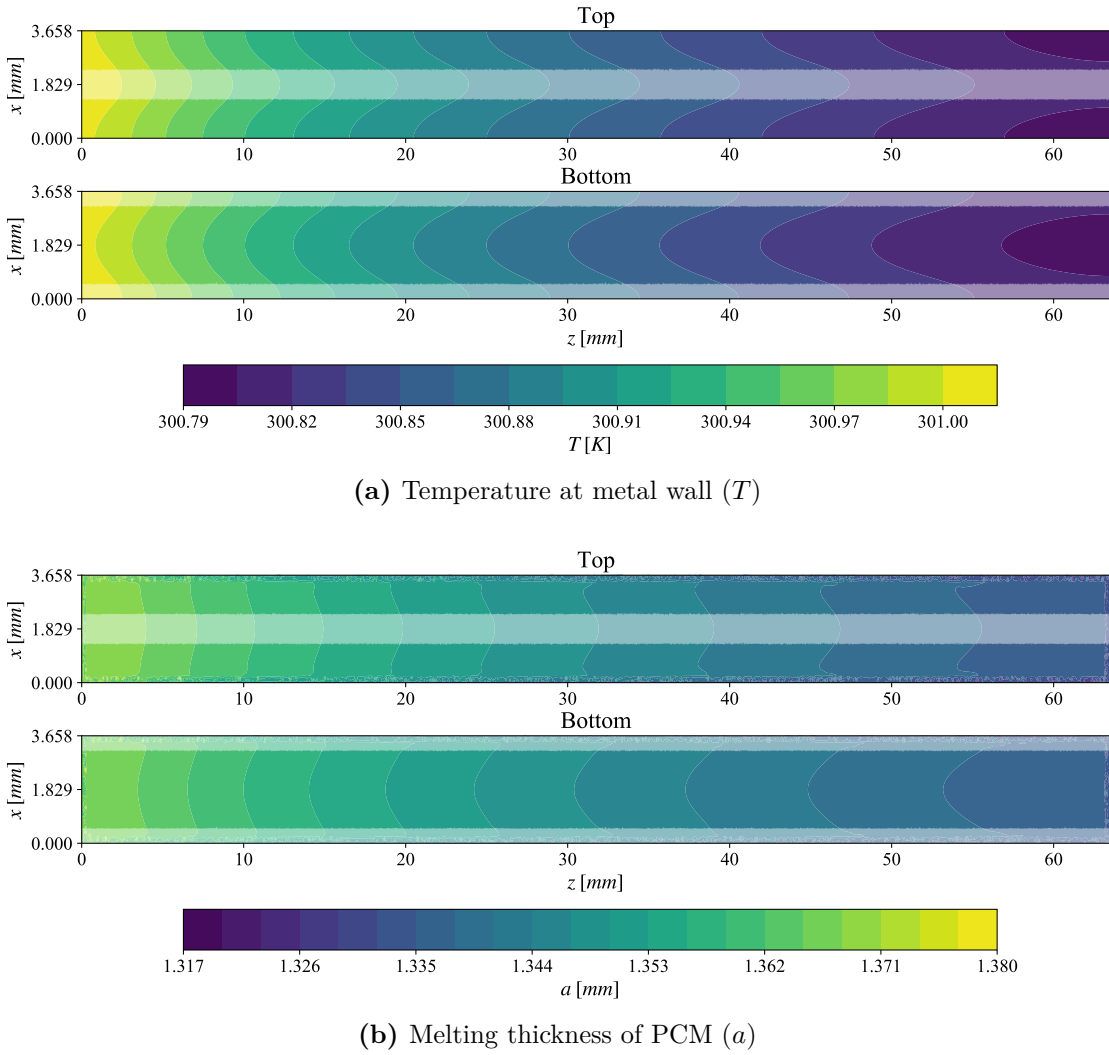


Figure 5.39: Variation along the depth (x) and length (z) of the continuous fin compact heat exchanger at $t = 100$ s. The white shaded area indicates the location of the fin in the fluid directly underneath the PCM.

of the heat exchanger. From Figure 5.40.b, it is possible to see that increasing the Reynolds number doesn't significantly increase the melt fraction of the PCM over time, but a more significant change in the temperature drop of the fluid channel is apparent in Figure 5.40.a. A larger temperature drop that decreases more slowly over time is observed at low Reynolds number. As the Reynolds number increases the change in temperature drop is less significant.

When looking at Figure 5.41, it is possible to see that the melting thickness varies little at the inlet of the heat exchanger over the range of studied Reynolds number. This difference increases over the length of the heat exchanger, so that at the outlet, a more perceptible difference is observed between $Re = 50$ and higher Reynolds number values, similarly to what was observed previously for the strip fin heat exchanger.

5.5. Effect of different fluid channel geometries

The three different finned surfaces that were introduced in this chapter (no fins, continuous fins and stripped fins) will be compared in this section. For this purpose, a plate spacing matching the 1/8-13.95 strip fin by Kays & London (1964) was selected for each heat exchanger design, leading to a consistent volume across the various designs. To simplify the comparison, a consistent mass flow rate was considered for all cases, where the mass flow rate matched the range of Reynolds

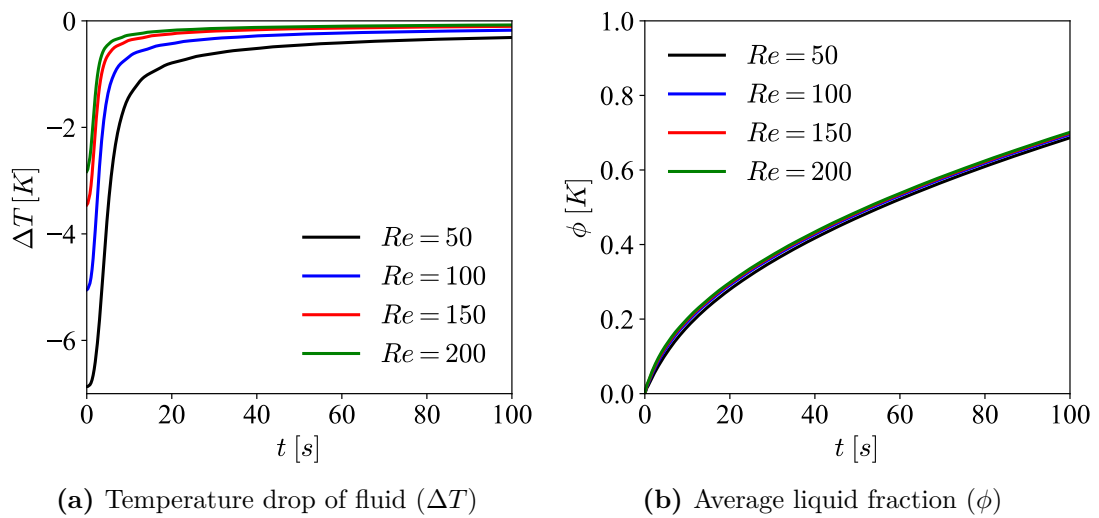


Figure 5.40: Effect of Reynolds number on bulk properties over time for a continuous fin geometry.

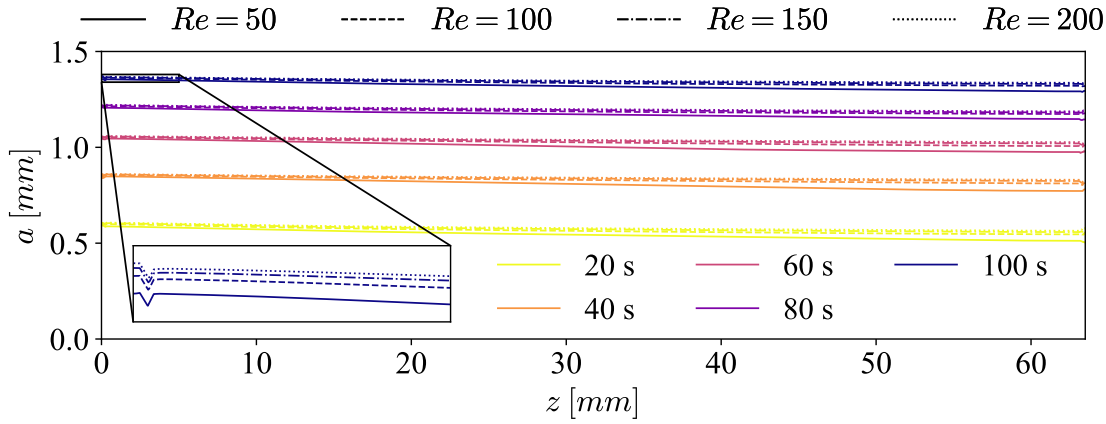


Figure 5.41: Variation in the melting thickness (a) over the length of the continuous compact heat exchanger (z) for different values of the Reynolds number ($Re = [50, 100, 150, 200]$) and time.

number studied for the offset strip fin and continuous fin cases, as shown in Table 5.7.

Table 5.7: Mass flow rate and equivalent Reynolds number for offset strip fin (OSF), continuous fin (CF) and No Fin (NF) case.

$\dot{m}$ [kg/s]	Re	
	OSF/CF	NF
5.51×10^{-4}	50	301
1.10×10^{-3}	100	601
1.65×10^{-3}	150	901
2.21×10^{-3}	200	1203

Figure 5.42 gives a summary of the dimensions of the different cases. These are similar to the ones used in the previous sections of this chapter for the offset strip fin (OSF) and continuous fins (CF) case, while a slightly shorter length was chosen for the no fins (NF) case. The no fins case was run as a 2D simulation to reduce the computational cost of the simulations, where the inlet velocity was chosen to match the mass flow rate based on the chosen dimensions.

5.5.1. Hydrothermal performance of finned and unfinned surface at initial condition

The hydrothermal performance of the different surfaces are compared in Figure 5.43 based on the effectiveness and the pressure drop over the length of the heat exchanger. The effectiveness shows the performance of a given heat exchanger and can

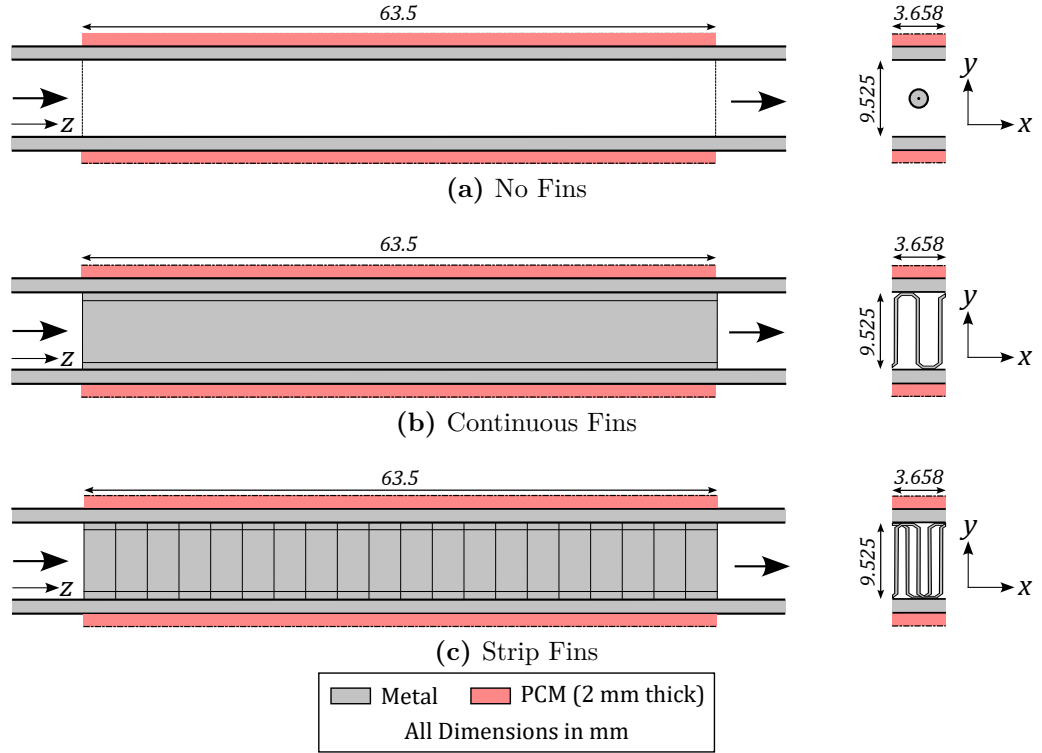


Figure 5.42: Geometry and Dimensions of the compared cases: (a) No fins, (b) Continuous Fins, (c) Strip Fins.

be defined for the PCM heat exchanger as the current heat flux over the maximum heat flux:

$$\epsilon = \frac{q}{q_{max}} = \frac{T_{in} - T_{out}}{T_{in} - T_{PC}}, \quad (5.11)$$

where T_{PC} is the phase change temperature and was chosen as the average between the solidus and liquidus temperature as 291.15 K. Note that the effectiveness will be proportional to the temperature drop in the fluid channel for phase change problems, such that the effectiveness will decrease over time. In Figure 5.43.a, the effectiveness is seen to increase significantly between finned and unfinned surfaces at constant mass flow rate, where the effectiveness is on average 8 times larger for the stripped fin case when compared with the non-finned case. Furthermore, the stripped fins lead to an improvement to the heat transfer when compared to the continuous fins, with an average improvement of 1.25 times in the measured effectiveness.

While the addition of fins leads to a significant increase in the effectiveness of the heat exchanger, it is worth noting that the use of fins is associated with a higher pressure drop. This is studied over the range of studied mass flow rates in Figure 5.43.b, where it is possible to see that the pressure drop is two orders of magnitude

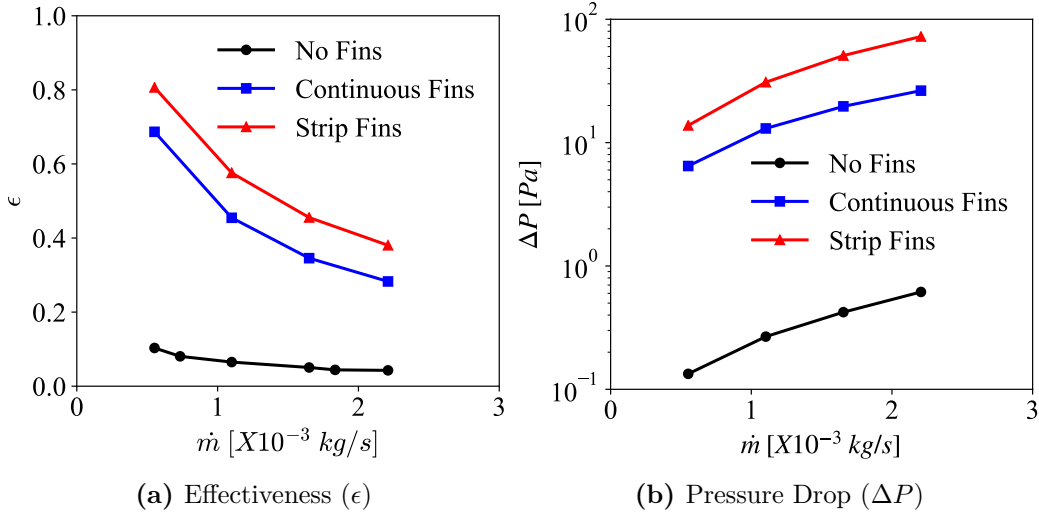


Figure 5.43: Hydrothermal performance of different heat transfer surfaces at the initial condition.

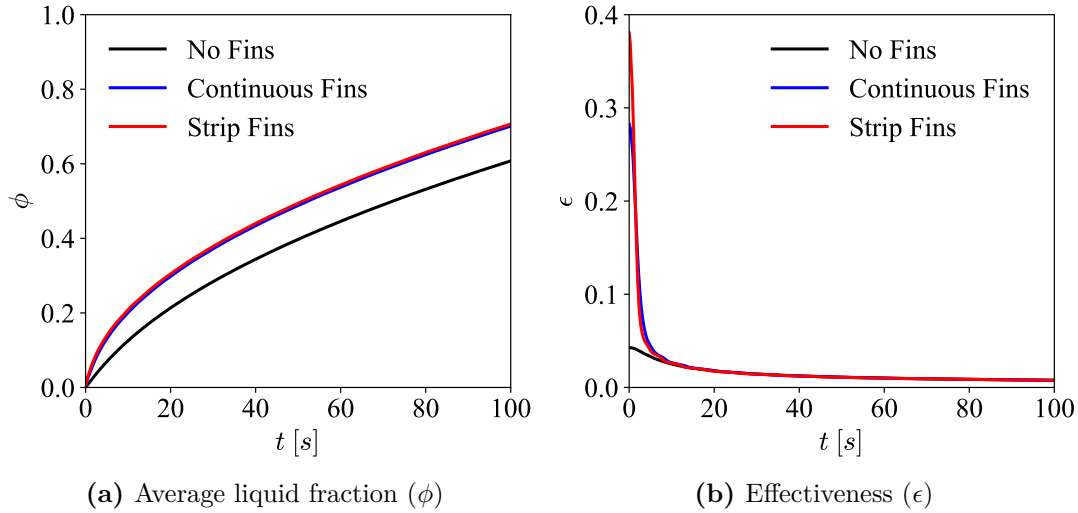


Figure 5.44: Effect of different finned surfaces on bulk properties at $\dot{m} = 2.21 \times 10^{-3}$ kg/s over time.

smaller for the non-finned case when compared to the strip fin case. The pressure drop for the continuous fins heat exchanger is 2-3 times smaller than the pressure drop compared to the strip fin case for the range of mass flow rates that were studied.

5.5.2. Energy Storage and Effectiveness during melting

Figure 5.44 compares the different finned surfaces based on the liquid fraction of the PCM and the effectiveness over time at the highest studied mass flow rate,

$\dot{m} = 2.21 \times 10^{-3}$ kg/s. These two plots show that the addition of fins lead to a significant improvement in the heat transfer and the melting rate of the PCM, where a 16.3% increase in the amount of melted PCM was observed at the end of the melting cycle for the stripped fin case. Looking at the effectiveness plot, this is most likely linked to the significantly lower initial effectiveness of the case without fins, leading to a smaller amount of melted PCM in the initial phase of the melting process. The effectiveness plot highlights, while there is a relatively high effectiveness at the beginning of the melting process, this significantly drops after the first couple of seconds as more of the PCM is melted. Notably, the effectiveness for the different surfaces becomes equivalent after this period.

The melting thickness over the length of the heat exchanger is shown for the same mass flow rate in Figure 5.45. In line with previous results, the melting thickness for continuous fin and strip fins are similar. It is worth noting that along the outlet, a more considerable drop in the melting thickness is observed for the continuous fin. This is likely to be linked to the lower effectiveness at the beginning of the simulation. The melting thickness for the No Fins case is lower than for the other cases and the drop over the length of the heat exchanger is more significant, strongly highlighting the advantage of using finned surfaces on the fluid side for mitigating the drop in melting thickness over the length of the heat exchanger.

To gain a better understanding of the effect of different surfaces on the general melting behaviour of the heat exchanger over the range of studied mass flow rates, a time averaged effectiveness ($\bar{\epsilon}$) was calculated as:

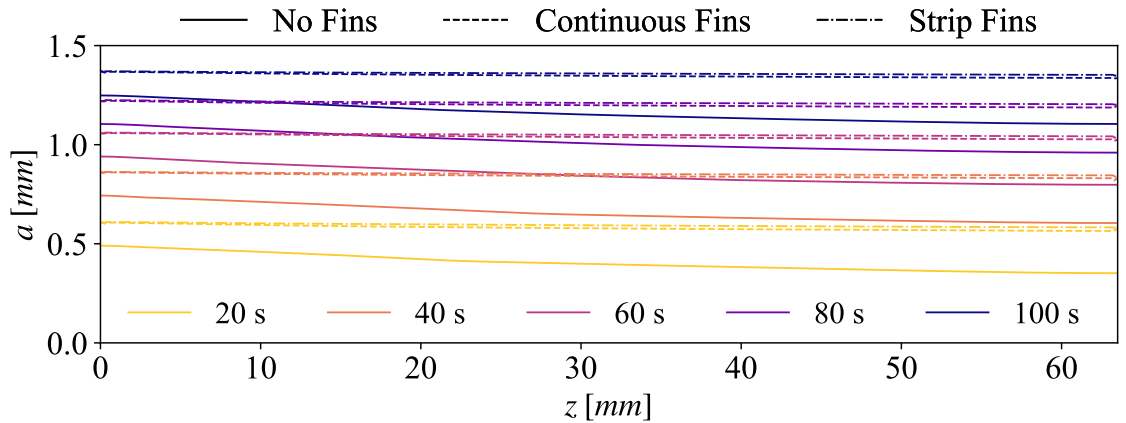


Figure 5.45: Variation in the melting thickness (a) over the length of the heat exchanger (z) over time for different fluid side fin geometries time at $\dot{m} = 2.21 \times 10^{-3}$ kg/s

$$\bar{\epsilon} = \frac{1}{T} \int_0^T \epsilon(t) dt \quad (5.12)$$

Additionally, the cumulative amount of energy that is stored in the PCM in the form of sensible and latent heat was calculated by assuming constant properties throughout the phases, similar to the simulation setup:

$$E_{PCM}(t) = m^P (c_p^P [T(t) - T_{init}] + \phi(t)L) , \quad (5.13)$$

where the superscript P indicates material properties of the PCM. Both these performance evaluation criteria are shown in Figure 5.46 for a range of mass flow rates. As can be seen from this, the effectiveness of the finned surfaces are almost equivalent by the end of the melting cycle, leading to an almost identical amount of energy being stored by the PCM. An increase in the mass flow rate is not shown to significantly increase the amount of stored energy for the finned cases. Compared to the finned cases, the non-finned case has a significantly lower effectiveness, leading to less energy being stored in the PCM. As the mass flow rate is increased for this case, a greater deviation in the amount of stored energy is observed.

Based on this analysis, the use of continuous fins is shown to outperform both the strip fin case and the no fins case, as a similar amount of heat can be stored in the PCM to the strip fin case, but at a significantly lower pressure drop. This means that a similar heat transfer performance can be obtained at considerably lower pumping power.

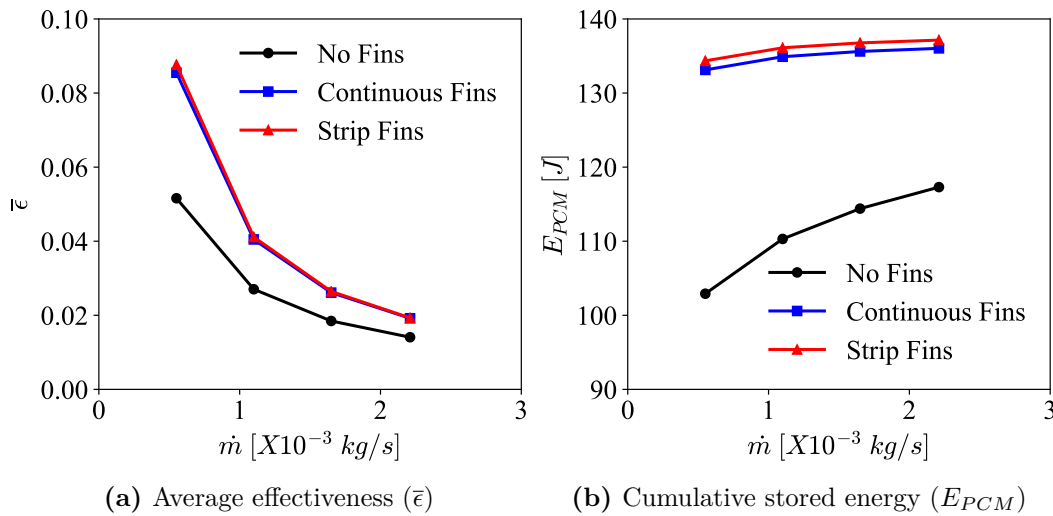


Figure 5.46: Overall thermal performance comparison of different finned surfaces over range of mass flow rates after $t = 100s$.

5.6. Summary

In this chapter, the effect of different fluid channel geometries on the melting of PCM in adjacent unfinned channels was studied. By applying the thermal load profile provided by the project partner, which is characterised by a small melting cycle and low difference between the PCM melting temperature and the fluid temperature, it could be observed that the PCM would remain in the conduction dominated regime throughout the melting cycle.

The melting of PCM over the length of plain plate heat exchanger was studied over a large range of Reynolds number ($200 \leq Re \leq 40000$). This showed that the amount of melted PCM was greatly dependent on the Reynolds number, where an increase in Reynolds number led to an overall increase in the melted PCM. The increase in Reynolds number was shown to not only melt more PCM close to the inlet, but also contributed to significantly decreasing the drop in melted PCM over the length of the heat exchanger, due to a more uniform fluid bulk temperature.

Following this, two finned fluid channels were analysed. Both the offset strip fin and continuous fins were found to be very efficient at transferring the heat to the PCM, leading to a similar fraction of melted PCM, where a small drop in the melting thickness was observed over the range of the studied Reynolds number ($50 \leq Re \leq 200$).

When comparing the heat exchanger incorporating the different fluid channel geometries, it was found that the continuous fin case outperformed the other studied fluid geometries. This is due to the similar amount of heat that could be stored in the PCM to the finned case, at 2-3 times lower pressure drop, greatly reducing the operating costs of the heat exchanger.

CHAPTER 6

Enhancement of melting by using extended surfaces in PCM

Contents

6.1	Introduction	154
6.2	Melting in a 2D rectangular cavity	154
6.2.1	Problem setup	154
6.2.2	Mesh and time Independence	156
6.2.3	Effect of cavity size on melting	157
6.2.4	Effect of varying the dimensions and operating conditions on the melting of PCM	159
6.2.5	Onset of convection	164
6.3	Melting in a finned PCM compact heat exchangers . . .	166
6.3.1	Problem setup	166
6.3.2	Mesh and Time Independence Study	168
6.3.3	Description of the melting process	172
6.4	Comparison with heat exchanger with plain PCM layer	174
6.4.1	Comparison of the melting over time	174
6.4.2	Overall heat transfer performance	175
6.5	Summary	175

6.1. Introduction

The addition of extended surfaces has shown to greatly improve the performance of both conventional and PCM heat exchangers. This chapter will explore the effect of adding fins to the PCM layer of a strip-fin compact heat exchanger. The melting of PCM within a 2D rectangular cavity bounded by finned walls will first be studied in Section 6.2 to better understand the melting within small cavities and estimate appropriate dimensions for a finned PCM cavity based on the example loading profile provided by the industrial partner. The results will then be used within a three dimensional conjugate heat transfer model of the strip-fin heat exchanger to study the melting performance in Section 6.3. Finally, the performance of the heat exchanger with finned PCM channels will be compared with the plain PCM channel heat exchanger geometry studied in the previously chapter in Section 6.4.

6.2. Melting in a 2D rectangular cavity

6.2.1. Problem setup

To find appropriate dimensions for the fins inside the PCM layer, the effect of fin spacing (z) and plate height (y) were analysed by conducting a parametric study on

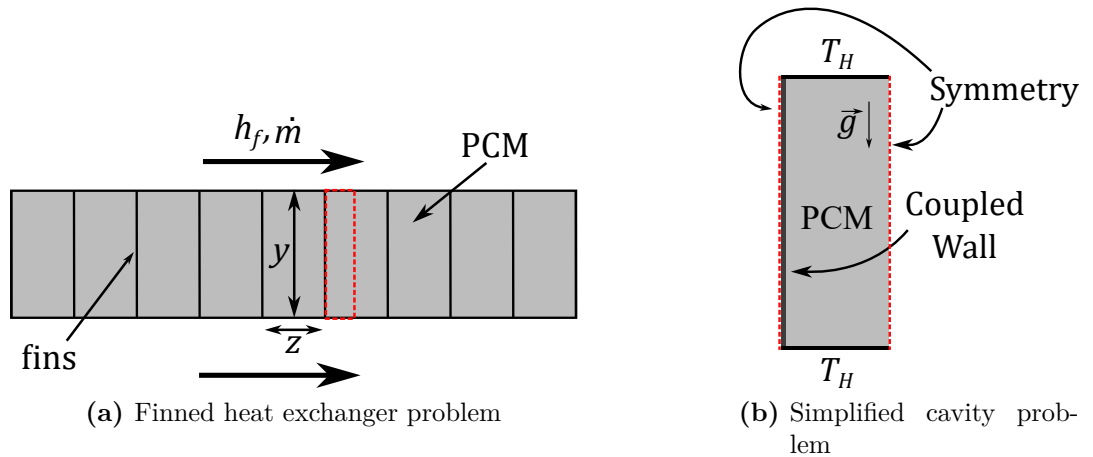


Figure 6.1: (a) Geometry of heat exchanger with finned PCM layer. Red square represents the area approximated by the simplified cavity problem. (b) Simplification of finned heat exchanger problem, where the hot fluid is represented as a constant temperature (T_H) boundary condition. Half the cavity is modelled by applying a symmetry boundary condition to reduce computational cost. Finally the fins are modelled as solid lines.

Table 6.1: Material Properties used in 2D cavity simulations

Material Properties	RT18 HC	Aluminium
Density (l) [kg/m ³]	770	2719
Viscosity [mPa.s]	2.31	-
Specific heat capacity [J/kgK]	2000	871
Thermal conductivity [W/mK]	0.2	202.4
Heat Storage capacity [kJ/kg]	260	-
Melting Range [K]	290.15 - 292.15	-
Thermal Expansion Coefficient	6×10^{-4}	-
Mushy Zone Constant	10^6	-

a simplified model of the fin layer, approximated by a rectangular cavity as shown in Figure 6.1. The model consists of a cavity filled with RT18 HC PCM, that is heated uniformly at the lower and upper wall. Aluminium fins are modelled by using a conjugate heat transfer approach where the fins heat the PCM from the sides. For this, the fins are set to a constant thickness 0.254 mm , matching the 1/8-13.95 strip fin geometry from [Kays & London \(1964\)](#). The material properties used in this simulation for the PCM and aluminium are provided in Table 6.1. To reduce computational cost, only half the cavity is modelled by applying a symmetry boundary condition, as shown in Figure 6.1.b.

The effect of gravity is taken into account and natural convection is modelled by using the Boussinesq approximation, following a similar methodology to what was previously outlined in Chapter 5. The model assumes that the solid PCM in the channel is constrained out of plane, such that the solid PCM does not sink due to density differences between the solid and liquid phases. This was done as modelling the sinking of solid PCM requires more complex models where the melting front is tracked to approximate the sinking velocity of the solid PCM (see [Faden et al. \(2018\)](#) for detail). As a result, smaller heat exchanger lengths have to be considered, such that a full conjugate heat transfer model of the heat exchanger is restricted.

The problem was solved in Ansys FLUENT, where the SIMPLE pressure-velocity coupling was used to increase the stability of the simulation. The PRESTO! scheme was used for pressure discretization, and second order accurate schemes were used for the momentum and energy discretization. For increased stability, CFL-based adaptive time advancement was selected, where the time step was slowly ramped up from a small initial time step to $\Delta t_{\max} = 0.1 \text{ s}$, which was found to be sufficiently small based on the study conducted in the following section.

6.2.2. Mesh and time Independence

A mesh independence study was conducted by creating three grids of varying sizes outlined in Table 6.2, for a square cavity of length $y = z = 10$ mm.

Table 6.2: Mesh sizes for mesh independence study of 2D cavities.

Mesh	Mesh size (10^3 cells)
Coarse	1.3
Medium	5.2
Fine	8.1

The average temperature of the metal and the PCM and the average liquid fraction were tracked over time as shown in Figure 6.2. This indicates that average properties are in close agreement for all meshes with the medium mesh converging towards the fine mesh for all properties. Figure 6.3 shows the impact of different time step sizes. A good agreement for the average liquid fraction and temperature for the studied maximum time steps varying between 0.1 s and 0.05 s is observed, such that a time step size of $\Delta t = 0.1$ s was recommended for the study.

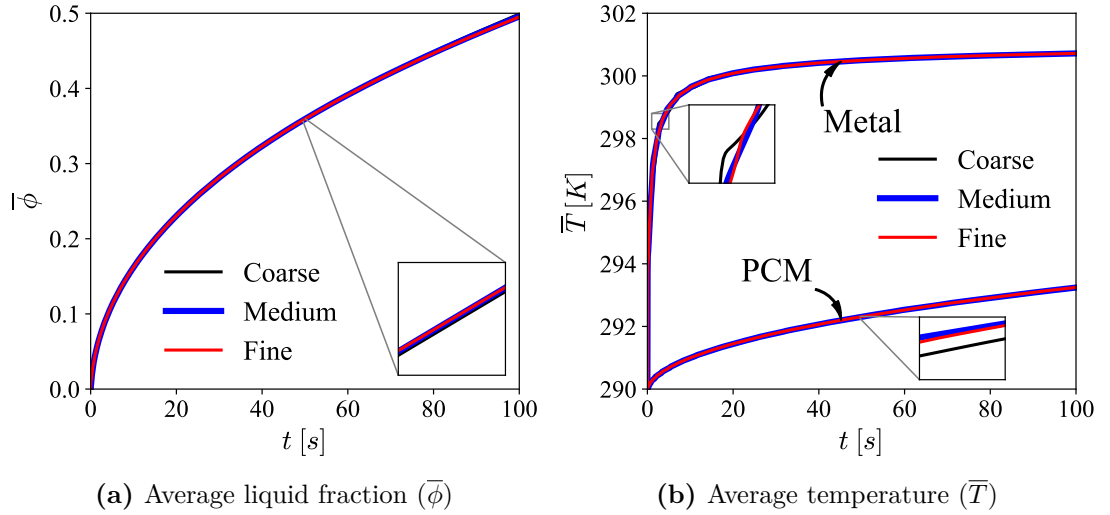


Figure 6.2: Mesh Independence study for constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature ($\bar{T}$). The study was conducted with a maximum time step size of $\Delta t_{\max} = 0.1$ s.

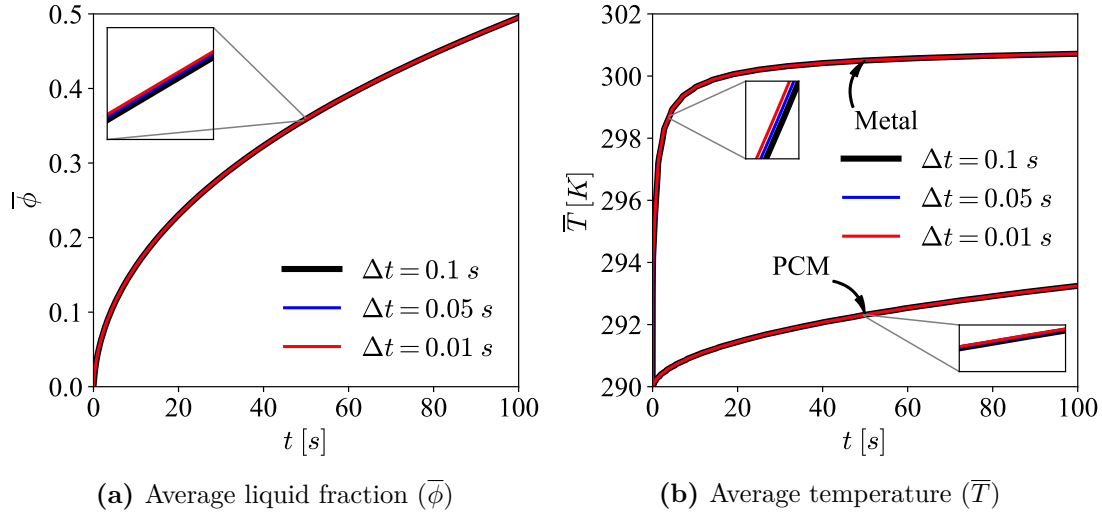


Figure 6.3: Time Independence study for constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature ($\bar{T}$). The study was conducted by using the medium mesh.

6.2.3. Effect of cavity size on melting

A parametric study was conducted to evaluate the correct size for a square PCM cavity to be able to melt within the allocated melting cycle by varying the length of the cavity between 4 and 10 mm. Figure 6.4.a, shows the average liquid fraction over the melting cycle, indicating the amount of melted PCM. Figure 6.4.b shows the amount of stored heat in the PCM obtained by assuming constant density and specific heat capacity for all phases as:

$$E = m_{PCM} [L\phi + c_p(\bar{T}(t) - T_{init})], \quad (6.1)$$

where m_{PCM} is the total mass of PCM assuming constant density in the liquid and solid phase. From this graph, it is possible to observe that as the cavity becomes larger, the amount of heat that is stored within the PCM will increase. The 10×10 mm cavity stores around 3 times more heat than the 4×4 mm cavity, linked to a greater surface area. On the other hand, Figure 6.4.a indicates that the square cavity with dimensions 4×4 mm is almost fully melted at the end of the melting cycle, while for the 10×10 mm cavity, only half of the PCM is melted. This leaves a significant amount of the PCM unused, leading to an unnecessary increase in the volume and mass of the system. Based on this analysis, a 5×5 mm cavity length was chosen for future analysis, as this would leave a small safety margin, while allowing for a decrease in mass and volume of the system.

Figure 6.5 shows streamlines of the PCM fluid flow for a 5×5 mm cavity at the end of the melting cycle, where two main recirculation zones are apparent. The major recirculation zone, where the fluid velocity is the highest flows upwards along the side of the fin and then follows shape of the melt front downwards. The vortex carves a smooth dome like shape into the melt front on the upper surface. At the lower surface, the melt front is curved inwards where a secondary recirculation zone

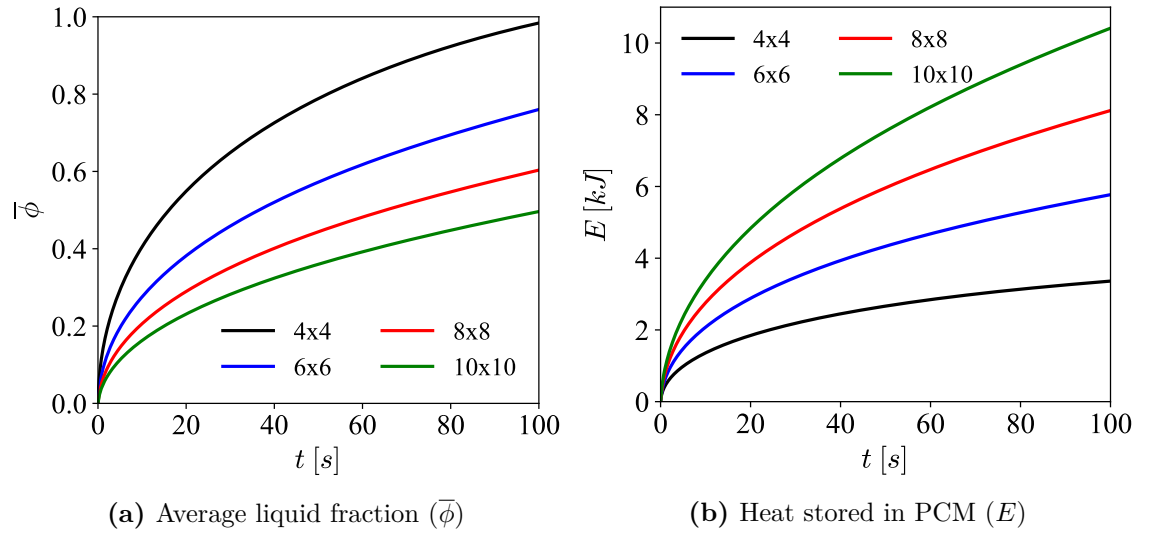


Figure 6.4: Effect of size [mm] on melting in constrained cavity: (a) Average liquid fraction $\bar{\phi}$, (b) heat stored in PCM (E).

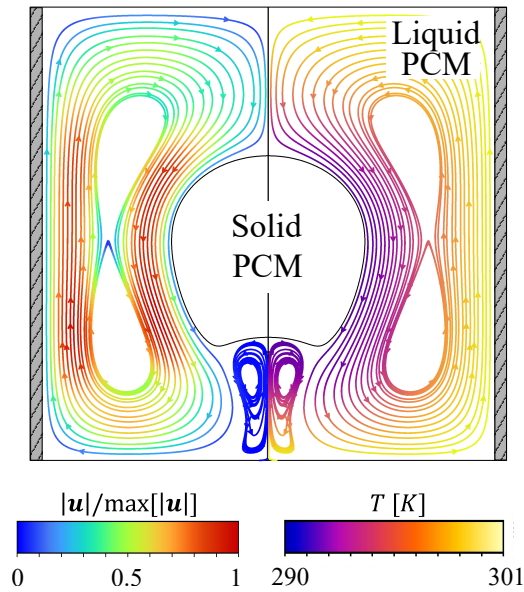


Figure 6.5: Streamlines of melted PCM coloured by normalized velocity magnitude ($|\mathbf{u}|/\max[|\mathbf{u}|]$) and Temperature (T) for 5×5 mm cavity at $t=100$ s. The position of the melt front is shown by a black line.

transports warm fluid from the lower wall to the solid/liquid front and shapes the interface at this location.

6.2.4. Effect of varying the dimensions and operating conditions on the melting of PCM

6.2.4.1 Effect of aspect ratio at constant free flow area

The effect of aspect ratio ($AR = z/y$) on the melting in a cavity with constant free flow area ($A_{front} = 25 \text{ mm}^2$) was studied, where for $AR = 1$, the length of the cavity was set to $z = 5 \text{ mm}$. Figure 6.6 provides the different dimensions used in the model and compares the corresponding aspect ratio cavities. Note that $AR = 0.5$ and $AR = 2$ have opposite dimensions such that $y_{AR=0.5} = z_{AR=2}$.

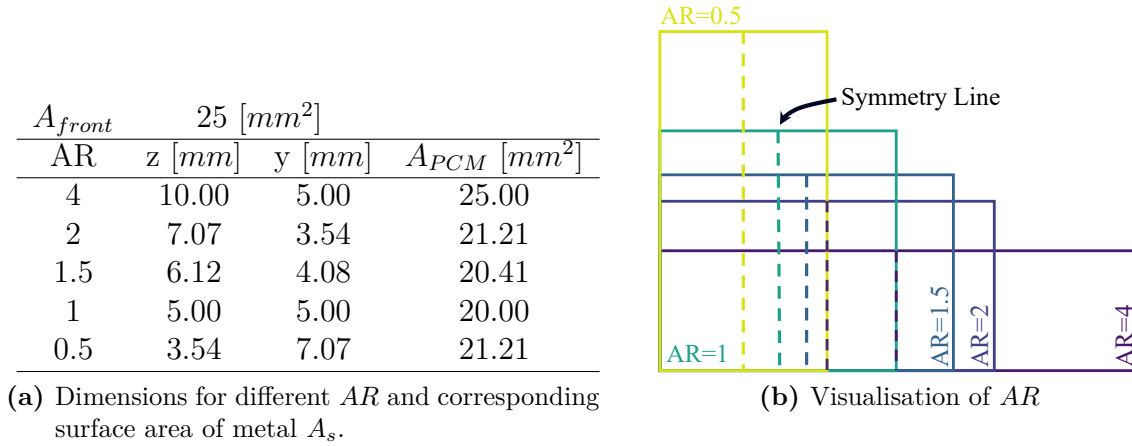


Figure 6.6: Dimensions of cavity used for different Aspect Ratio (AR)

Figure 6.7 shows the effect of aspect ratio on the average liquid fraction and the amount of heat stored in the PCM. A similar trend is observed for the liquid fraction and the stored energy in the cavity. As AR approaches 1, the amount of PCM that is melted decreases, while an increase and decrease of AR away from 1 leads to an increase in the amount of PCM that is melted at a given time. This can be explained by the increase in the fin surface area A_{PCM} such that more of the PCM is in contact with the metal. To further investigate the improvement in the energy absorption rate, the melting front is plotted in Figure 6.8 for increasing aspect ratio, where $AR \geq 1$. For an increasing aspect ratio, the length of the inward curved melt front increases, such that the secondary recirculation zone is larger underneath the solid PCM. Note that for $AR = 4$ at $t = 100 \text{ s}$, there is no melt front that could be measured, as $\phi > 0.5$ in all cells of the domain.

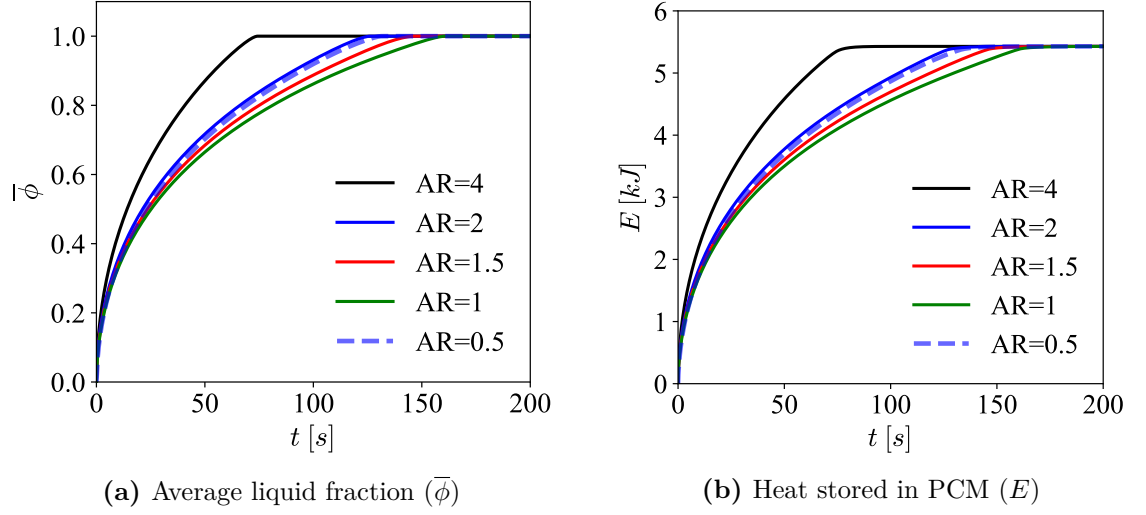


Figure 6.7: Effect of Aspect Ratio on melting in constrained cavity: (a) Average liquid fraction ($\bar{\phi}$), (b) heat stored (E) in PCM.

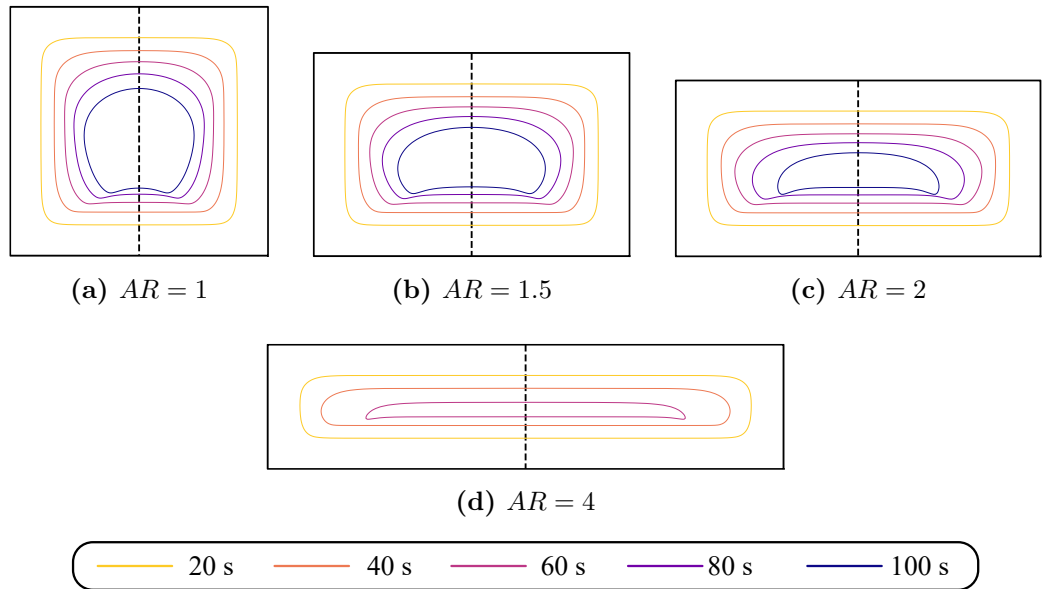


Figure 6.8: Position of the melt front ($\phi = 0.5$) over time for increasing aspect ratio.

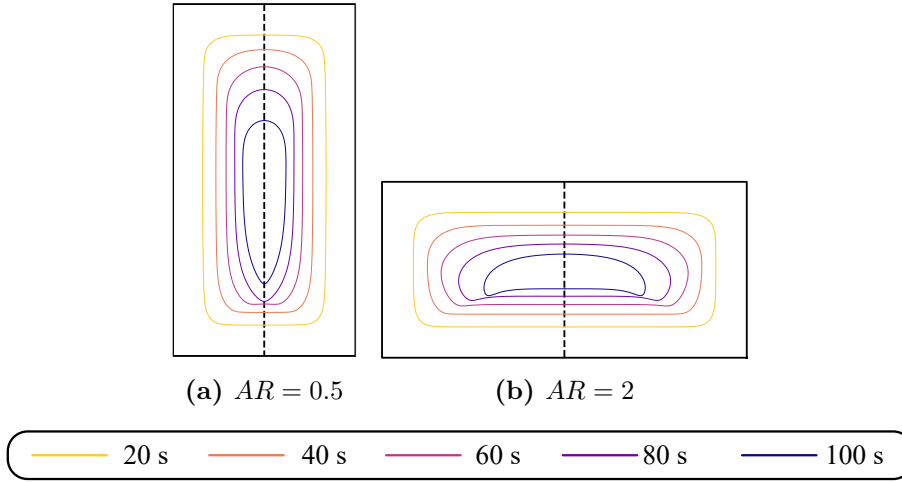


Figure 6.9: Comparison of position of melt front ($\phi = 0.5$) for (a) $AR = 0.5$ and (b) $AR = 2$.

To compare the effect of cavity orientation, two cavities with the same surface area were studied based on $AR = 2$ and $AR = 0.5$. $AR = 2$ showed a slight improvement on the amount of melted PCM and stored heat when compared to $AR = 0.5$ as shown in Figure 6.7. Figure 6.9 shows the melt front for these values of AR , where it is apparent that a greater secondary recirculation zone is visible for $AR = 2$ than $AR = 0.5$, which could explain this small improvement. For $AR = 0.5$, a difference in the shape is visible, where the melt front takes on the shape of a bullet, leading to a single recirculation zone following the length of the fin and the side of the solid PCM phase.

6.2.4.2 Effect of temperature

The effect of temperature on the melting within the cavity was studied by varying the wall temperature T_H between 295.15 and 301.15 K in Figure 6.10. It is possible to observe that a decrease in the wall temperature leads to a smaller amount of melted PCM over time, as well as a decrease in the amount of stored heat over time. It is possible to note that as all the PCM is melted, the total amount of energy is lower at lower wall temperature, due to the lower total amount of sensible heat that can be stored within the PCM. By looking at the position of the melt front, it is possible to see that the wall temperature has an effect on the shape of the melt front, where the onset of natural convection is delayed as the temperature decreases.

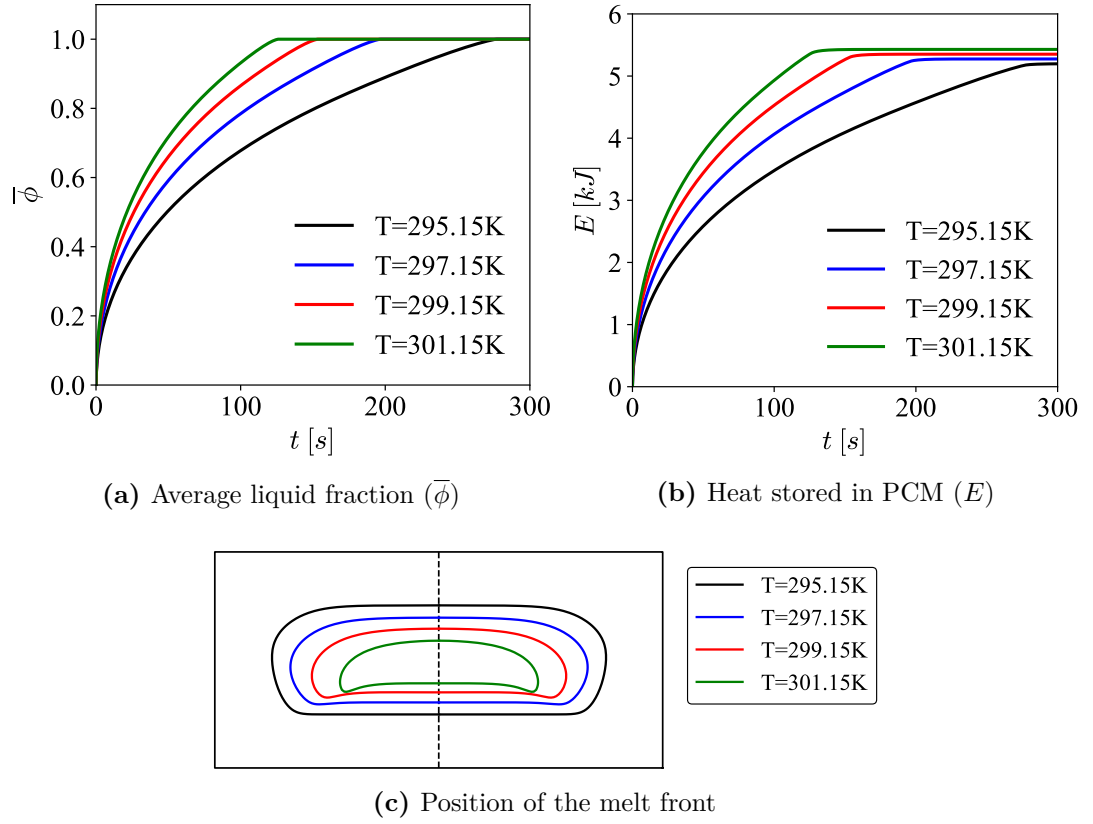


Figure 6.10: Effect of wall temperature on melting in constrained cavity at $AR=2$: (a) Average liquid fraction ($\bar{\phi}$), (b) Heat stored in PCM (E), (c) Position of the melt front at $t = 100\text{ s}$.

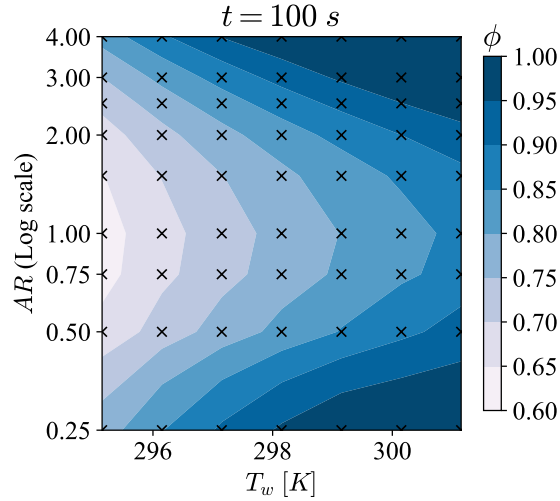


Figure 6.11: Effect of varying the aspect ratio (AR) and wall temperature (T_w) on the liquid fraction (ϕ) at the end of the melt cycle. Crosses (x) represent the data points used to generate the contour plot.

Figure 6.11 shows a summary contour plot of the effect of varying the wall temperature on the final liquid fraction at the end of the melting cycle for different aspect ratio values. The plot further indicates how moving away from $AR = 1$ increases the amount of PCM that can be melted at a point in time and provides guidance for choosing appropriate dimensions for the heat exchanger.

6.2.4.3 Effect of surface area

A potential way of mitigating the drop in the melting thickness is to create a finned PCM layer, where the fin spacing is varied over the length of the heat exchanger, such that the frontal area of PCM is reduced. The effect of fin and plate surface area was studied based on the $AR = 2$ geometry. The plate spacing y was kept constant and the surface area was varied based on the dimensions given in Table 6.3. The effect of surface temperature on the melting was then studied with results shown in Figure 6.12. The contour plot shows that as the wall temperature increases and the surface area decreases, more of the PCM is melted. This happens as there is a higher driving force for the heat transfer at higher temperature and as the cavity size decreases, the volume of the PCM in the cavity decreases.

While the parametric study provides useful results to understand the general trend

A_{PCM} [mm ²]	z [mm]	y [mm]	A_{front} [mm ²]
24	8.46	3.54	29.95
23	7.96		28.18
22	7.46		26.41
21	6.96		24.64
20	6.46		22.87
19	5.96		21.10
18	5.46		19.33
17	4.96		17.56
16	4.46		15.79

Table 6.3: Dimensions of cavity for constant plate spacing

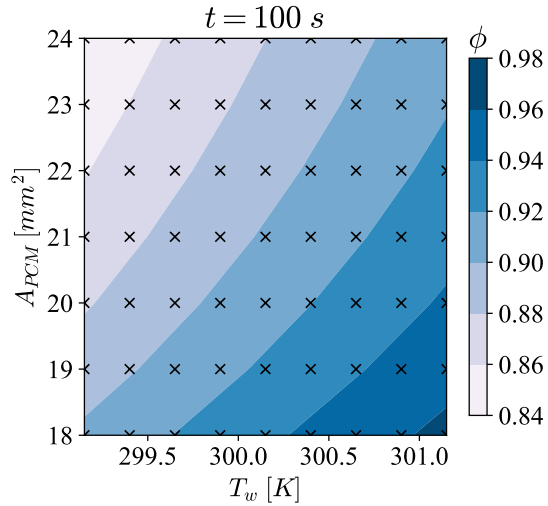


Figure 6.12: Effect of varying the surface area and wall temperature (T_w) on the liquid fraction (ϕ) at the end of the melting cycle. Crosses (x) represent the data points used to generate the contour plot.

of melting, a more detailed study is still required. In the previous chapter it was seen that the fluid temperature drop decrease significantly over time as a function of the amount of melted PCM.

6.2.5. Onset of convection

The melting of PCM is impacted by convection for the cavity case subjected to the current loading profile, which leads to a change in the shape of the melting front. To investigate the impact of convection on averaged properties and the shape of the melting front, cases with natural convection were compared with cases where the convection was suppressed. This was achieved by setting the gravitational acceleration to:

$$\begin{cases} g = -9.81 \text{ m/s}^2 & \text{With Convection,} \\ g = 0 \text{ m/s}^2 & \text{Conduction Only.} \end{cases} \quad (6.2)$$

Results for $AR = 1$ and $AR = 2$ on the stored heat are compared in Figure 6.13.a, where it is possible to see that the onset of convection occurs significantly earlier than in the previous chapter, with an onset of convection at around $t = 25 \text{ s}$ for AR and $t = 20 \text{ s}$ for $AR = 2$. This indicates that the point at which convection starts to become dominant is different for the two aspect ratios, where a larger aspect ratio is shown to have an earlier onset of convection. At the end of the melting cycle at

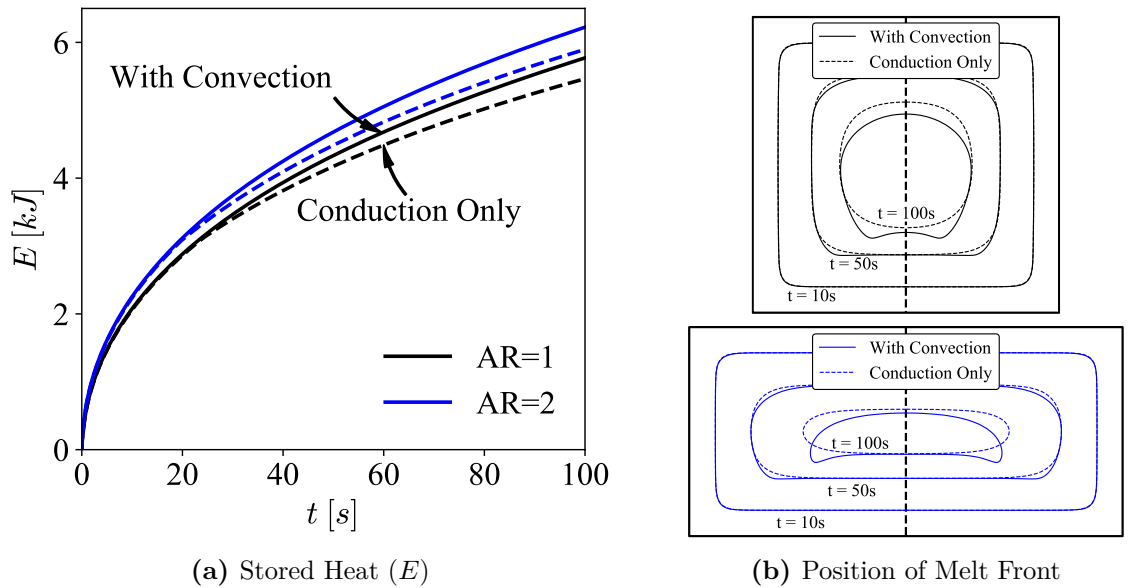


Figure 6.13: Comparison of conduction only (---) and with convection (—) modelling assumption, for $AR = 1$ and $AR = 2$ at $T_w = 301.15\text{K}$.

$t = 100$ s, the conduction only case predicts the amount of stored heat within 1% of the case with convection, with the largest difference being measured at $AR = 1$, with a percentage difference of 0.815%.

Figure 6.13.b shows the melting front at different time steps for the different modelling assumptions. It is possible to see that by neglecting the convection, a relatively similar position of the solid PCM is found, although the shape is not fully reproduced as time progresses where convection plays an important role in shaping the solid PCM.

Furthermore, the effect of wall temperature on the convection was analysed, as shown in Figure 6.14. This shows that for the predicted amount of energy that is stored within the PCM, there is not a significant difference for the case with convection and the conduction only case when the wall temperature is under 299.15 K, with a percentage difference of less than 1%, as the lower temperature leads to a later onset in convection. However, when looking at the shape of the melt front, it is possible to identify a small difference for $T \leq 299.15$ K. For the case where the convection is neglected, the shape of the melt front stays similar to the shape of the cavity, such that it resembles a square without rounded corners. As discussed previously, when convection is accounted for, a difference in the shape of the melt front is observed between the top and bottom half of the cavity, due to the two recirculation zones observed in Figure 6.5

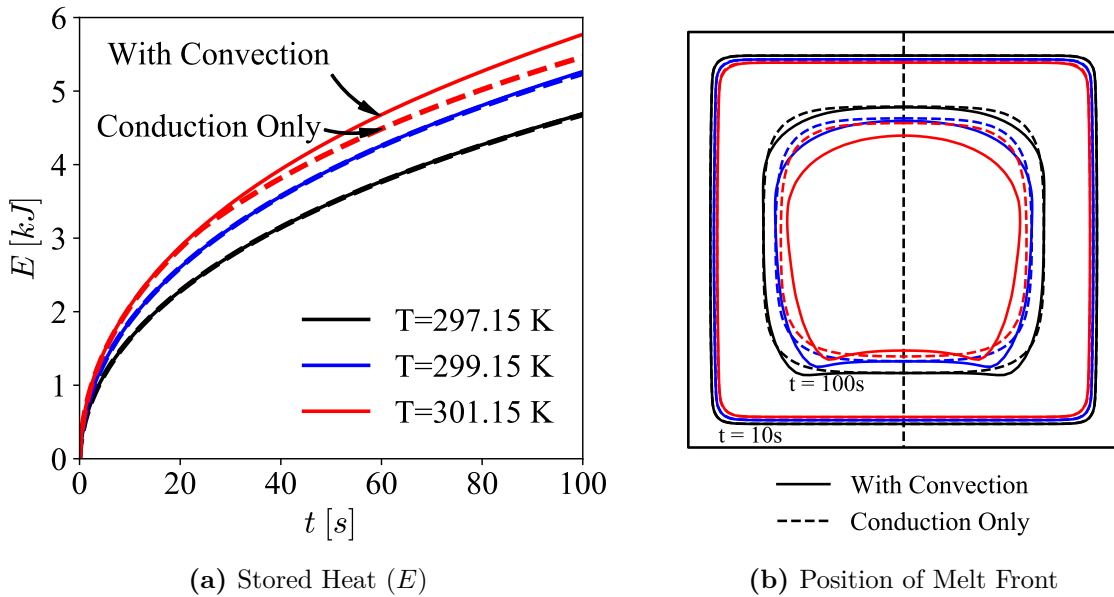


Figure 6.14: Comparison of conduction only (---) and with convection (—) modelling assumption, for different wall temperature $T = 297.15 - 301.15$ K at $AR = 2$.

Based on the sensitivity analysis, it is possible to conclude that if the shape of the melt front is to be faithfully reproduced, the natural convection of the liquid PCM pool has to be modelled, as the onset of convection occurs early in the melting process. This is different to what was previously observed when no fins are present in the heat exchanger, where the heat exchanger was not affected by convection for the given loading profile.

6.3. Melting in a finned PCM compact heat exchangers

Following the initial assessment on 2D cavities, a 3D conjugate heat transfer model of the PCM compact heat exchanger is created and studied in this section. The model makes it possible to obtain results for more complex operating conditions, such that the complex effect of the fluid temperature drop on the melting of PCM in the adjacent layer can be captured.

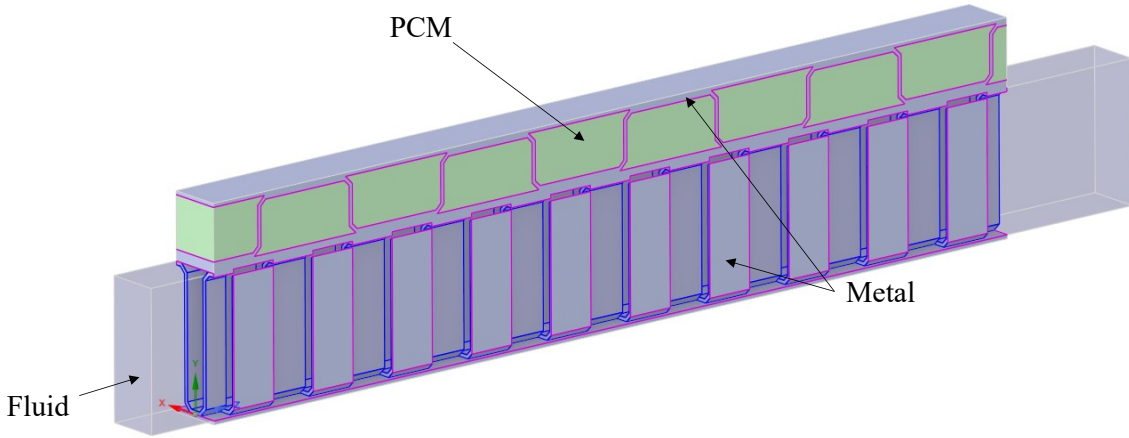
6.3.1. Problem setup

The results from the previous parametric study were used to provide an appropriate size for the PCM layer in the PCM-compact heat exchanger geometry shown in Figure 6.15.a. As a result, the PCM cavity dimensions were chosen such that $y \times z \approx 25 \text{ mm}^2$ with an aspect ratio $AR = 2$. The length of the heat exchanger was chosen based on 10 periodic lengths of fin interruptions, giving a total length of 63.5 mm , similar to the heat exchanger studied in Chapter 5. This leads to around 9 PCM cavities filled with RT18 HC, where the last cavity has a smaller surface area than the remaining cavities. Water is used as the working fluid, while aluminium is used for the separating plates and fins. The material properties for the simulation were set based on the material properties shown in Table 6.4.

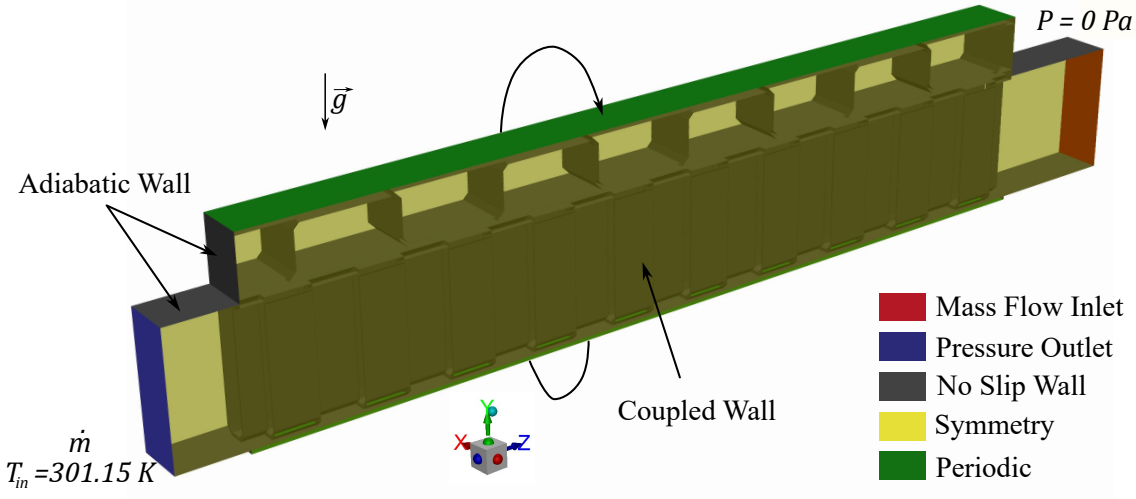
The boundary conditions used in this problem are shown in Figure 6.15.b. As convection cannot be neglected based on the previous parametric study, gravity was taken into account in the simulation, where the full layer of PCM was modelled without interruption. A periodic boundary condition was therefore used within the separating plate to connect the upper part of the PCM layer to the lower part of the water channel. Outer walls were set to no slip adiabatic walls, while no slip coupled walls were used to connect the different material zones to each other. The inlet was set to a mass flow inlet based on the operating Reynolds number, with a

flow temperature of $T_{in} = 301.15 \text{ K}$ and the outlet was set to a pressure outlet at $P = 0$. The sides of the heat exchanger were extended by using symmetry boundary conditions normal to the $\hat{x}$ axis.

Similar to the previous heat exchanger melting cases, an initial velocity and temperature profile in the fluid and metal layer was obtained by maintaining the PCM at a constant temperature of $T = 290.14 \text{ K}$. This was used as the initial condition to start the melting cycle.



(a) Geometry of PCM-Compact Heat Exchanger with finned PCM layer

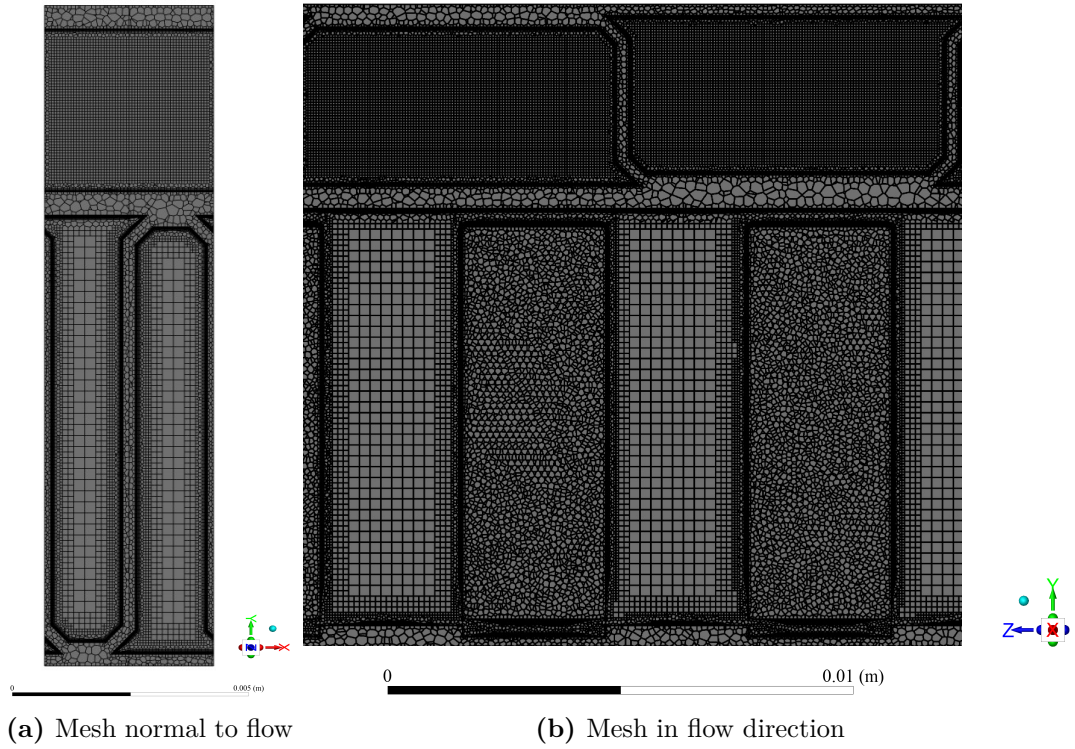


(b) PCM-Compact Heat Exchanger domain with boundary conditions

Figure 6.15: Model of the PCM-Compact Heat Exchanger with finned PCM layer.

Table 6.4: Material Properties used in compact heat exchanger simulations

Material Properties	RT18 HC	Water	Aluminium
Density (l) [kg/m ³]	770	998.2	2719
Viscosity [mPa.s]	2.31	1.003	-
Specific heat capacity [J/kgK]	2000	4182	871
Thermal conductivity [W/mK]	0.2	0.6	202.4
Heat Storage capacity [kJ/kg]	260	-	-
Melting Range [K]	290.15 - 292.15	-	-
Thermal Expansion Coefficient	6×10^{-4}	-	-
Mushy Zone Constant	10^6	-	-

**Figure 6.16:** Picture of the medium mesh

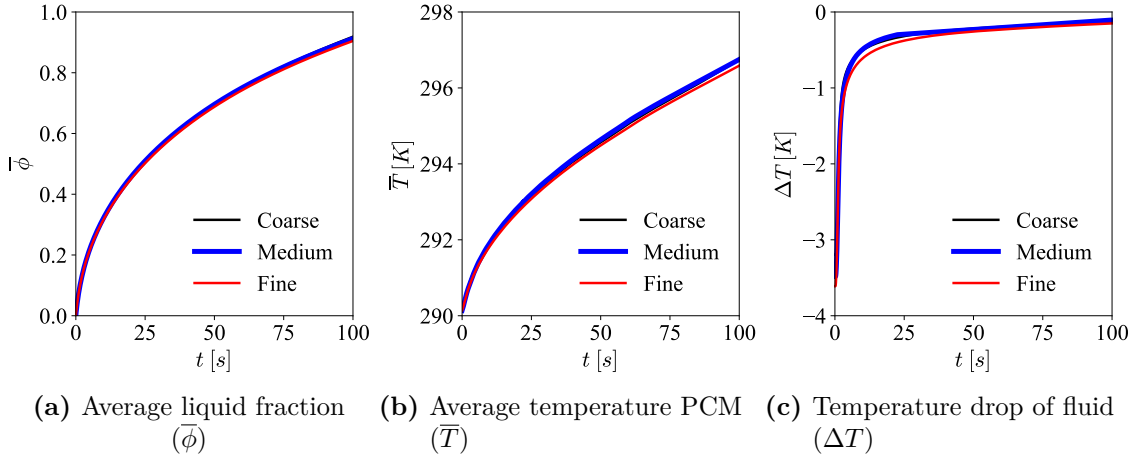
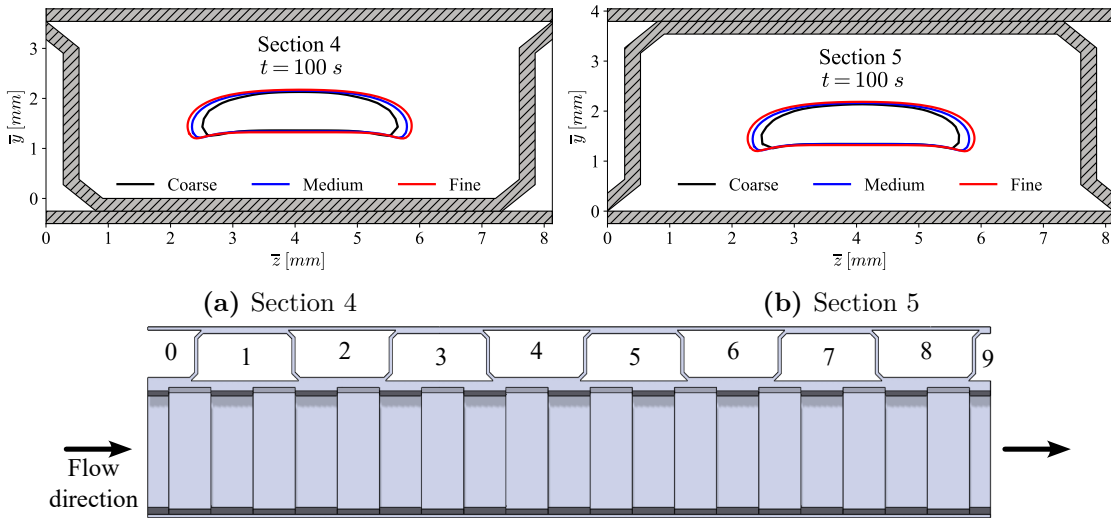
6.3.2. Mesh and Time Independence Study

The mesh and time independence study was conducted for an inlet temperature of $T_{in} = 301.15 \text{ K}$ and a Reynolds number of 200, corresponding to a mass flow rate of $\dot{m} = 2.21 \times 10^{-3} \text{ kg/s}$. For this three different meshes were created where the refinement at surfaces was increased leading to mesh sizes varying between 8.56M and 26.2M cells, see Figure 6.16 for a picture of the medium mesh. Table 6.5 captures the results of the mesh independence study on the initial condition, where a percentage difference below 5% was recorded for the different properties.

Table 6.5: Mesh Independence Study at initial condition

Size	Size $\times 10^6$	ΔT [K]	% diff	u_{max} [m/s]	% diff	ΔP [Pa]	% diff
Coarse	8.56	-3.328		0.1341		77.08	
Medium	14.5	-3.485	4.60%	0.1347	0.51%	75.77	-1.71%
Fine	26.2	-3.607	3.45%	0.1348	0.02%	76.45	0.89%

A mesh independence study on the melting process was then started. For this, an initial time step of $\Delta t = 1 \times 10^{-4}$ s was used, which was increased by a factor of 1.2 every 200 time steps following the time independence study that will be shown further down. For the Fine mesh, a smaller initial time step size of $\Delta t = 5 \times 10^{-5}$ s

**Figure 6.17:** Mesh independence Study: average properties over time.**Figure 6.18:** Mesh Independence Study: position of the melting front over.

and a step size update value of 400 was used to ensure the simulation was time step independent. Some discrepancies are observed between the different meshes, with a slightly lower average temperature between the medium and fine mesh at the end of the simulation and some small differences in the temperature drop at the beginning of the simulation. This could be linked to issues with the time step size update method used for the fine mesh, where the time step size was not sufficiently small enough. Furthermore, the position of the melting front was plotted for different sections in Figure 6.18. This shows good agreement in the position of the melt front between the medium and fine mesh. Due to the relatively good agreement between the different meshes for the averaged properties and the position of the melt front, the Medium mesh was used for further consideration.

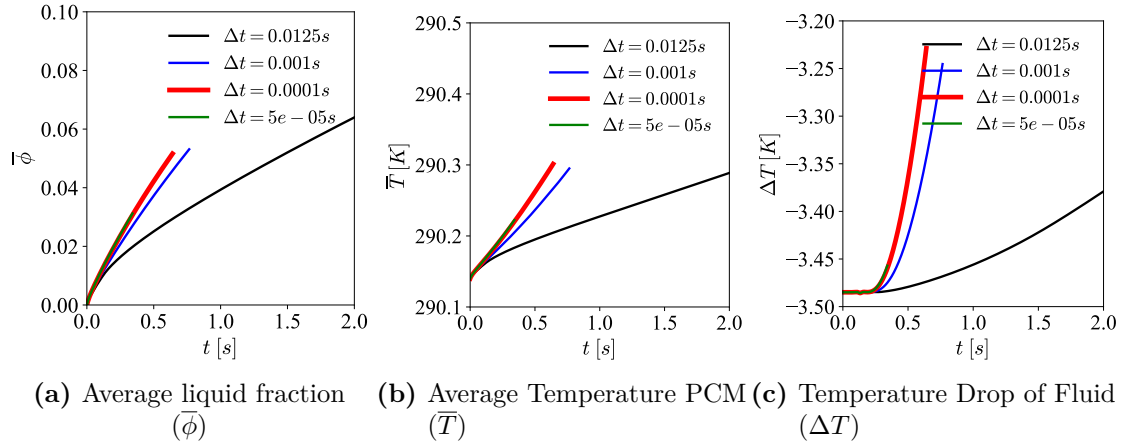


Figure 6.19: Initial time step independence study with constant time step size.

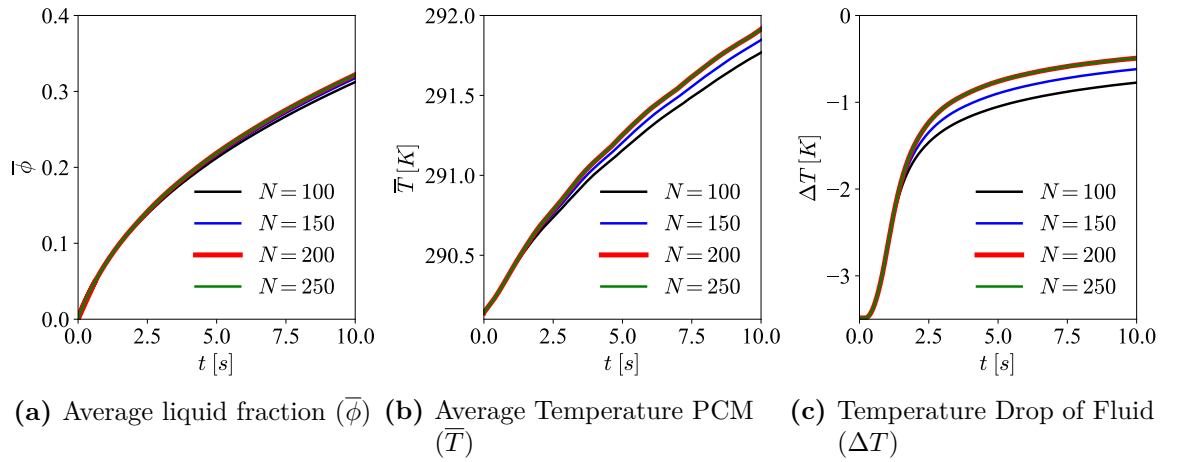


Figure 6.20: Time step independence study where the time step is updated every N th timestep

A time step independence analysis was initially conducted on the medium mesh, as the previously used time step size of $\Delta t = 5 \times 10^{-2} s$ was found to show significant divergence between the meshes, which was not in line with the mesh independence study at the initial condition. Initially, the time step size was consistently reduced and average properties for PCM temperature, liquid fraction and temperature drop in the fluid were captured over time as shown in Figure 6.19. The results shown in the figure were obtained by running the simulations at varying time step sizes for 48h on 24 cores on ARC3. The average properties captured in those graphs indicate that as the time step size was reduced, a closer agreement between results could be observed with independence being reached for a time step size of $\Delta t = 1 \times 10^{-4} s$. By comparing to the different time step sizes that were used throughout this work, the current time step size is two orders of magnitude smaller. The difference could be linked to the larger amount of heat that is transferred from the water channel to the PCM in the first couple of seconds, where the thickness of melted PCM is still small. As the thickness of PCM increases, the effectiveness of the heat exchanger decreases, and less heat is transferred into the PCM. This suggests that as more of the PCM is melted, the chosen time step size can be increased, such that the computational cost of the simulation can be reduced. The time step size was increased by a factor of 1.2 every N time steps in Figure 6.20 where time step independence is noted from $N = 200$, which was used for the remaining analysis.

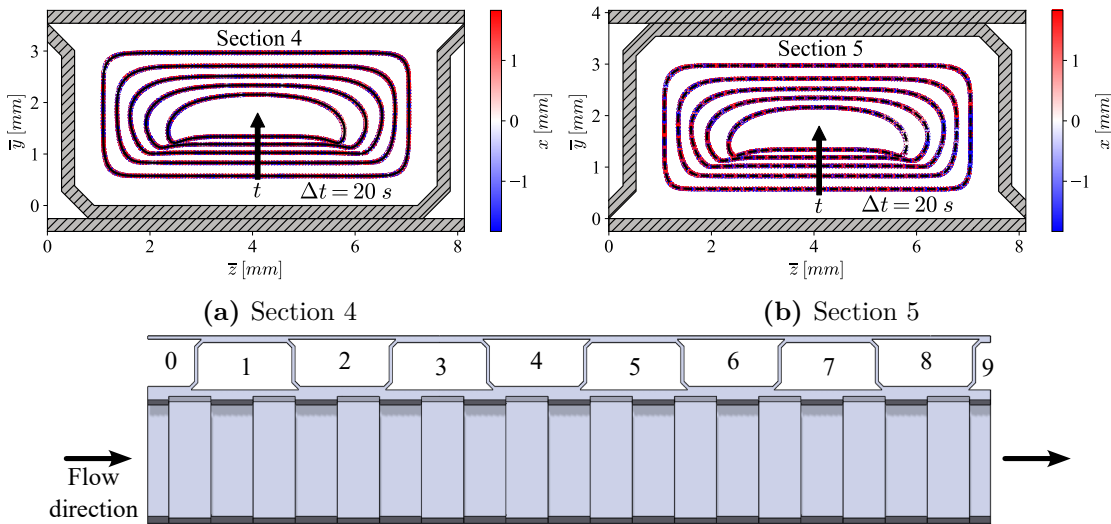


Figure 6.21: Change in the position of the melting front over the depth x of the heat exchanger.

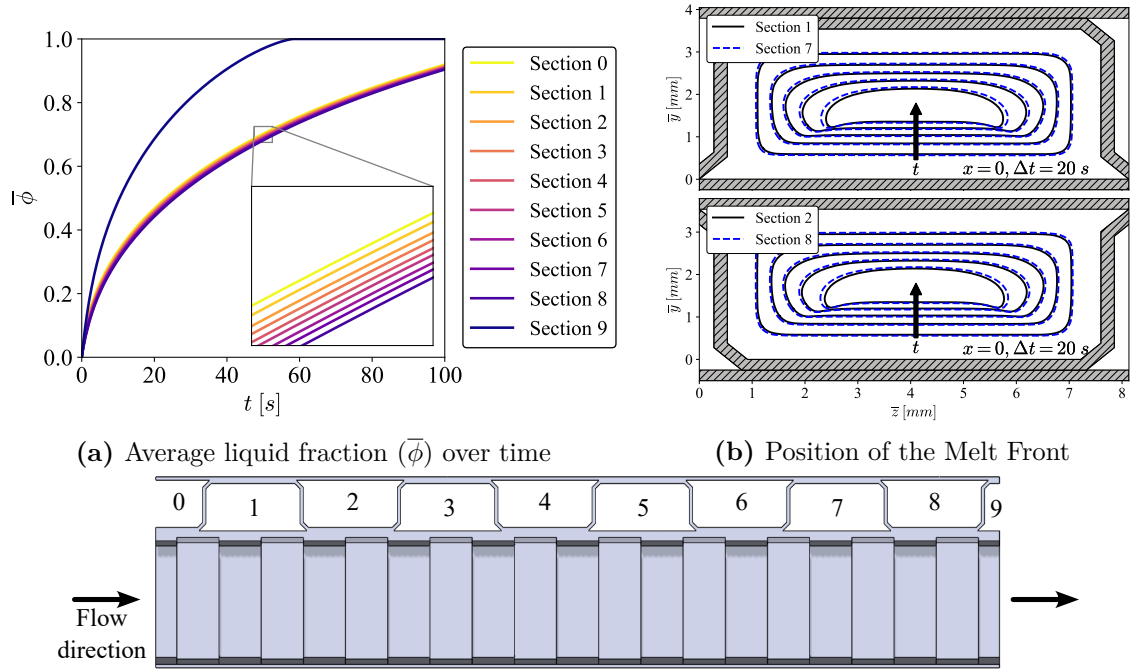


Figure 6.22: Change in melting over the length of the heat exchanger shown by: (a) Average liquid fraction ($\bar{\phi}$) over time, (b) Position of the melt front over the length of the heat exchanger.

6.3.3. Description of the melting process

6.3.3.1 Melting discontinuity of PCM in heat exchanger

Figure 6.21 shows the change in the melting thickness over the depth (x) of the heat exchanger as a result of the variation of temperature due to the fin positioning. The scatter plot indicates that the melting occurs very uniformly along the depth of the heat exchanger, similar to the observations made in Chapter 5. For the remaining work, isosurfaces at $x = 0$ will be extracted for simplification.

Figure 6.22 shows the effect of the temperature drop over the length of the heat exchanger on the average liquid fraction and the position of the melt front over time. Little difference in the average liquid fraction over the length of the heat exchanger is observed, with the exception of Section 9. However, this section has a smaller volume of PCM such that it melts more rapidly. The small difference in the melting over the length of the heat exchanger is further highlighted by Figure 6.22.b, where the position of the melt front is less advanced by only a small amount for the sections close to the outlet, when compared to the sections close to the inlet.

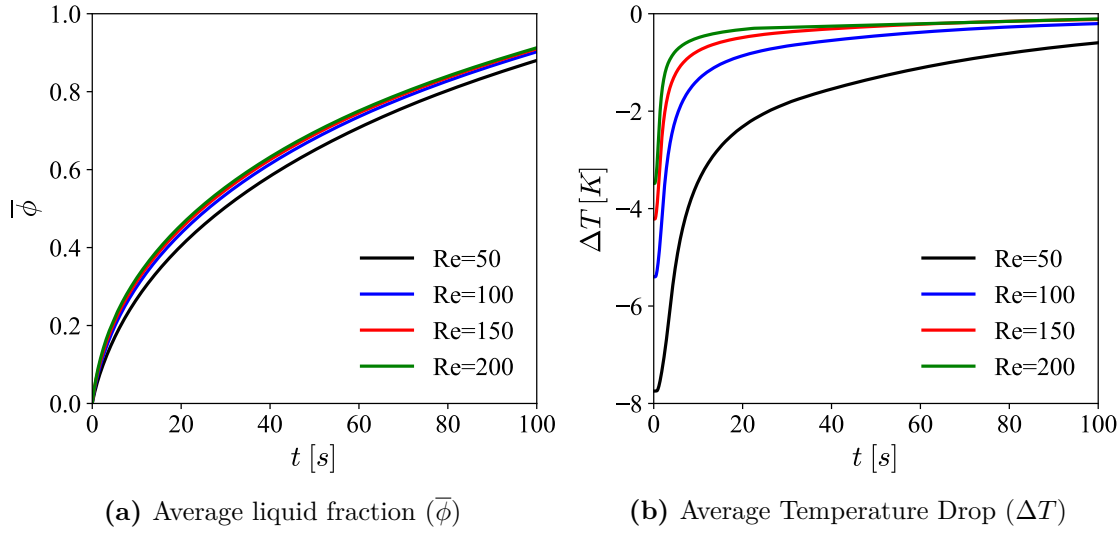


Figure 6.23: Comparison of average properties over time for different Reynolds number: (a) Average liquid fraction ($\bar{\phi}$), (b) Average temperature drop (ΔT).

6.3.3.2 Effect of Reynolds number

Figure 6.23 compares the average liquid fraction and temperature drop in the heat exchanger over a range of Reynolds number. From the liquid fraction plot, it is possible to see that increasing the Reynolds number past $Re = 100$ does not lead to a significant increase in stored latent heat in the PCM, with a larger difference being observed between $Re = 50$ and $Re = 100$. A larger difference is observed between the different Reynolds number on the temperature drop, where increasing the Reynolds number leads to a lower initial temperature drop value and a faster decrease in the temperature drop in time.

Figure 6.24 shows the position of the melt front at the end of the melting cycle for different values of the Reynolds number towards the inlet and the outlet of the heat exchanger. Only a small difference is observed at the inlet of the heat exchanger, where the position of the melt front is similar for all Reynolds number. Towards the outlet, a bigger difference is observed mostly for $Re = 50$, where significantly less of the PCM is melted when compared to the other Reynolds number values. This is linked to the more significant difference in the fluid temperature observed in Figure 6.23.b. For the other values of Re , the difference at the outlet is less pronounced, where the amount of melted PCM is almost identical as Re increases.

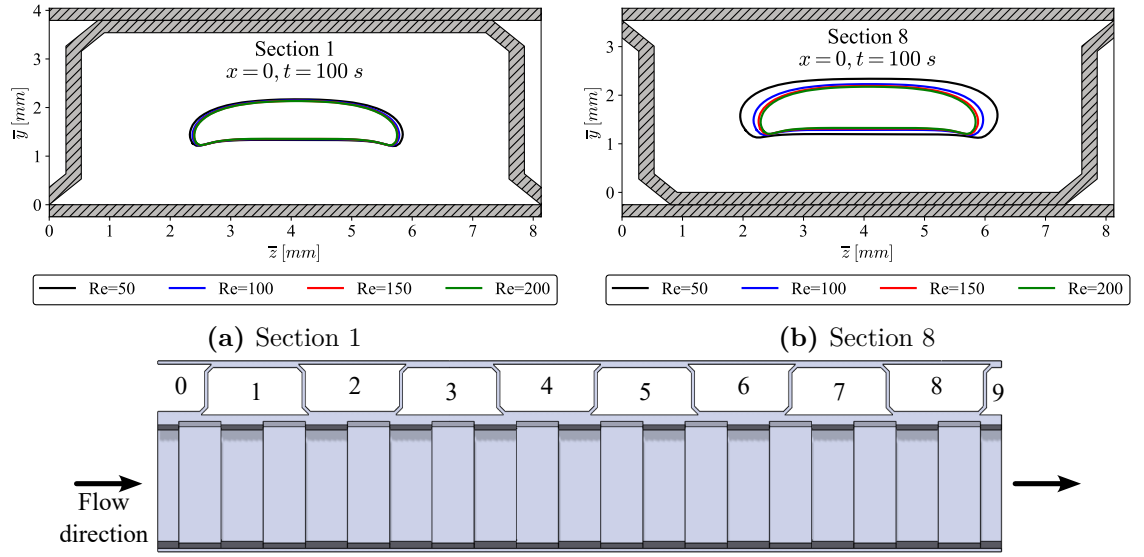


Figure 6.24: Comparison of the position of the melt front over the length of the heat exchanger for different Reynolds number at $t = 100$ s at (a) Section 1 and (b) Section 8.

6.4. Comparison with heat exchanger with plain PCM layer

The two geometries shown in Figure 6.25 are used as a comparison point to determine the effect of adding fins within the PCM on the performance of the PCM heat exchanger. The plain fin geometry based on the strip-fin heat exchanger studied in Chapter 5 has an equivalent volume of PCM to the finned geometry, with the same finned geometry in the fluid layer.

6.4.1. Comparison of the melting over time

Figure 6.26 compares the melting of PCM over time for the plain and finned PCM channel. This shows, that the finned geometry leads to a greater fraction of PCM that is melted over time when compared to the plain heat exchanger, leading to more heat being stored within the PCM. Initially, a similar effectiveness is recorded for the plain and finned heat exchanger, where the finned case has a slightly lower initial effectiveness, linked to the greater volume of aluminium extending into the PCM. As the melting starts, a slower decrease in the effectiveness is observed for the finned case, due to the greater surface area of fins leading to a more effective heat transfer between the fluid and PCM layers. Towards the end of the melting

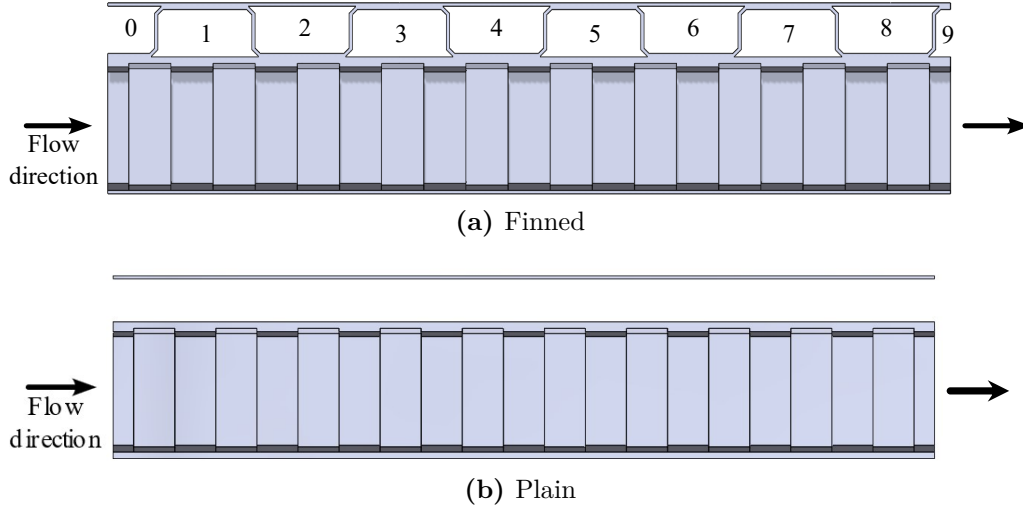


Figure 6.25: Geometries used for comparison.

cycle, the effectiveness of the finned heat exchanger tends towards the effectiveness of the plain heat exchanger.

6.4.2. Overall heat transfer performance

Figure 6.27 shows the average heat transfer performance of the two geometries over the duration of the melting cycle based on the average effectiveness and the amount of heat stored in the heat exchanger. The addition of fins to the heat exchanger led to an increase in average effectiveness varying between 60 and 105 % in the Reynolds number range studied here. The increase is attributed to the higher surface area created by the fins. As more of the PCM could be melted due to the greater surface area, more heat was also stored in the PCM for the finned case, with an average percentage increase of 8.7%. This is a significant improvement in the performance of the heat exchanger, as the addition of fins in the PCM layer does not change the pressure drop in the system, contrary to the addition of fins within the fluid layer.

6.5. Summary

The effect of adding fins to the PCM layer on the melting was studied in this chapter. Initially, 2D simulations of a rectangular cavity subjected to an aerospace loading profile were conducted, which was used to determine the effect of cavity size, aspect ratio and wall temperature on the melting of the PCM in the cavity. The analysis found that increasing the cavity size led to a smaller proportion of PCM that was melted in the cavity. More heat was stored in the PCM, due to the greater volume

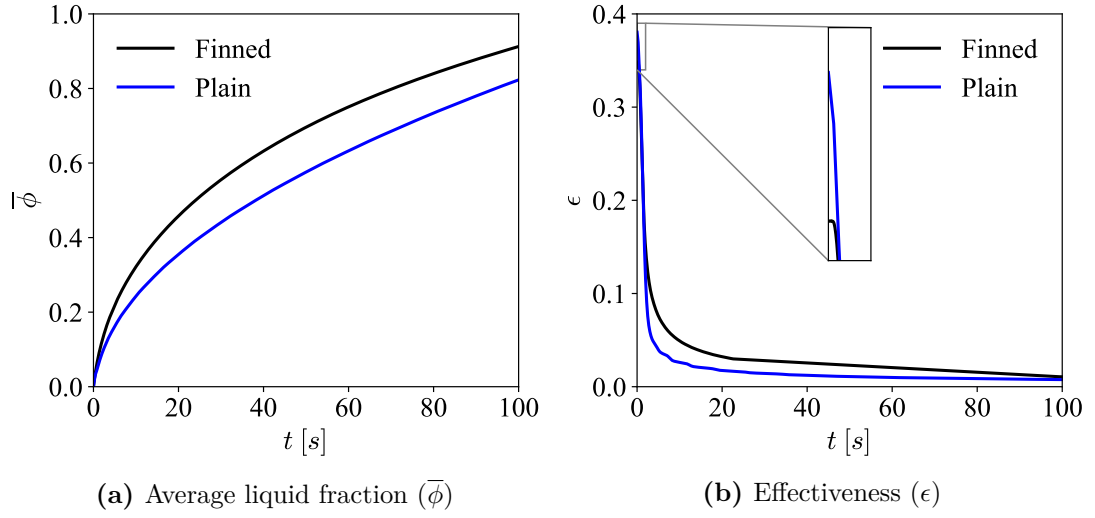


Figure 6.26: Comparison of bulk properties over time for $Re = 200$: (a) Average liquid fraction ($\bar{\phi}$); (b) Effectiveness (ϵ).

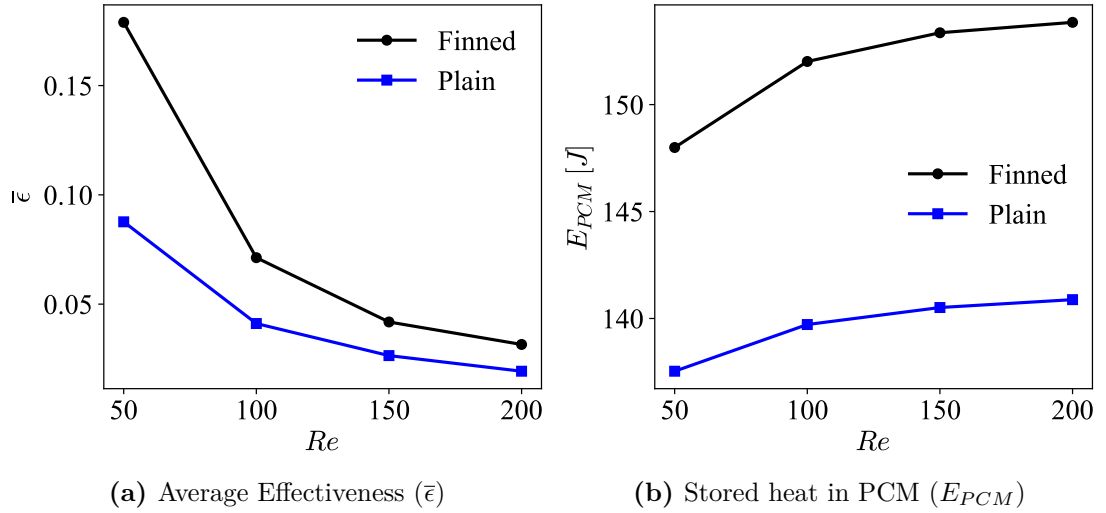


Figure 6.27: Comparison of the average heat transfer performance of finned and plain PCM-CHX over the melting cycle based on: (a) Average Effectiveness ($\bar{\epsilon}$); (b) Stored Heat in PCM (E_{PCM}).

of PCM that was melted. The lowest amount of melting was observed for a square cavity. As the aspect ratio of the cavity moved away from 1, the PCM melted faster, due to the increase surface area of fins. Decreasing the wall temperature led to slower melting of the PCM and a delay in the onset of convection.

The findings of the 2D simulations were used to create an appropriate heat exchanger geometry, based on the offset strip fin geometry introduced in the previous chapter. 3D conjugate heat transfer simulations were conducted on this geometry over a range

of Reynolds number. The results indicated that as the Reynolds number increased, more PCM was melted, where the difference in melting was mostly pronounced towards the outlet. The melting non-uniformity was attributed to the larger overall temperature drop observed at low Reynolds number.

Finally, the effect of adding fins to the PCM layer was studied by comparing the finned heat exchanger with the plain heat exchanger studied in the previous chapter. The comparison showed that the addition of fins led to a significant increase in the average effectiveness of the heat exchanger and heat stored in the PCM over the range of studied Reynolds number.

CHAPTER 7

Development of a low cost model for PCM compact heat exchanger

Contents

7.1	Introduction	179
7.2	Effectiveness-NTU model with phase change based on a linear Stefan Condition	179
7.2.1	Development of the model	179
7.2.2	Validation of effectiveness-NTU model	189
7.3	Alternative approach based on energy conservation . . .	193
7.3.1	Development of effectiveness-NTU model based on energy conservation	194
7.3.2	Comparison with linear Stefan effectiveness-NTU model .	197
7.4	Incorporating fins into the effectiveness-NTU model . .	197
7.4.1	Treatment of finned PCM cavity	198
7.4.2	Addition of fins to the full Effectiveness-NTU model . . .	212
7.5	Using the effectiveness-NTU model as a design tool . .	216
7.5.1	Motivation	216
7.5.2	Determining the dimensions of PCM compact heat ex- changer geometries	216
7.5.3	Determining optimal fin spacing for uniform melting of PCM over length of heat exchanger	220
7.6	Summary	226

7.1. Introduction

In the previous chapters, different designs of PCM-compact heat exchanger were subjected to a demand profile and their thermo-hydraulic response was analysed over a range of Reynolds numbers by using a conjugate heat transfer CFD model. This came at a significant computational cost, where the models had to be run on a high performance computer (HPC) for multiple hours to days.

During the initial design phase of heat exchangers, a large amount of designs have to be compared over a range of operating conditions. This makes the use of CFD inadapated due to the important computational cost associated with these models. For traditional compact heat exchanger, different analytical methods can be used to size the heat exchanger including the logarithmic mean temperature difference (LMTD) and effectiveness-NTU method ($\epsilon - NTU$). Furthermore, some models have been developed for sizing PCM heat exchangers.

Two different approaches for evaluating the thermal performance of PCM compact heat exchanger based on the $\epsilon - NTU$ model will be described for unfinned PCM compact heat exchanger in Section 7.2-7.3. Section 7.4 will describe the incorporation of fins into the $\epsilon - NTU$ model and Section 7.5 will provide two example cases of how the model could be used to design PCM-Compact heat Exchanger.

7.2. Effectiveness-NTU model with phase change based on a linear Stefan Condition

To simplify the problem, a compact heat exchanger without fins in the PCM layer and finned fluid layers will firstly be considered. The exchange of heat between the different layers will be controlled by a steady state effectiveness-NTU ($\epsilon - NTU$) model, while a pseudo-steady state Stefan model is used to predict the phase change.

7.2.1. Development of the model

7.2.1.1 Effectiveness-NTU model

Figure 7.1 shows the heat transfer problem for a heat exchanger with unfinned PCM layers sandwiched between finned fluid layers. In a PCM heat exchanger, the PCM acts as a heat sink or source, where some of the heat is absorbed or released during phase change in the form of latent heat. As a result of the phase change, the interface

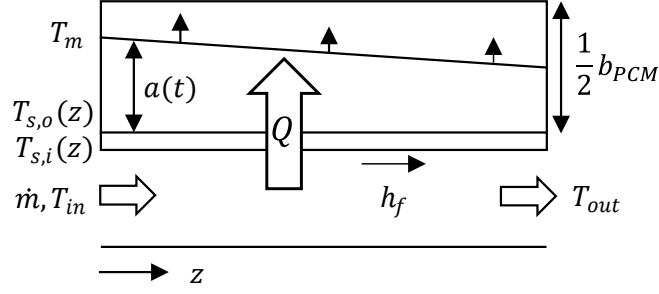


Figure 7.1: Diagram of the heat transfer problem in a PCM heat exchanger.

between the solid and liquid PCM will move, such that the thickness of melted PCM (a) changes over time. As the PCM absorbs heat from the fluid, the temperature of the fluid will decrease, such that a will be smaller at the outlet than the inlet, as discussed in Chapter 5. This problem can be represented as an ϵ -NTU problem.

The effectiveness (ϵ) is defined as the ratio between the actual and the maximum heat flux. Here, it is assumed that the latent heat is significantly larger than the sensible heat, such that the variation in the sensible heat in the solid layer over time is ignored, such that the solid/liquid interface and the solid layer will be kept at a constant melting temperature T_m . The maximum heat flux could theoretically be obtained if the length of the heat exchanger is infinitely large, such that the temperature of the fluid reaches the PCM temperature. ϵ can then be expressed as:

$$\epsilon = \frac{q}{q_{max}} = \frac{T_{in} - T_{out}}{T_{in} - T_m}, \quad (7.1)$$

where T_{in} and T_{out} are the inlet and outlet temperatures of the heat transfer fluid respectively. For a perfect heat sink, the number of transfer units (NTU) is related to ϵ by:

$$\epsilon = 1 - \exp(-NTU), \quad (7.2)$$

where NTU can be determined from the overall heat transfer coefficient (U):

$$NTU = \frac{UA_f}{\dot{m}c_p}. \quad (7.3)$$

A_f is the the heat transfer area of the fluid side, $\dot{m}$ is the mass flow rate and c_p is the specific heat capacity of the fluid. The overall heat transfer coefficient (U) can be determined from a thermal circuit, where the finned surface on the fluid side, the metal wall and the layer of PCM up to the solid/liquid interface are each represented by a thermal resistance circuit, where the individual resistances are placed in series.

Here, we will assume that any effect of buoyancy in the PCM is negligible, so that only conduction is considered in the PCM. As such, for planar surfaces U can be written as:

$$U = \left[\frac{1}{\eta_{0,f} h_f} + \frac{t_w}{\frac{A_w}{A_f} k_w} + \frac{a(t)}{\frac{A_{PCM}}{A_f} k_{PCM}} \right]^{-1}. \quad (7.4)$$

$\eta_{0,f}$ and h_f are the surface effectiveness of a finned geometry and the heat transfer coefficient respectively, for the fluid side. The addition of $\eta_{0,f}$ is necessary to account for the drop in temperature along the height of the fin extending into the fluid, where $\eta_{0,f} = 1$, if no fins are available. t_w and a are the thickness of the wall and the liquid PCM respectively. k is the thermal conductivity and A_i is the heat transfer surface area of each layer i .

Finally, the outlet temperature is estimated from equation 7.1 as:

$$T_{out} = T_{in} - \epsilon(T_{in} - T_m) \quad (7.5)$$

7.2.1.2 Use of extended surfaces in fluid channel

Compact Heat Exchange commonly use extended surfaces (e.g. fins) to increase the amount of surface area, which ultimately leads to an increase in the effectiveness of the heat exchanger. In the case where extended surfaces are employed, there will be a temperature gradient along the fins that will extend into the fluid, where the temperature will be lower close to the PCM wall (assuming a melting case). This will lead to a loss in effectiveness when using fins, such that an additional effectiveness term $\eta_{0,f}$ is added to the overall heat transfer coefficient (see Equation 7.4). Relations have been proposed by Gardner, following the Gardner-Murray assumptions to calculate the surface effectiveness $\eta_{0,f}$:

$$m_f = \sqrt{\frac{2h_f}{k_w \delta_f}}. \quad (7.6)$$

This is valid for plate fin heat exchangers according to [Kays & London \(1964\)](#). This can then be used to calculate the fin efficiency:

$$\eta_f = \frac{\tanh\left(\frac{mb_f}{2}\right)}{\frac{mb_f}{2}}. \quad (7.7)$$

The surface effectiveness is then found as:

$$\eta_0 = 1 - \frac{A_f}{A}(1 - \eta_f). \quad (7.8)$$

7.2.1.3 Stefan problem

As the melting thickness a changes as the material melts, the thermal resistance of the PCM will increase in time. This will lead to a lower overall heat transfer coefficient of the heat exchanger over time. This is a non trivial problem, as this suggests that the position of the melting thickness needs to be known to be able to calculate ϵ and the temperature drop of the heat exchanger.

The time dependence of the overall heat transfer coefficient has been addressed in multiple ϵ -NTU models for PCM in the literature. To get around this problem, [Tay et al. \(2012\)](#) use a PCM heat exchanger with specified dimensions and consider a phase change fraction parameter, rather than a melting thickness. As a result, the problem is no longer time dependent and it is possible to calculate an average effectiveness by considering different values of the phase change fraction. This however suggests that appropriate dimensions for the heat exchanger are already known in advance, as it is necessary to ensure that a sufficient amount of PCM is melted during the prescribed cycle. Only using this approach in the initial heat exchanger sizing phase could therefore lead to the wrong dimensions for a given cycle. This also means that a full performance profile of the heat exchanger over time cannot be obtained by using this method.

[Tian et al. \(2021\)](#) address both these problems in a dynamic model based on energy conservation where the Stefan condition is coupled to the ϵ -NTU model. This results in a differential equation, which can be solved numerically. This makes the model more computationally expensive than the model proposed by [Tay et al. \(2012\)](#). Tian et al. justify this choice as this makes it possible to account for variations in the operating conditions in time. Here, a simplified approach to the one used in [Tian et al. \(2021\)](#) is proposed, where the Stefan condition will be linearised as described in section 7.2.1.3.

Due to the small period of the solidification and melting cycle respectively, the melting thickness around which the heat exchanger will be designed will be relatively small (around 2 mm of melted PCM were observed in the previous simulations in Chapter 5). On top of this, the temperature difference between the melting temperature and the maximum temperature on the fluid side is also relatively small (Here, 10 K). This will lead to a small impact on the melting from convection, such

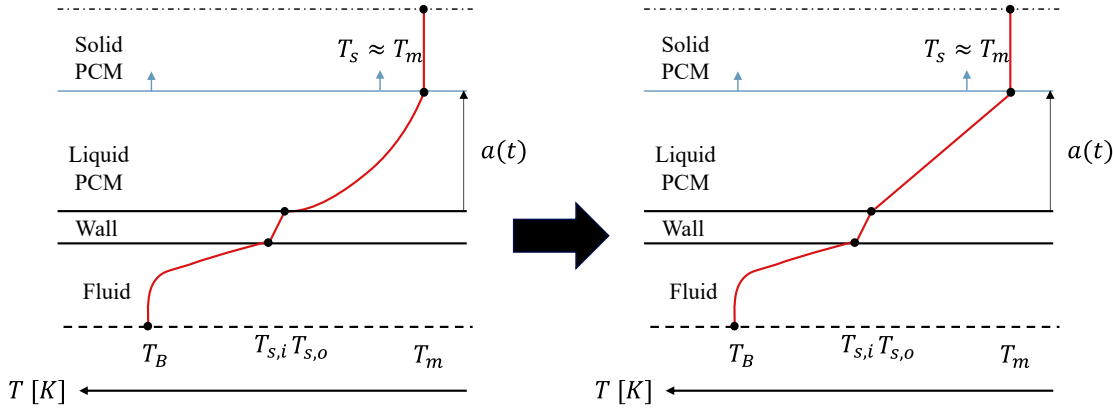


Figure 7.2: Visualisation of the effect of the modelling assumption on the temperature profile compared to the original model

that this can be neglected in the model, as conduction will be the dominant mode of heat transfer.

The melting thickness can therefore be determined from the Stefan Condition for a diffusion only case. The following relationship exists at the solid-liquid interface:

$$-\rho L \frac{da}{dt} = k_1 \left. \frac{\partial T}{\partial x} \right|_{x=a^-} - k_2 \left. \frac{\partial T}{\partial x} \right|_{x=a^+}. \quad (7.9)$$

Previously, it was assumed that the temperature in the solid PCM for the melting problem would stay at the melting temperature, such that Equation 7.9 can be simplified to:

$$-\rho L \frac{da}{dt} = k \left. \frac{\partial T}{\partial x} \right|_{x=a^-}, \quad (7.10)$$

where a is the position of melt front, ρ is the density of the liquid PCM, L is the Heat storage capacity of the material and k is the thermal conductivity of the liquid PCM.

If the velocity of the melting front a is sufficiently low, it is possible to assume that the temperature in the liquid PCM will be in a conductive equilibrium similar to a conduction only case without any moving interfaces (i.e. temperature changes along the height of the liquid PCM is linear, see Figure 7.2). This simplifies the problem to a linear first order ODE, where:

$$-\rho L \frac{da}{dt} = k \frac{T_w - T_m}{a}. \quad (7.11)$$

This assumption is valid as long as the Stefan number (St) is very large, where the

Stefan number is defined as the ratio between latent and sensible heat:

$$St = \frac{L}{c_p(T_w - T_m)} \gg 1. \quad (7.12)$$

In our case $St \in [13; 26]$. By solving Equation 7.11 for the melting thickness a :

$$a(t) \approx \sqrt{\frac{2k(T_w - T_m)t}{\rho L}} \quad (7.13)$$

To validate the use of the linearised Stefan formulation, a 1D CFD model of a cavity containing RT18 HC (see Table 5.3 for material properties) was created. The lower wall of the cavity is heated at a constant temperature T_w and the initial temperature of the PCM is set to $T_0 = 290.14 \text{ K}$. The PCM is subsequently melted for 100 s, and the melting thickness a is extracted from the results (see Figure 7.3). Good agreement is observed between the linear approximation and the 1D CFD results, confirming the validity of the Stefan Condition for approximating the location of the melting front. It is possible to see that a smaller difference between the wall temperature and the melting temperature leads to a thinner layer of melted PCM. This suggests that as the fluid temperature drops over the length of the heat exchanger, the melting thickness will decrease over the length of the heat exchanger.

Equation 7.13 requires the knowledge of the wall temperature T_w to predict the melting thickness. Assuming that the system behaves as a quasi-steady state, we can assume that the heat flux through the different zones of the material is constant, such that the wall temperature can be determined from the bulk temperature of the fluid T_B :

$$q = q_f = q_w = q_{PCM}, \quad (7.14a)$$

$$q = \frac{T_B - T_w}{U_T}. \quad (7.14b)$$

By rearranging for the wall temperature we obtain:

$$T_w = T_B - \frac{q}{A_f U_f + A_w U_w}, \quad (7.15)$$

where U_f and U_w are the heat transfer coefficient contribution of the fluid and wall side respectively. Note here, that to get a good prediction of the melting thickness, it is necessary to know how much heat was absorbed from the working fluid.

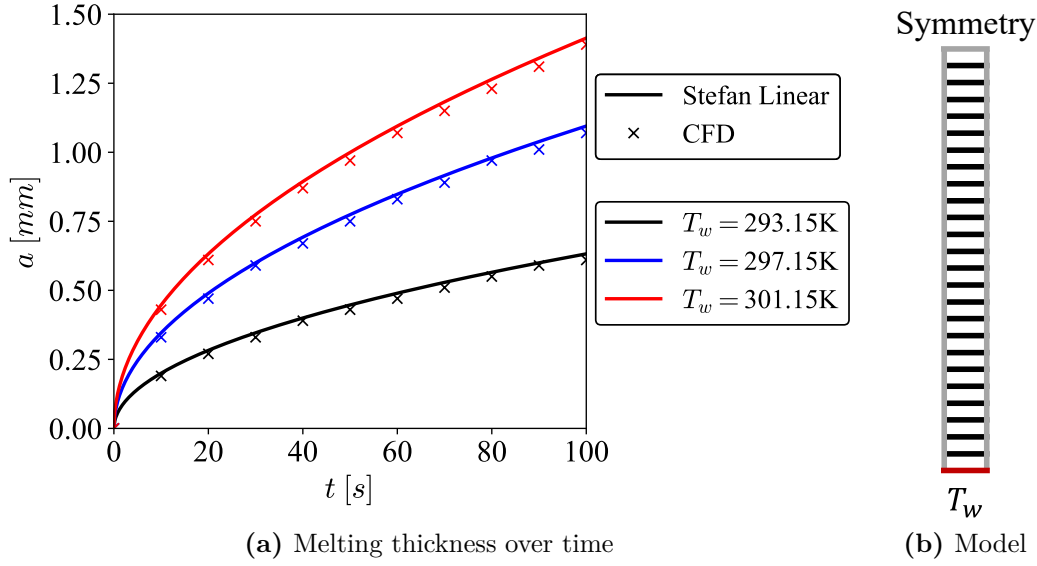


Figure 7.3: Comparison of the melting thickness a between a linear approximation of the Stefan condition and a 1D CFD model (see (b) for geometry). RT18HC is melted over 100 s for different wall temperature T_w by using similar material properties for both models. For the Stefan condition, a sharp melting interface at $T_m = 291.15\text{ K}$ is assumed, while for the CFD a continuous melting interface is used with $T_m = 290.15 - 292.15\text{ K}$.

7.2.1.4 Elemental approach to model drop in melting thickness

As the PCM melts, heat is absorbed from the fluid, so that the fluid experiences a temperature drop over the length of the heat exchanger. As this happens, equation 7.13 predicts a drop in the melted PCM thickness over the length of the heat exchanger.

This was taken into account in the model by splitting the length of the heat ex-

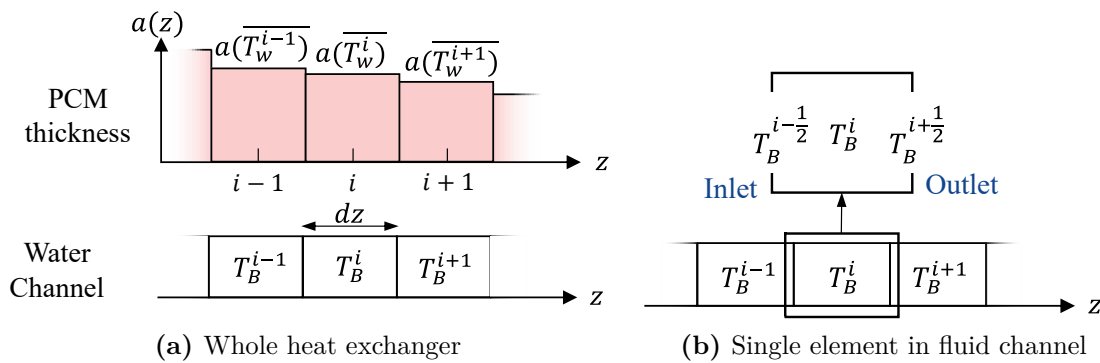


Figure 7.4: Discretization of the length of the heat exchanger into elements.

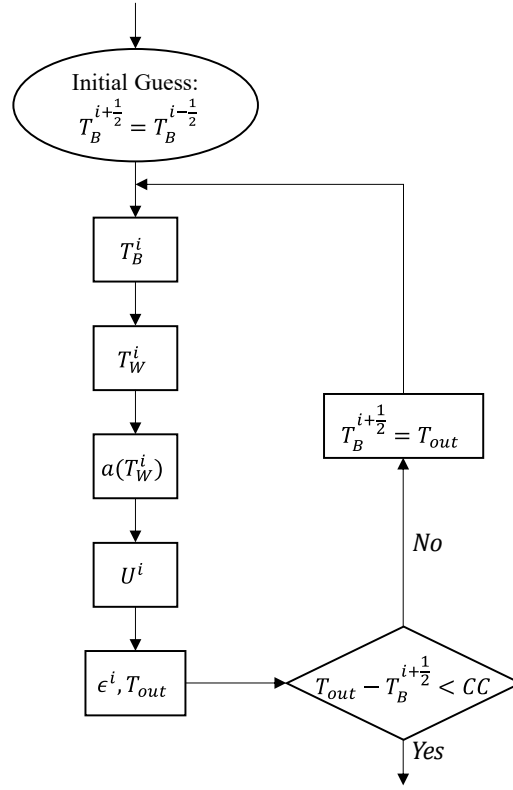


Figure 7.5: Flowchart outlining the different steps of the convergence algorithm within the elemental loop.

changer into elements and by calculating the corresponding melting thickness and temperature drop for each element, similarly to what has previously been done for vapour-compression liquid chillers in [Browne & Bansal \(2001\)](#). Figure 7.4a shows how the domain is split up in elements, where the melting thickness and the bulk temperature of the fluid are calculated at the center of each element. Figure 7.4b highlights the inlet and outlet of each element and the corresponding bulk temperature field of the fluid.

In this case, the bulk temperature of each element is approximated from the temperature at the inlet ($T_B^{i-\frac{1}{2}}$) and the outlet ($T_B^{i+\frac{1}{2}}$) of each element. This is defined from a linear relationship, as:

$$T_B^i = \frac{T_B^{i+\frac{1}{2}} + T_B^{i-\frac{1}{2}}}{2} \quad (7.16)$$

It was previously noted that the fluid temperature and the melting thickness are dependent on each other. To overcome this issue, a convergence loop (see Figure 7.5) is introduced in the algorithm, where the appropriate wall temperature and melting thickness is calculated from an initial guess. For each element, the outlet

temperature ($T_B^{i+\frac{1}{2}}$) is initially set to the inlet temperature of the current element ($T_B^{i-\frac{1}{2}}$) (or the outlet temperature of the previous element). This is used to estimate the wall temperature (T_W) to calculate the melting thickness (a). From this a new outlet temperature (T_{out}) is measured and the value is compared to the initial guess. If this value is above a chosen Convergence Criterion (CC), $T_B^{i+\frac{1}{2}}$ is set to the outlet temperature calculated from the ϵ -NTU model (T_{out}), and the loop is repeated. Once CC is satisfied, the last calculated values are recorded and the code moves on to the next element.

7.2.1.5 Time dependency of phase change problems

Over time, the thickness of the PCM in each element will increase, leading to a higher resistance to the heat transfer, such that the overall heat transfer coefficient of the heat exchanger element will decrease over time. By considering the different relationships described in Section 7.2, it can be shown that this will decrease the temperature drop in each element in time, in line with what was observed in the various heat exchanger geometries studied in Chapter 5.

In the current model, the dependence of the temperature field inside of the fluid channel was taken into account by adding an additional loop representing time. The model uses an analytic solution to predict the amount of PCM that is melted in each element, which requires a constant wall temperature as an input. An average value of the wall temperature over time is calculated for each element at each time step, which is then used to predict the melting thickness.

7.2.1.6 Summary of the model

A model to determine the amount of stored energy in PCM compact heat exchanger was developed from two separate models. The ϵ -NTU model that is commonly used to predict heat exchanger performance in industry was combined with a linearised formulation of the Stefan Problem that can be used the location of the phase change interface for simple boundary and initial conditions. Figure 7.6 provides a full overview of the PCM- ϵ -NTU model that was coded into MATLAB.

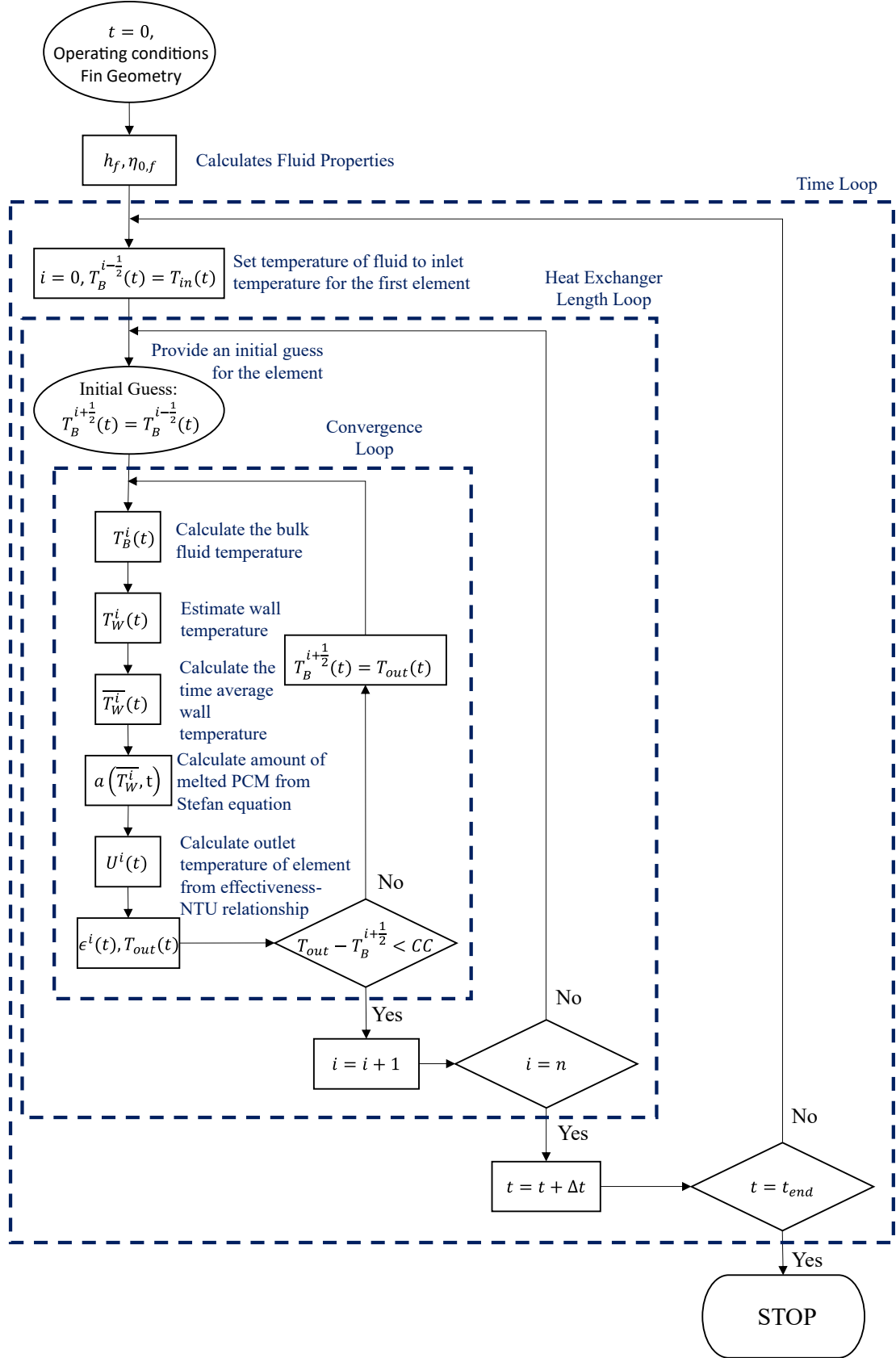


Figure 7.6: Flowchart presenting the full PCM- ϵ -NTU model.

7.2.2. Validation of effectiveness-NTU model

The strip-fin compact heat exchanger geometry considered in Chapter 5 (see Figure 7.7) was used as an example geometry to validate the model. For this, similar operating conditions were used to what is described in Chapter 5, where the non-finned PCM layer is filled with RT18HC and water circulates in the finned fluid channel entering the heat exchanger at $T_{in} = 301.15 \text{ K}$. For the purpose of this study, an average melting temperature of 291.15 K was considered for RT18HC for the ϵ -NTU model, while the full melting range was considered for the CFD.

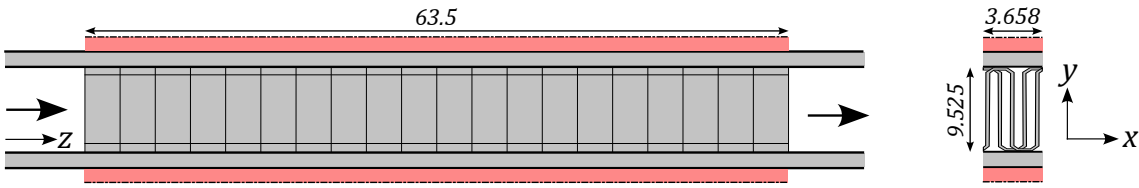


Figure 7.7: Strip Fin heat exchanger geometry with 1/8-13.95 fins in fluid layer. (Dimensions are in mm).

7.2.2.1 Heat transfer coefficient prediction for fluid

The ϵ - NTU model requires a prediction of the heat transfer coefficient in the fluid channel, which can be obtained from experimental results from j vs Re curves or by conducting CFD simulations. In this case, previous CFD results as computed in Chapter 5 were used, as reproduced in Figure 7.8 . The formulation for the j factor for the laminar and turbulent regimes are as follows:

$$\begin{aligned} j &= 2.079Re^{-0.711} & Re < 850, \text{ Laminar Flow,} \\ j &= 0.071Re^{-0.212} & Re \geq 850, \text{ Turbulent Flow.} \end{aligned} \quad (7.17)$$

From this, the heat transfer coefficient (h) can be predicted from the Stanton number (St):

$$St = \frac{j}{Pr^{\frac{2}{3}}}, \quad (7.18)$$

$$h = \frac{St\dot{m}c_p}{A_f}, \quad (7.19)$$

where $\dot{m}$ is the mass flow rate of the fluid, c_p is the specific heat of the fluid, and A_f is the heat transfer area of the fluid.

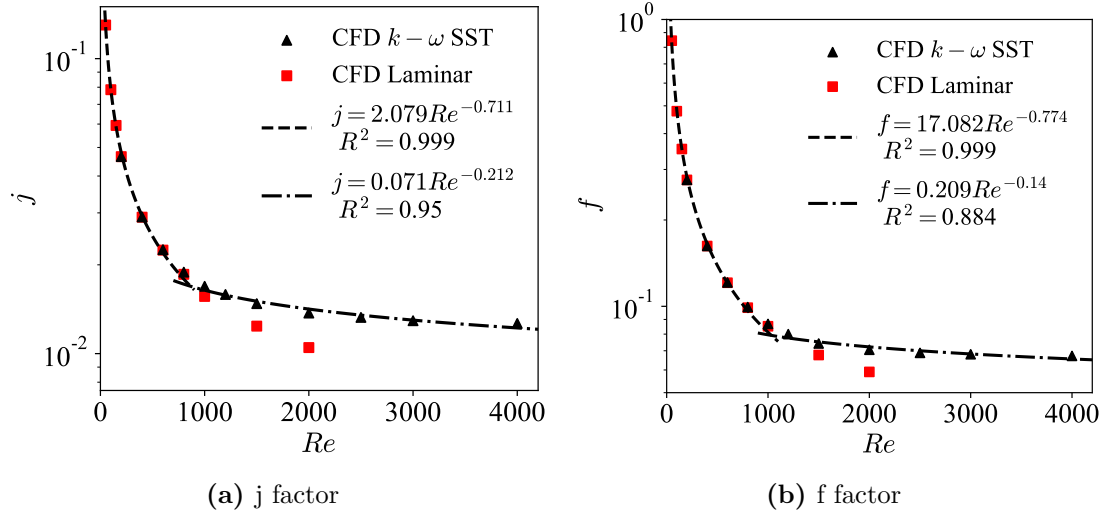


Figure 7.8: Correlations for j and f factor for 1/8-13.95 fins obtained from previous CFD results in Chapter 5.

7.2.2.2 Mesh and time step Independence Study

A mesh and time step independence study was conducted at $Re = 200$ by varying the number of elements between 5 and 50 and by varying the time step between 0.1 and 0.01 s (see Figure 7.9). From this, 20 elements and a time step size of 0.02 s were found to be sufficient to resolve the temperature drop and melting thickness. For a single run, for mesh and time independent results, the algorithm took 2.3 s to execute (excluding writing results and plotting, with the MATLAB code running on a Dell Precision 3581 machine with 13th Gen i7-13700H processor). This represents

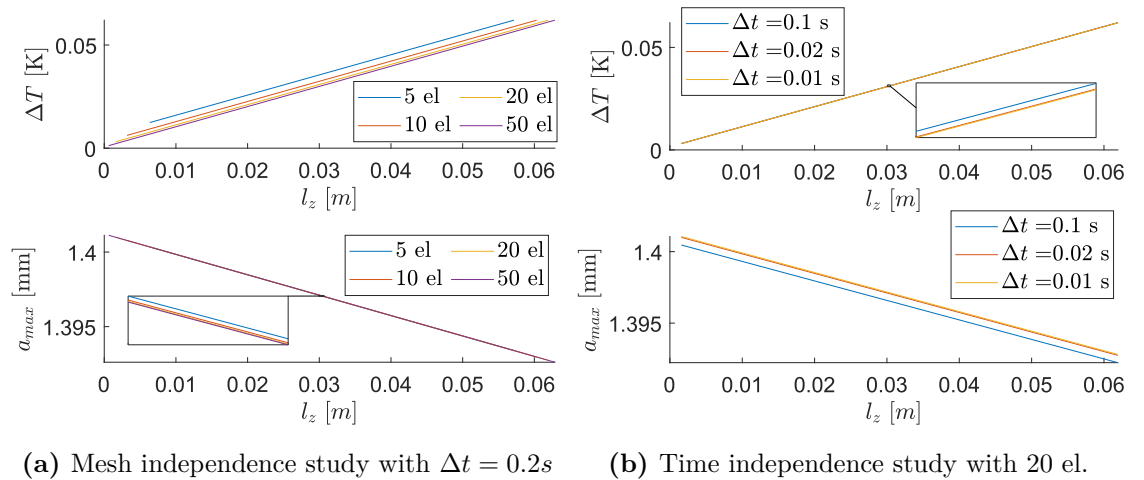


Figure 7.9: Mesh and Time independence study of PCM- ϵ -NTU model for Strip Fin geometry

a significant decrease in the computational cost when compared to simulating the melting of PCM with CFD.

7.2.2.3 Comparison with CFD

Figure 7.10 compares CFD results obtained in Chapter 5 with results from the $\epsilon - NTU$ model over time at $Re = 200$. A close agreement is observed between the $\epsilon - NTU$ model and the CFD results over time for the average properties as well as the location of the phase change interface over time. In Figure 7.10.a, the liquid fraction in the heat exchanger is tracked over time. At the beginning of the melting

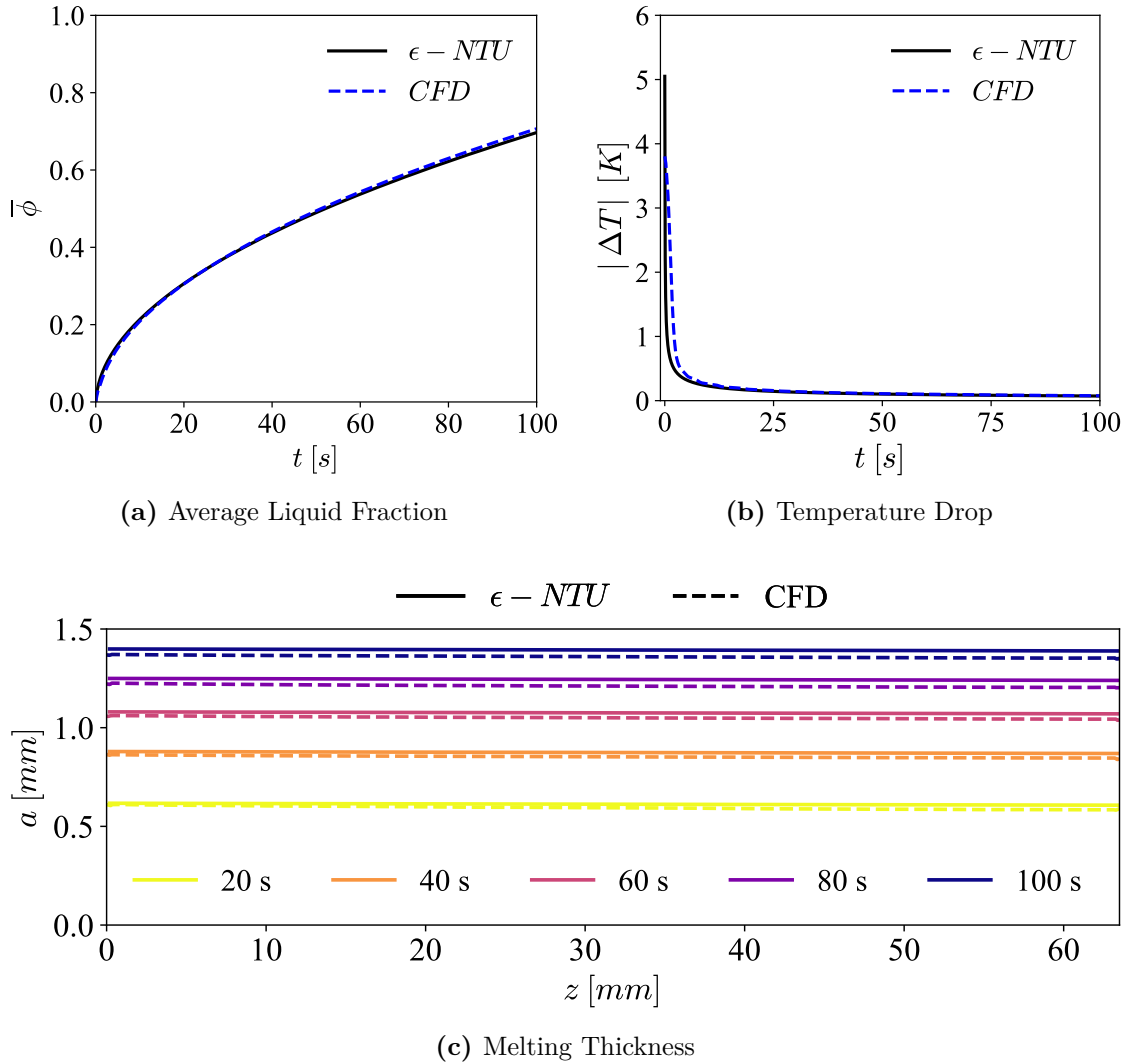


Figure 7.10: Comparison of $\epsilon - NTU$ and CFD model results over time at $Re = 200$: (a) Average Liquid Fraction, (b) Temperature Drop, (c) Local melting thickness.

cycle where $t < 20s$, the $\epsilon - NTU$ model over-predicts the liquid fraction, while for $t > 40s$, the semi-analytical model under-predicts the liquid fraction giving an average percentage difference of 0.39% from the CFD results. Figure 7.10.b shows the average temperature drop over time. Here, the largest difference is observed close to the initial condition, where the $\epsilon - NTU$ predicts a higher temperature drop before the temperature drop rapidly decreases. For the CFD results, a transient response is observed in the initial seconds, as the heat transfer fluid adapts to the changes from the initial condition. Following the transition phase, the temperature drop prediction from the $\epsilon - NTU$ model starts approaching the CFD results. Figure 7.10.c compares the melting thickness from the ϵ -NTU model with CFD results at $Re = 200$. From this, it is possible to see that the ϵ -NTU is capable of accurately predicting the amount of melted PCM over time, where the local prediction of the melting thickness is within 5% of the CFD results. The greatest difference with the CFD results is observed close to the outlet, linked to the smaller drop in the melting thickness that is calculated over the length. This suggests that as the length of the heat exchanger is increased, a greater difference in the results could be observed.

The output over a larger range of Reynolds numbers is given in Figure 7.11 for the melting thickness at $t=100s$ and the temperature drop over time. Figure 7.11.a shows that the Stefan model consistently over-predicts the melting thickness for different Reynolds number with a maximum deflection of 3.6 % from the numerical

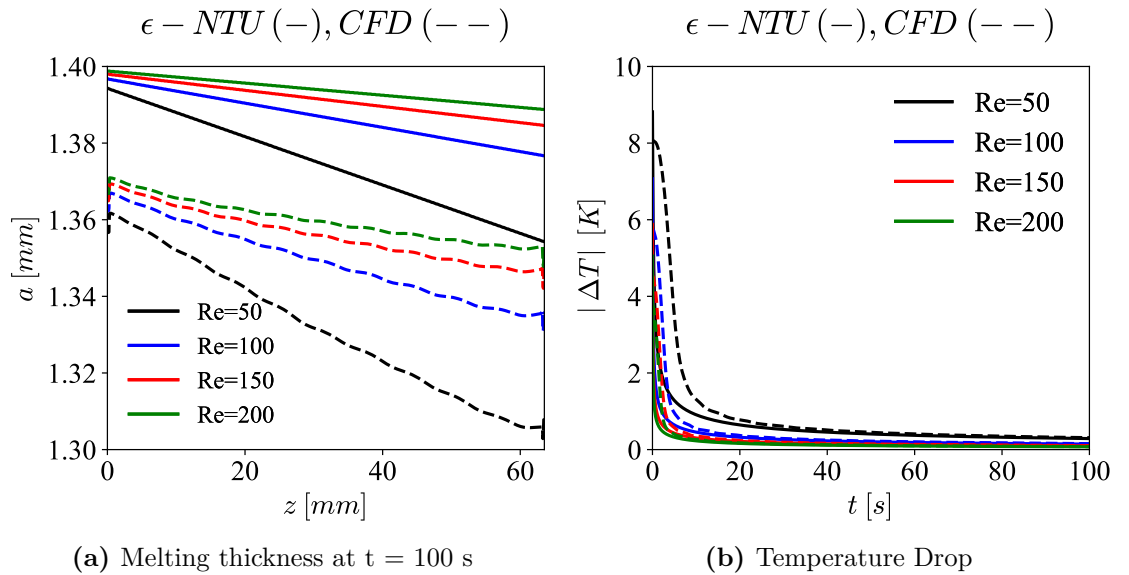


Figure 7.11: Comparison of ϵ -NTU model (—) results with CFD results (---) over range of Reynolds number for Strip Fin Geometry.

results. The results show that the drop in the melting thickness over the length of the heat exchanger follows a similar trend over a variation of Reynolds number. From the temperature drop comparison in Figure 7.11.b, it is possible to see that the model predicts a similar trend of the temperature to the CFD results, where the temperature drop decreases over time as the thermal resistance of the PCM increases. The ϵ -NTU initially predicts a higher temperature drop to the CFD and a faster decrease in the temperature drop in the first 5-20 seconds, depending on the Reynolds number.

The $\epsilon - NTU$ model has a few shortcomings which could explain the difference in results. The model can only take into account constant material properties for the PCM, such that a melting range cannot be used for the model. The prediction of the temperature drop is dependent on the accuracy of the heat transfer coefficient prediction, which has to be obtained separately, either by experiments or simulations. Finally, the most likely reason for the observed discrepancies, due to the quasi-steady state nature of the $\epsilon - NTU$ model, variations in the initial temperature of the PCM and the transient response of the metal and fluid layers of the heat exchanger are neglected.

Overall, the model is capable of predicting the amount of melted PCM over time with reasonable accuracy and therefore the amount of heat that will be absorbed by the heat exchanger for the current operating conditions.

7.3. Alternative approach based on energy conservation

The $\epsilon - NTU$ model based on the linearised Stefan approximation developed in the previous section provided accurate results for predicting the amount of melted PCM over the length of the heat exchanger at a reasonable computational cost. However, the simplification is only valid for planar 1D cases, such that its application is limited, and the model is hard to adapt to finned PCM layers. This justifies the development of a more general case scenario capable of predicting similar results. A different approach for predicting the amount of melted PCM by using an energy conservation will be presented in this section.

7.3.1. Development of effectiveness-NTU model based on energy conservation

In the energy conservation approach, it is assumed that all of the heat released from the fluid will be absorbed by the PCM. The model follows a similar approach to what is proposed by Tian et al. (2021). For this, the volume of melted PCM is tracked over time to ensure that energy is properly conserved based on the diagram shown in Figure 7.12. Additionally to Tian et al. (2021)'s model, where only latent heat was absorbed by the PCM, the current model absorbs sensible heat by assuming that the PCM melts in a quasi-steady state.

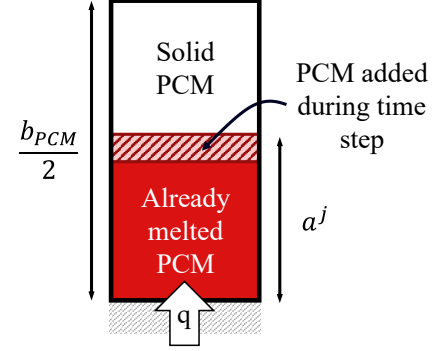


Figure 7.12: Schematic of the energy conservation model.

7.3.1.1 Predicting the melting thickness at each time step

The energy conservation is defined by assuming that all the energy that is released from the working fluid (E_{released}) is fully absorbed by the melted PCM in the form of latent heat (E_{latent}) and sensible heat (E_{sensible}):

$$E_{\text{released}} = E_{\text{absorbed}}, \quad (7.20a)$$

$$q_f dt = dE_{\text{latent}} + dE_{\text{sensible}}, \quad (7.20b)$$

where q_f is the heat flux absorbed from the fluid over a small interval of time dt . The change in latent heat and sensible heat are defined as follows:

$$dE_{\text{latent}} = \rho L \frac{dV}{dt}, \quad (7.21a)$$

$$dE_{\text{sensible}} = \underbrace{\rho c_p \left(\frac{dV}{dt} + V^{j-1} \right) (T^j - T^0)}_{\text{Sensible heat at current time step j}} - \underbrace{\rho c_p V^{j-1} (T^{j-1} - T^0)}_{\text{Sensible heat at previous time step j-1}}. \quad (7.21b)$$

Where V is the volume of melted PCM and T is the average temperature of the melted PCM. This can then be rearranged in terms of the change in volume and previous volume of melted PCM:

$$dE_{\text{sensible}} = \rho c_p \left[\frac{dV}{dt} (T^j - T^0) + V^{j-1} (T^j - T^{j-1}) \right]. \quad (7.22)$$

Assuming that the velocity of the melting front remains small, the temperature over the height of the liquid PCM will be linear such that the average temperature of the liquid PCM can be calculated as:

$$T(t) \approx \frac{(T_{PC} + T_w)}{2} \quad (7.23)$$

The energy balance can then be rearranged in terms of the volume of melted PCM as:

$$\frac{dV}{dt} = \frac{q_f dt - \rho c_p V^{j-1} (T^j - T^{j-1})}{\rho (L + c_p (T^j - T^0))} \quad (7.24)$$

which can be used to determine the liquid fraction:

$$\phi_{\text{added}} = \frac{\frac{dV}{dt}}{V_{\text{PCM,total}}}. \quad (7.25)$$

The mass fraction of PCM that is melted over time (ϕ_{live}^j) is then increased at each time step by the added part of the melt fraction that was calculated from the heat flux:

$$\phi_{\text{live}}^j = \phi_{\text{live}}^{j-1} + \phi_{\text{added}}. \quad (7.26)$$

Finally, for a square PCM cavity, the melting thickness a is then calculated at each time step as:

$$a^j = \phi_{\text{live}}^{j-1} \frac{b_{\text{PCM}}}{2}, \quad (7.27)$$

where b_{PCM} is the plate spacing of the PCM.

7.3.1.2 Algorithm of new PCM effectiveness-NTU model

Figure 7.13 shows a flowchart outlining the different steps of the algorithm in detail. Similarly to the linearised Stefan model, the length is discretized into smaller elements, from which the local thickness of the PCM can be determined. The primary advantage of this methodology when compared to the linearised Stefan model is that there is no longer a requirement for a convergence loop, as there is no need for predicting the wall temperature. This also means that the mesh and time step can be significantly larger, further reducing the computational cost of the model. For mesh

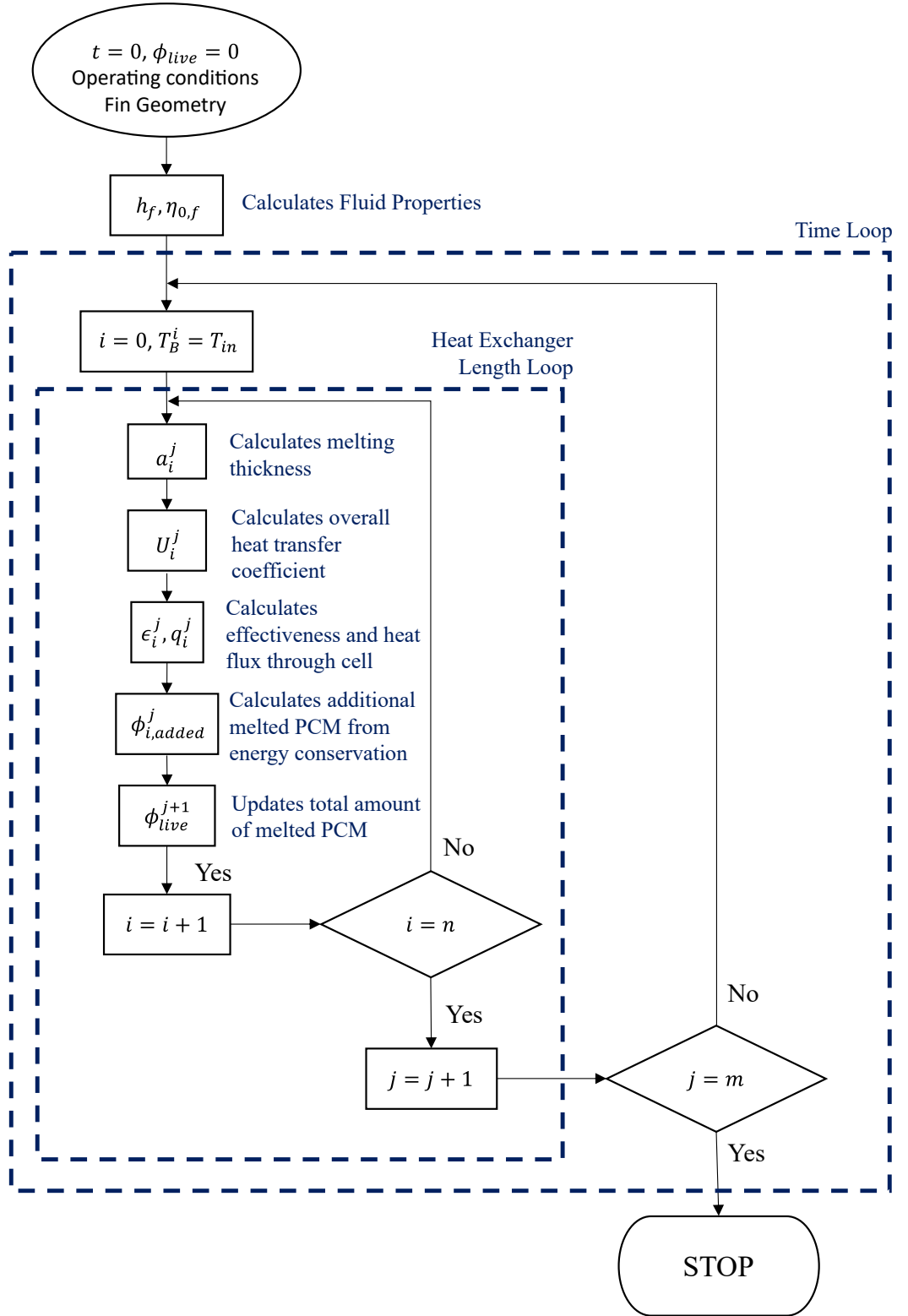


Figure 7.13: Flowchart of the new PCM ϵ -NTU model based on Energy Conservation

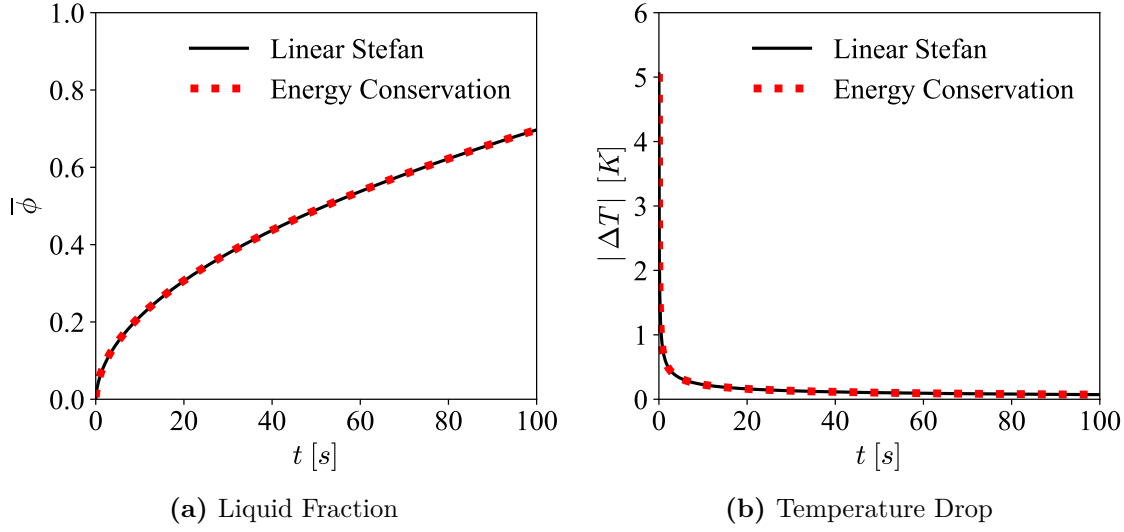


Figure 7.14: Comparison of Energy conservation model with Linear Stefan ϵ -NTU model for offset strip fin geometry at $Re=200$.

and time independent results, the model is capable of predicting the results for the offset strip fin geometry described above in 0.062 s excluding any data saving and plotting. As we no longer rely on a simple equation to predict the melting thickness, it is now also possible to start from a prediction where some of the PCM is already melted, by varying the live melt fraction (ϕ_{live}) at the beginning of the simulation.

7.3.2. Comparison with linear Stefan effectiveness-NTU model

Figure 7.14 compares the new model to the linear Stefan ϵ -NTU model for the offset strip fin geometry at $Re = 200$. This shows that the new model based on energy conservation generates close to identical results to the previously proposed ϵ -NTU model for both the average liquid fraction and temperature drop.

7.4. Incorporating fins into the effectiveness-NTU model

Extended surfaces are used to increase the surface area in conventional heat exchanger, where this can lead to an increase in the effectiveness of the heat exchanger. Similar observations have been made in PCM heat exchanger, where the addition of fins was shown to lead to an increase in the melting rate. On top of this, fins help in increasing the strength of the heat exchanger. This motivates the addition of fins

within the $\epsilon - NTU$ model. In this section, the model for calculating the melting rate within finned cavities will first be developed by isolating the PCM cavity from the rest of the heat exchanger. Following the validation of the model against CFD results for an isolated cavity, the finned cavity model will be incorporated into the remaining PCM compact heat exchanger model and the prediction of the melting will be compared against CFD results for the full heat exchanger.

7.4.1. Treatment of finned PCM cavity

7.4.1.1 Fin efficiency and extended surface efficiency

For the case where the layer is being heated, the fin cools along its length as it gets further away from the plate. As a result, the fin surface temperature will be lower than the temperature at its base, i.e. $T(x) < T_0$. This is taken into account in the conventional ϵ -NTU method by using the fin efficiency η_f and the extended surface efficiency η_0 .

In this section, η_f and η_0 will be derived for a case where the layer contains a PCM, rather than a fluid, for which derivations already exist. This is covered extensively in [Shah \(2003\)](#) and this will be used as a first basis for deriving the finned PCM model. The remaining work will assume that the PCM is being melted, although this should be relatively analogous to a solidification case.

The following assumptions, as well as their validity in the scope of this work are discussed here:

1. The PCM melts in a quasi steady system. This has an impact on the following:
 - (a) The movement of the melt front has a negligible impact on the heat transfer in the fins. This is a valid consideration for a majority of the melting cycle as the melting rate will be small, due to the low thermal conductivity of the PCM.
 - (b) It is assumed that the fins don't store any heat over time. This should be a decent enough approximation as the volume of fins is small when compared to the PCM, such that the sensible heat stored in the fins is negligible when compared to the sensible and latent heat stored in the PCM.
2. The thickness of the liquid pool is relatively uniform over the length of the fin. For a majority of the melting process, this is observed as the melting front

remains relatively perpendicular to the fin surface.

3. The mushy zone region is relatively small, such that a distinct melting front can be defined, where the melting front is maintained at a constant temperature and acts as a perfect heat sink.
4. There is one dimensional heat conduction in the fin (the "thin" fin assumption), such that the the temperature in the fin is only a function of x .
5. It is assumed that the cross-sectional area A_k of the fin is constant. Derivations for nonuniform fin areas for conductive cases are however available, such that it might be possible to extend this derivation to non uniform cross sections.
6. Radiation heat transfer is neglected and material properties of the fins are constant.

Temperature Distribution in the Fin

To start off with, let's consider the diagram in Figure 7.15, showing a finned PCM layer. The fin has a length l , thickness δ and width L_f . At the base ($x = 0$), the temperature of the fin is T_0 and the temperature T along the length of the heat exchanger decreases.

Assuming steady state conditions following similar conditions described previously, and no energy being generated by the fin, we obtain the following relationship:

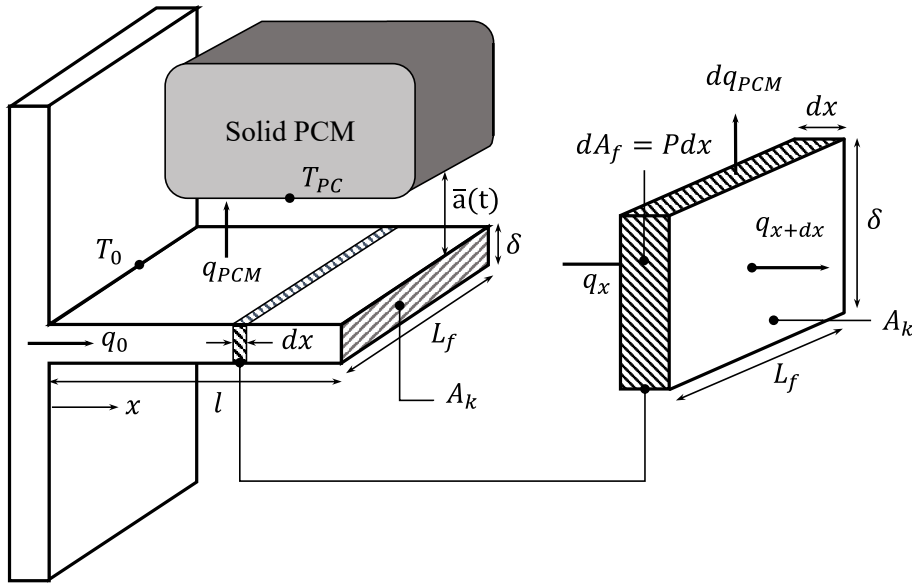


Figure 7.15: Diagram of a finned PCM layer showing a detailed view of an element of the fin.

$$q_x - q_{x+dx} - dq_{PCM} = 0, \quad (7.28)$$

where

$$dq_{PCM} = k_{PCM} dA_f \frac{T - T_{PC}}{\bar{a}}. \quad (7.29)$$

We can then obtain the following differential equation:

$$k_f A_k \frac{d^2 T}{dx^2} - k_{PCM} P \frac{T - T_{PC}}{\bar{a}}. \quad (7.30)$$

By substituting the temperature difference $\theta = T - T_{PC}$ between the fin and the phase change temperature T_{PC} , we obtain the following differential equation:

$$\frac{d^2 \theta}{dx^2} - \frac{k_{PCM} P}{k_f A_k \bar{a}} \theta = 0. \quad (7.31)$$

We can then define the fin effectiveness parameter m as:

$$m^2 = \frac{k_{PCM} P}{k_f A_k \bar{a}} = \frac{2k_{PCM}(L_f + \delta)}{k_f L_f \bar{a} \delta} \approx \frac{2k_{PCM} L_f}{k_f L_f \bar{a} \delta} \quad (7.32)$$

and substitute this into the differential equation giving:

$$\frac{d^2 \theta}{dx^2} - m^2 \theta = 0 \quad (7.33)$$

with a solution of the form of:

$$\theta(x) = C_1 e^{mx} + C_2 e^{-mx}. \quad (7.34)$$

Note that this is the same differential equation that is solved when the convective heat transfer is considered. By applying the following BC, corresponding to a fin with adiabatic tip as applicable to most plate fin compact heat exchanger surfaces, we obtain:

$$\begin{aligned} \theta_0 &= T_0 - T_{PC}, \text{ at } x = 0, \\ \frac{d\theta}{dx} &= 0, \text{ at } x = l. \end{aligned} \quad (7.35)$$

By solving for the constants, the following solution is obtained for the temperature distribution in the fin:

$$\frac{\theta}{\theta_0} = \frac{\cosh m(l - x)}{\cosh ml} \quad (7.36)$$

Heat Flux to PCM

To get the total heat transfer rate to the PCM:

$$dq_{PCM} = \frac{k_{PCM} P dx}{\bar{a}} \theta \quad (7.37)$$

where the total conductive heat transfer rate q_{PCM} can be obtained by integrating over the length:

$$q_{PCM} = \frac{k_{PCM} P dx}{\bar{a}} \int_0^l \theta dx. \quad (7.38)$$

By solving the integral we find the following solution:

$$q_{PCM} = m k_f A_k \theta_0 \tanh ml = \frac{k_{PCM} P}{\bar{a} m} \theta_0 \tanh ml. \quad (7.39)$$

Fin Efficiency

The fin efficiency is defined as the ratio between the actual heat flux through the fin q_0 to the heat flux for a perfect fin q_{max} , or in other words, the heat flux for a fin with infinitely large thermal conductivity k_f , such that $T(x) = T_0$:

$$q_{max} = \frac{k_{PCM} P l}{\bar{a}} (T_0 - T_{PC}). \quad (7.40)$$

This gives a fin efficiency η_f for a thin fin with adiabatic tip and uniform rectangular cross section:

$$\eta_f = \frac{\tanh ml}{ml}, \quad (7.41)$$

where the length scale l for rectangular and triangular fins as shown in Figure 7.16 is:

$$l = \frac{b - \delta}{2} \approx \frac{b}{2}. \quad (7.42)$$

Extended Surface Efficiency

For a finned surface in a heat exchanger it is also important to take the primary surface (plate of surface area A_p) into consideration. This can be done through the extended surface efficiency η_0 :

$$\eta_0 = \frac{q_{total}}{q_{max}} = 1 - \frac{A_f}{A} (1 - \eta_f), \quad (7.43)$$

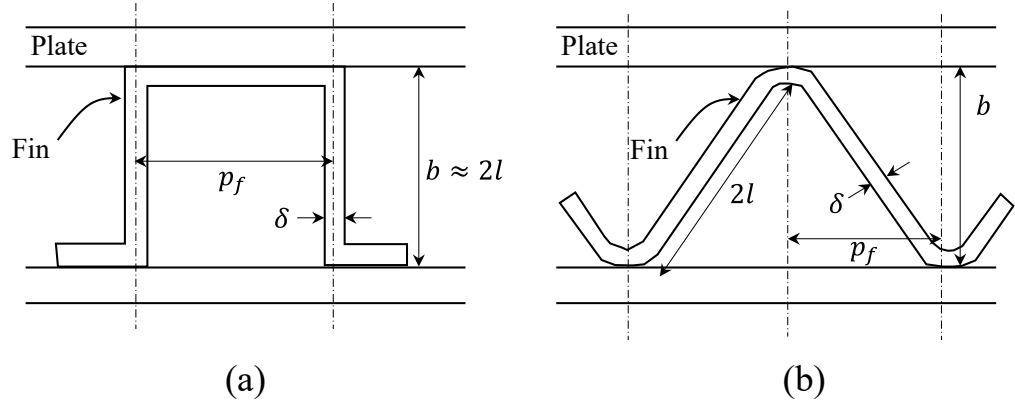


Figure 7.16: Choice of length scales for different fin geometries: (a) Idealized plain rectangular fin, and (b) Plain triangular fin. (Figure adapted from [Shah \(2003\)](#)).

where A is the total area such that $A = A_p + A_f$. Note that as the fin efficiency η_f is a function of the melting thickness $\bar{a}(t)$, the extended surface efficiency η_0 will also be a function of time, where the efficiency will increase as more of the PCM is melted.

7.4.1.2 Use of appropriate shape factor

Figure 7.17 shows the shape and position of the melt front for a square and triangular cavity over time with both convection and conduction only conditions. This shows

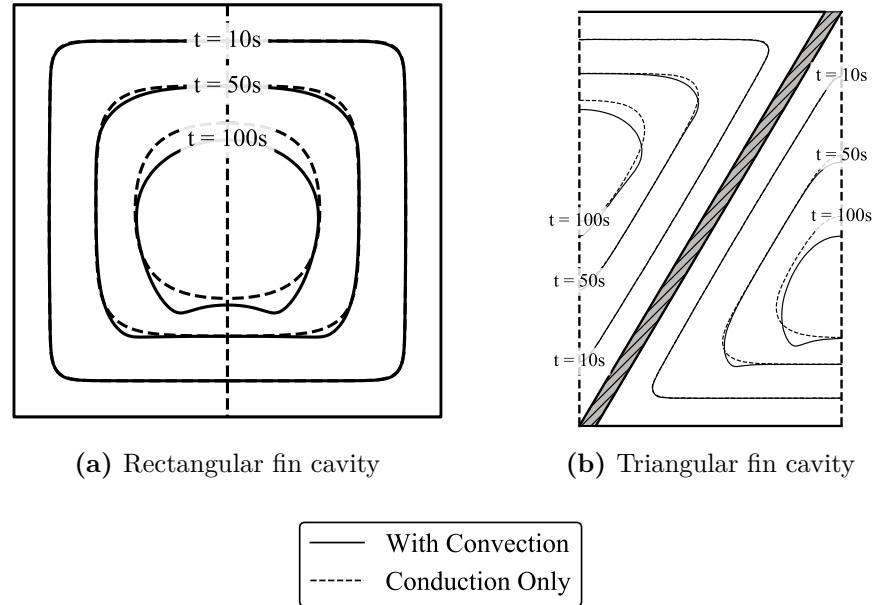


Figure 7.17: Shape of the melt front for CFD results for two common fin geometries at $AR = 1$, $T_w = 301.15K$ and $A_c = 25 \text{ mm}^2$.

that early in the melting process, the shape of the cavity is well respected, although the corners are more rounded. In the conduction dominated case of the square cavity, the shape of the melting front resembles that of a squircle, i.e. a super-ellipse with a factor $n=4$, where as the PCM melts further, the corners become more rounded, such that it approaches a circular shape.

As the heat transfer now occurs in a 2D space, the 1D heat transfer equations that was solved in the previous sections based on the melting thickness will no longer be valid. A shape factor is often used to model complex 2D and 3D systems with 1D heat transfer equations, where the shape factor S is defined as:

$$S = \frac{1}{kR} = \frac{Q}{k(T_0 - T_\infty)}. \quad (7.44)$$

The solutions for shape factors for two dimensional systems are limited to relatively simple geometries of circular or regular polygonal systems, as the analysis for more complex shapes becomes rapidly more challenging. On top of regular shapes (see [Rohsenow et al. \(1998\)](#) for a summary of relationships), the relationship by [Hassani et al. \(1993\)](#) provides a shape factor for a uniform thickness two-dimensional layer on cylinders of the same shape:

$$\frac{S}{L} = \frac{2\pi}{\ln [1 + (2\pi\delta/P_i)]}, \quad (7.45)$$

where δ is the distance between the two shapes and P_i is the perimeter of the inner shape. As it is relatively complicated to capture the superelliptical shape of the melting front in the simplified model, it was instead decided to assume that the melting front has a similar shape to the cavity, such that the uniform thickness assumption is respected. This makes it possible to use the relationship proposed by [Hassani et al. \(1993\)](#). In this case, the uniform thickness δ will be the melting thickness $\bar{a}$ and the perimeter P_i can be calculated from $\bar{a}$. Similarly to the extended surface efficiency parameter, the shape factor will change as more or less of the PCM is melted and $\bar{a}$ and P_i will vary.

7.4.1.3 Illustration of model based on rectangular and triangular cavities

Problem Description

To illustrate the analytical model, the two most common fin geometries, i.e. rectangular and triangular shapes are applied, based on the diagram shown in Figure

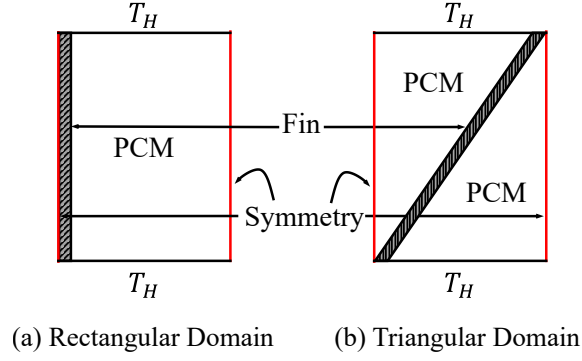


Figure 7.18: Geometry considered for testing the surface effectiveness model.

7.18 similar to the problem that was described in Chapter 6.

For the rectangular cavity, different studies are conducted on the effect of aspect ratio AR as well as surface area A_{PCM} following the same dimensions and wall temperature T_w inputs as for the CFD presented in Chapter 6.

For the triangular cavity, only the effect of aspect ratio is considered for a cavity with a constant free flow area of $A_c = 25 \text{ mm}^2$ and a wall temperature of $T_w = 301.15 \text{ K}$. Additional simulations were run by using a similar setup to what was described in Chapter 6. for a triangular fin geometry in Ansys FLUENT.

Problem setup in MATLAB

The code outlined in Figure 7.19 is implemented in MATLAB, where the fin effectiveness model presented above is used to determine the surface effectiveness η_{PCM} from the average melting thickness $\bar{a}$ at each time step i . The shape factor relationship in Equation 7.44 is used to determine the thermal resistance of the PCM R_{PCM} , which is then applied to calculate the heat flux through the wall q :

$$q = \frac{T_w - T_m}{R_{PCM}}. \quad (7.46)$$

The change in volume of melted liquid PCM dV/dt is then determined from energy conservation, based on a simplified formulation of the model described in Section 7.3. Similarly to previously, it is assumed that the energy that is released through the wall is absorbed by the PCM in form of latent and sensible heat based on:

$$qdt = dE_{latent} + dE_{sensible}. \quad (7.47)$$

As the wall temperature is now a constant, Equation 7.22 can be simplified, such that:

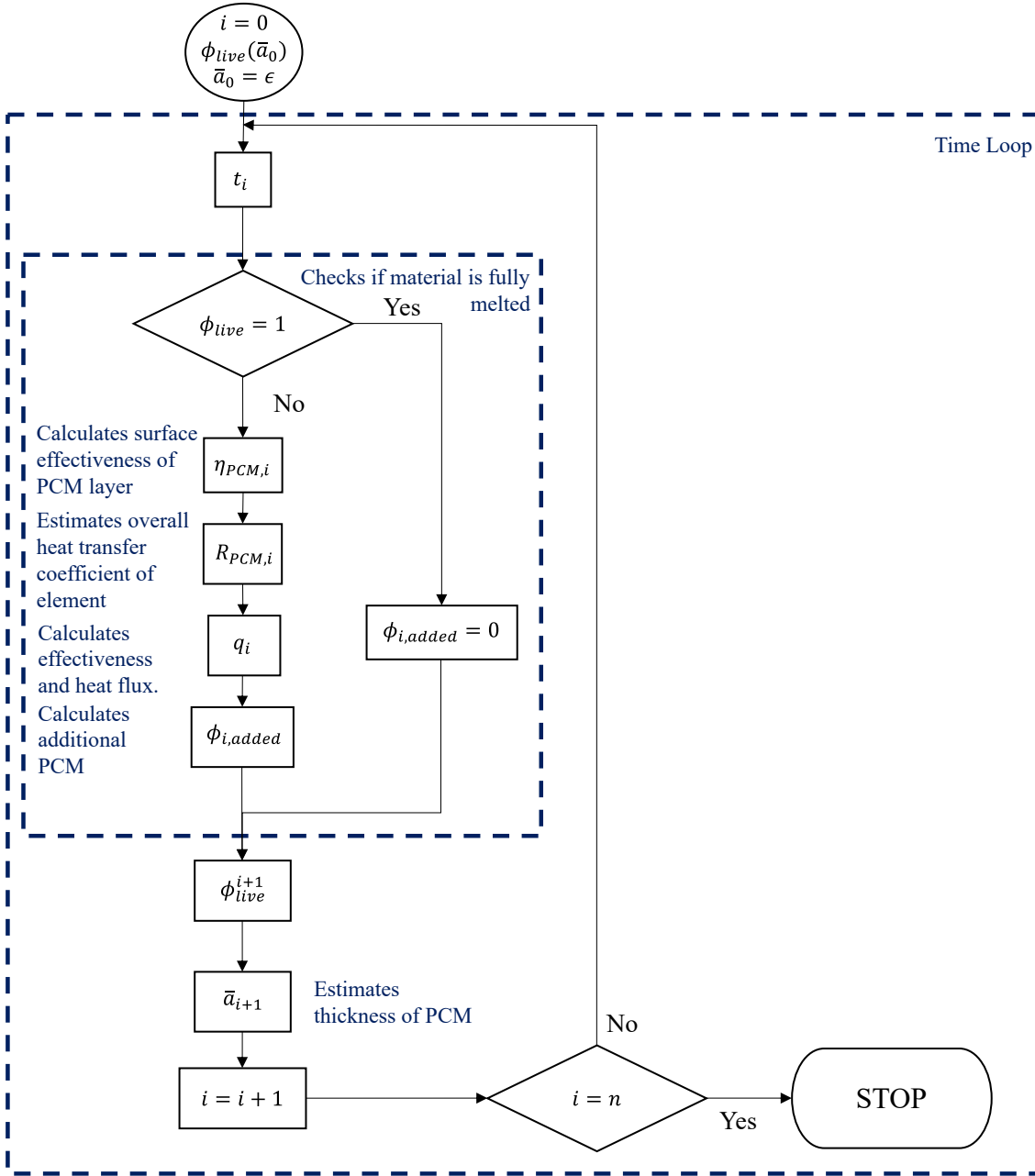


Figure 7.19: Flowchart of analytical model of fins in PCM

$$qdt = \rho \frac{dV}{dt} [L + c_p (T - T^0)] , \quad (7.48)$$

where T^0 is the initial temperature of the PCM and T is the average temperature of the PCM, where based on the quasi steady assumption:

$$T \approx \frac{1}{2} (T_{PC} + T_w) . \quad (7.49)$$

Note that T is a constant, as the wall temperature remains constant as well. Follow-

ing this, the liquid fraction is updated and the average melting thickness is calculated for the next time step, similar to the remaining energy conservation model.

Table 7.1: Summary of geometric parameters for the determination of the shape factor for different cavity geometries. The red shape for each geometry represents the simplified geometry of the melting front used within the model.

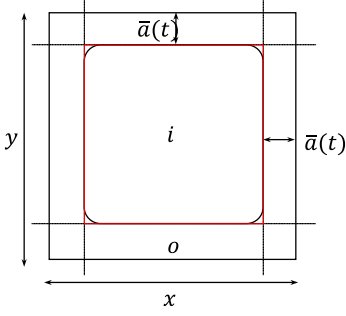
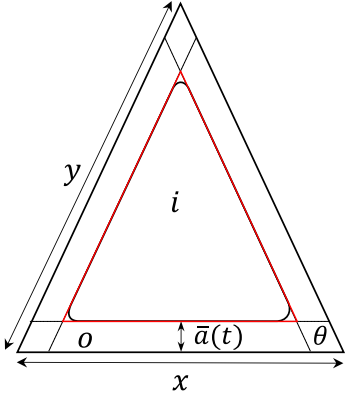
Fin Cavity Shape	Geometric Parameters
Rectangular 	$A_{PCM} = P_i = 2(x + y - 4\bar{a}) \quad (7.50)$ $A_i = (x - 2\bar{a})(y - 2\bar{a}) \quad (7.51)$ $A_c = A_o = xy \quad (7.52)$ $\phi = \frac{A_o - A_i}{A_o} \quad (7.53)$
Triangular 	$AR = \frac{x}{y} \quad (7.54)$ $\theta = \cos^{-1} \frac{x}{2y} \quad (7.55)$ $A_{PCM} = P_i = \left[x - 2\bar{a} \left(\frac{1}{\sin \theta} + \frac{1}{\tan \theta} \right) \right] + 2 \left[y - 2\bar{a} \left(\frac{1}{\sin \theta} + \frac{1}{\tan \theta} \right) \right] \quad (7.56)$ $A_i = \frac{1}{2}xy \sin \theta - \frac{1}{2} \left(x - 2\bar{a} \left[\frac{1}{\tan \theta} + \frac{1}{\sin \theta} \right] \right) \left(y \sin \theta - \bar{a} \left[1 + \frac{1}{\cos \theta} \right] \right) \quad (7.57)$ $A_c = A_o = \frac{1}{2}xy \sin \theta \quad (7.58)$ $\phi = \frac{A_o - A_i}{A_o} \quad (7.59)$

Table 7.1 provides the geometric parameters that are used to determine the average melting thickness from the liquid fraction and vice versa, for the rectangular and triangular fin cavities.

Comparison with CFD - Rectangular fin cavity

Figure 7.20 and Figure 7.21 compare the analytical and CFD results for varying aspect ratio at constant free flow area $A_c = 25 \text{ mm}^2$. It can be seen from the first figure that the model is capable of accurately predicting the amount of melted PCM in the cavity over time, and gives a reasonable prediction of the melt front over time, although the simplified model tends to predict the shape of the melt front more accurately for square cavities than for rectangular cavities.

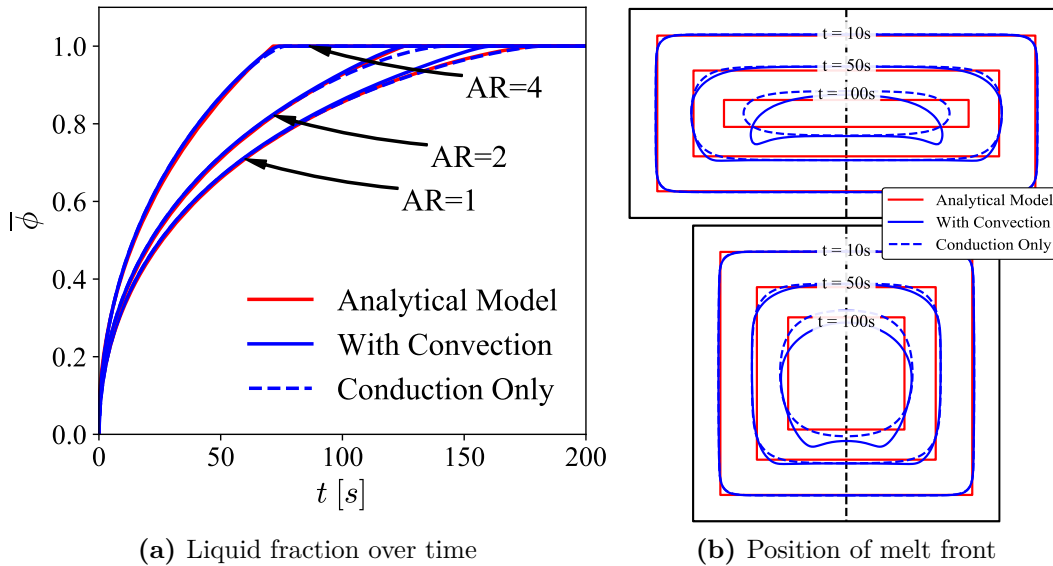


Figure 7.20: Comparison of analytical and CFD results for the rectangular fin cavity for varying aspect ratio at constant wall temperature $T_w = 301.15K$ and constant free flow area $A_c = 25 \text{ mm}^2$: (a) Liquid fraction over time; (b) position of melt front.

Figure 7.21 compares the results of the analytical model with the CFD over a greater range of parameters based on the average liquid fraction at the end of the melting cycle for varying AR and T_w . This shows that over the studied range, the model is capable of predicting a similar amount of melted PCM where the results from the simplified model are within $\pm 5\%$ of the CFD values.

Following this, the analytical prediction of the change in heat exchange surface area A_{PCM} at constant plate spacing is captured in Figure 7.22. This shows good agreement where the CFD results are within 7%, and where the analytical model

predicts a higher melt fraction when compared to the CFD.

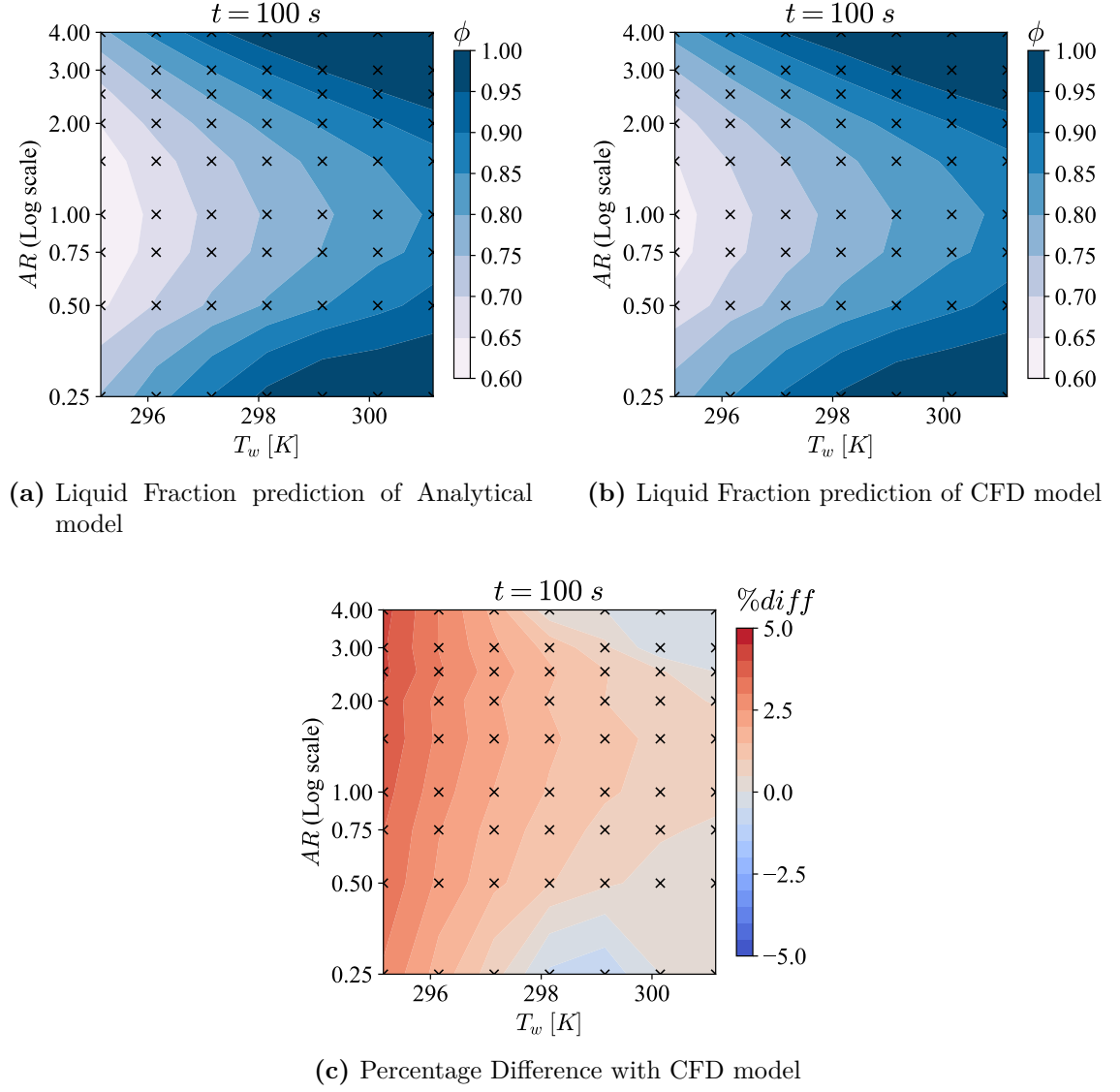


Figure 7.21: Comparison of analytical and CFD results for the rectangular fin cavity for varying aspect ratio for varying temperature at the end of the melting cycle ($t = 100$ s) at constant free flow area $A_c = 25 \text{ mm}^2$: (a) Liquid Fraction prediction of analytical model; (b) Liquid Fraction prediction of CFD model; (c) Percentage Difference between CFD and analytical results.

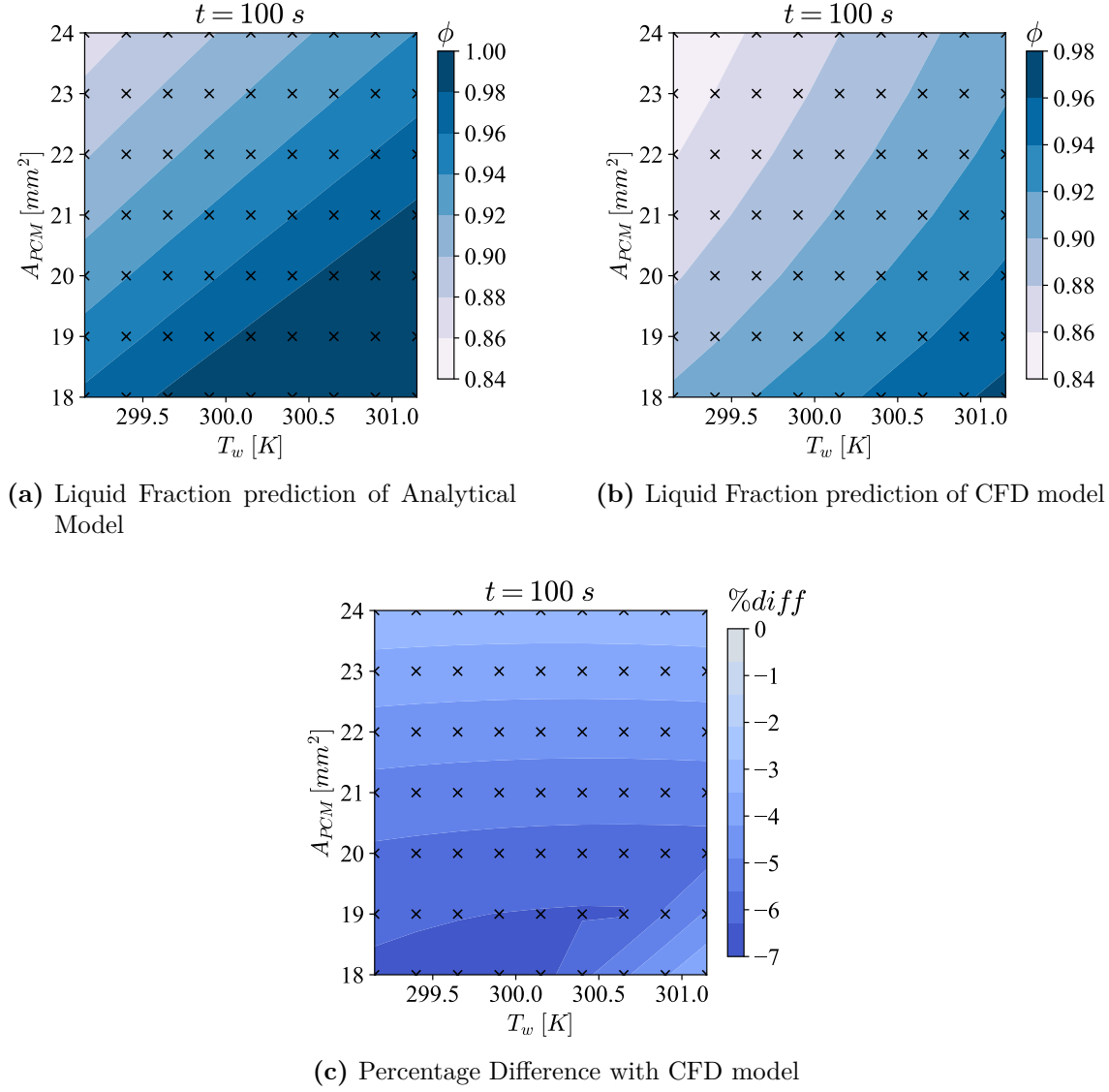


Figure 7.22: Comparison of analytical and CFD results for the rectangular fin cavity for varying surface area for varying temperature at the end of the melting cycle ($t = 100$ s): (a) Liquid Fraction prediction of analytical model; (b) Liquid Fraction prediction of CFD model; (c) Percentage Difference between CFD and analytical results.

Comparison with CFD - Triangular fin cavity

Figure 7.23 shows the average liquid fraction prediction for a triangular fin geometry for two different aspect ratios at constant free flow area $A_c = 25 \text{ mm}^2$. Good agreement is observed between the CFD and analytical model for the equilateral triangle at $AR = 1$. However, a greater discrepancy is observed at the higher aspect ratio, where there is good agreement in the initial 20 seconds. Following this, the analytical model under-predicts the melt fraction when compared to the CFD results.

The impact of aspect ratio on the prediction of the melting of PCM in triangular cavities by the analytical model is further highlighted in Figure 7.24, where the position of the melt front prediction has been captured over time for a wider range of aspect ratios. This shows that as the aspect ratio increases the melt front prediction of the analytical model deviates more and more from the CFD results. This change in agreement could be linked to the relationship that is used for the shape factor not being sufficient to represent the thermal resistance of the PCM.

7.4.1.4 Summary

In this section, a new model for predicting the melting within a finned heat exchanger cavity was developed, following an adaptation of the surface effectiveness method commonly used in conventional $\epsilon - NTU$ models. The model was implemented in MATLAB and the results for the liquid fraction and the location of the melt front were extracted. When compared to CFD results, the semi-analytical model showed good agreement with CFD results for the rectangular geometry over a wide range of dimensions and operating temperatures. Possible differences in results could be explained by inaccuracies arising from the shape factor relationship and the simplifications made to the thermal response of the phase change process, i.e. phase change at a constant temperature and quasi-steady phase change. A greater discrepancy was observed for the triangular fins, where only the results for $AR = 1$, i.e. for equilateral triangular fins, was capable of predicting the melting accurately. To be able to use the model on triangular fins, further research in different formulations of the shape factor is required.

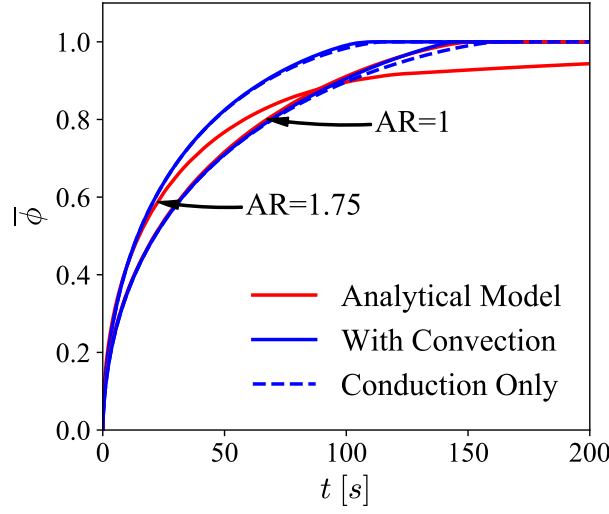


Figure 7.23: Comparison of analytical and CFD results for the triangular fin cavity for aspect ratio $AR = 1$ and $AR = 1.75$ at $A_c = 25 \text{ mm}^2$ and $T_w = 301.15\text{K}$: average liquid fraction over time.

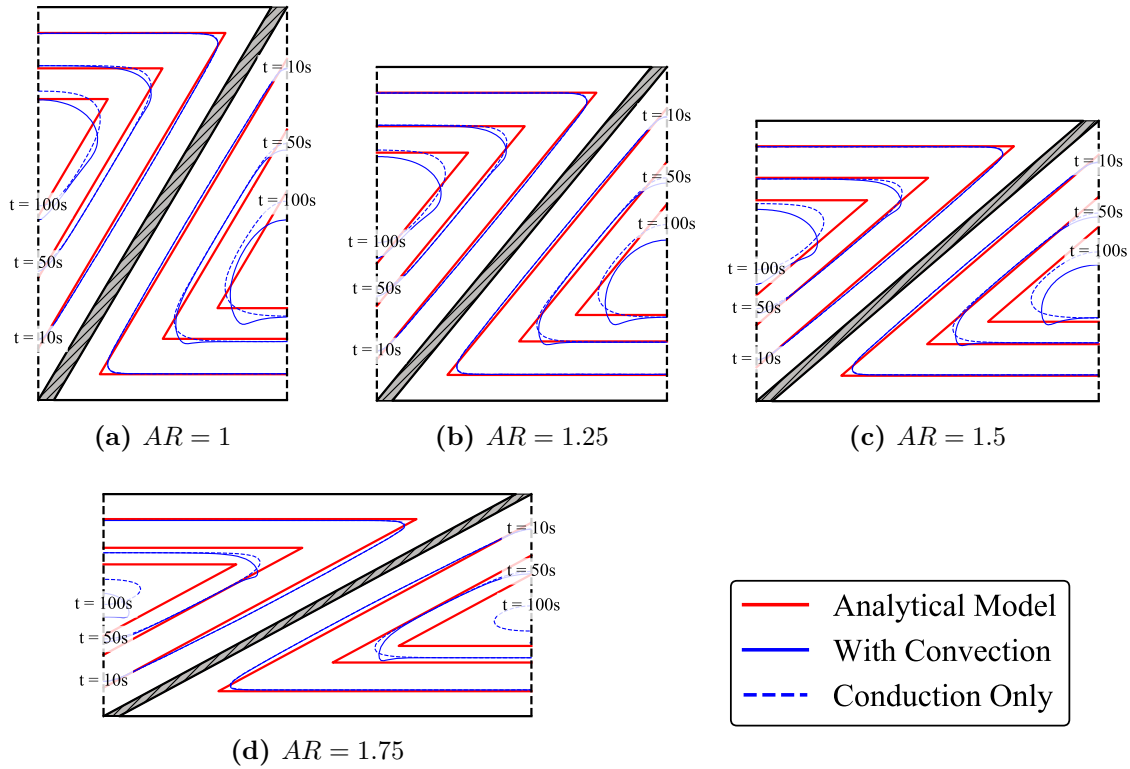


Figure 7.24: Comparison of analytical and CFD results for the triangular fin cavity for varying aspect ratio AR at $A_c = 25 \text{ mm}^2$ and $T_w = 301.15\text{K}$: Position of the melt front over time

7.4.2. Addition of fins to the full Effectiveness-NTU model

7.4.2.1 Implementation in MATLAB

The cavity model was implemented in the $\epsilon - NTU$ model by following the algorithm shown in Figure 7.25. The addition of fins has an impact on the $\epsilon - NTU$ model in the calculation of two parameters: the overall heat transfer coefficient and the calculation of the averaged melt thickness. The averaged melt thickness, which is used to determine the surface efficiency of the finned PCM layer, is calculated from the liquid fraction based on the fin geometry in a similar fashion to what was presented previously.

For the calculation of the overall heat transfer coefficient, the shape factor relationship presented in the previous section was adapted such that an arbitrary amount of elements can be used for the heat exchanger, rather than having to use one element per cavity, as was previously required. For this, the shape factor is scaled based on the element length dz and the PCM fin cavity length z and PCM fin thickness δ_{PCM} :

$$S = \frac{2\pi l_z}{\ln [1 + (2\pi \bar{a}/P_i)]} \frac{dz}{z + \delta_{PCM}}. \quad (7.60)$$

This approach also allows for more flexibility in the choice of heat exchanger length as incomplete fin cavities can be used. Furthermore the thermal resistance of the PCM was adapted, such that it takes the number of PCM layers (n_{PCM}) into account:

$$R_{PCM} = \frac{1}{n_{PCM} S k_{PCM} \eta_{0,PCM}}. \quad (7.61)$$

This is necessary, as the fluid area and wall area are already scaled by the number of PCM layers. This can then be substituted in the overall heat transfer coefficient relationship giving:

$$U_f = \left(\frac{1}{\eta_{0,f} h_f} + \frac{A_f t_p}{A_w k_w} + A_f R_{PCM} \right)^{-1} \quad (7.62)$$

7.4.2.2 Comparison with CFD model of PCM compact heat exchanger

To validate the finned geometry model, the PCM compact heat exchanger geometry shown in Figure 7.26 (identical to previous CFD simulations) was used with a length

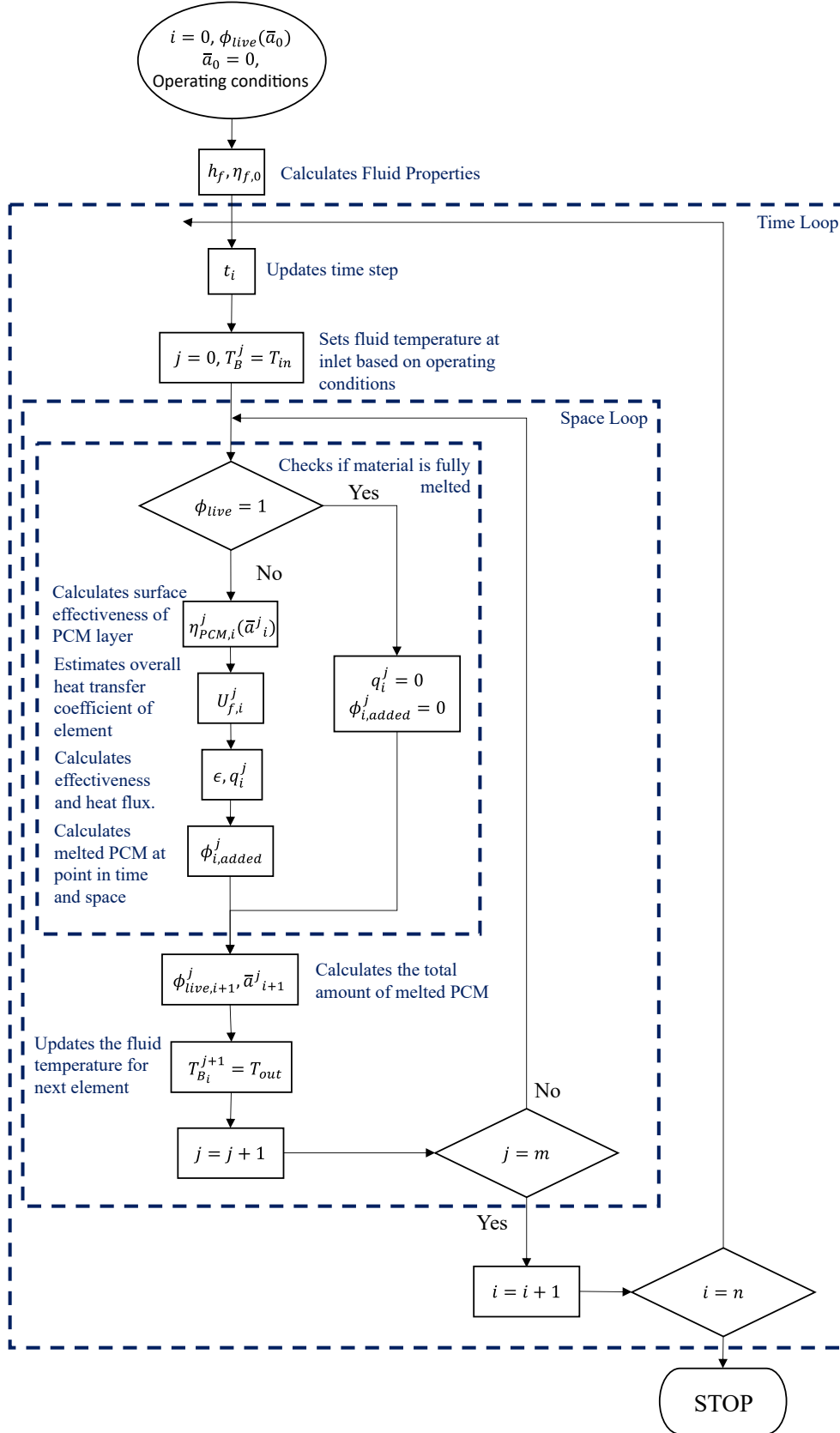


Figure 7.25: Flowchart of $\epsilon - NTU$ model incorporating fins in fluid and PCM layer.

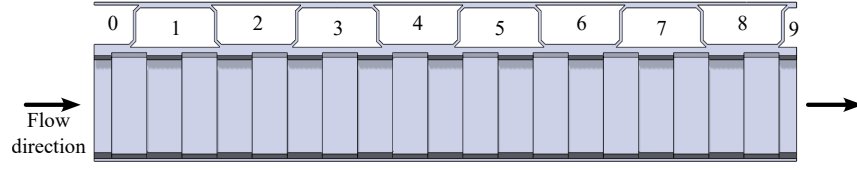


Figure 7.26: Geometry of finned PCM compact heat exchanger used for validating the $\epsilon - NTU$ model.

of 63.5 mm corresponding to 10 periodic flow lengths of interrupted fins in the fluid layer. For the $\epsilon - NTU$ model, it was assumed that the fin cavities were rectangular, where $z = 7.1 \text{ mm}$, $y = 3.5 \text{ mm}$, following properties from Table 7.1.

Figure 7.27 compares the average liquid fraction and temperature drop predicted by the two models at $Re = 50$ and $Re = 200$. Good agreement is observed between the two models at higher Reynolds number, while at lower Reynolds number, a greater difference is observed, mainly for the temperature drop. Figure 7.28 indicates the drop in the liquid fraction over the length of the heat exchanger at these Reynolds numbers. It can be observed that the model predicts the liquid fraction well close to the inlet for each value of Re , indicating that the effect of Reynolds number is well captured by the model. A greater difference can however be observed at the outlet, mostly apparent at lower Reynolds number, which is responsible for the divergence in results in the average liquid fraction observed in Figure 7.27.a at $Re = 50$.

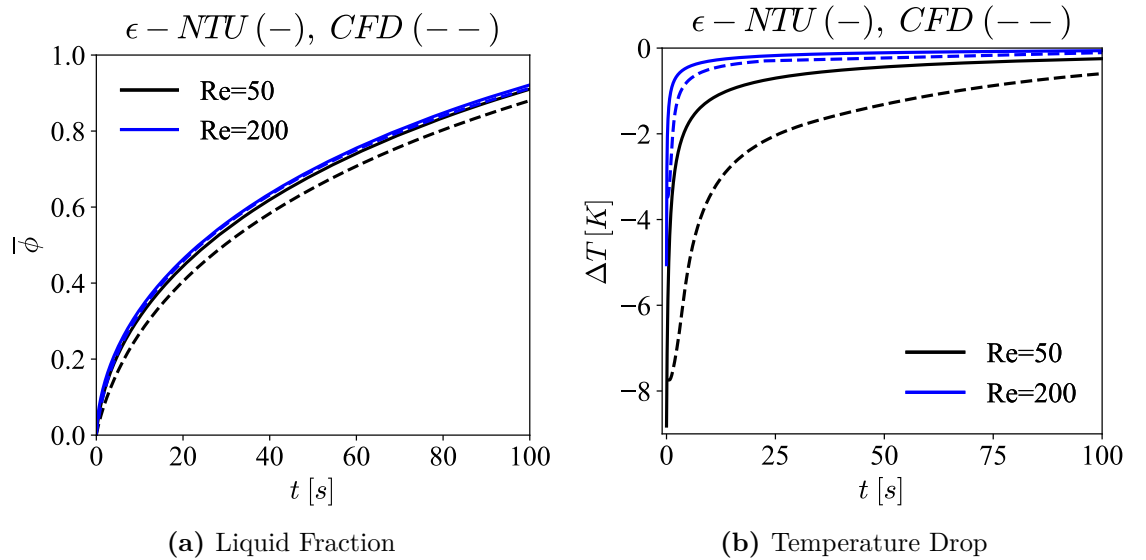


Figure 7.27: Comparison of the $\epsilon - NTU$ model with the CFD results for a compact heat exchanger with fins in the PCM layer for $Re = 50$ and $Re = 200$: (a) Liquid Fraction $\bar{\phi}$, (b) Temperature Drop ΔT .

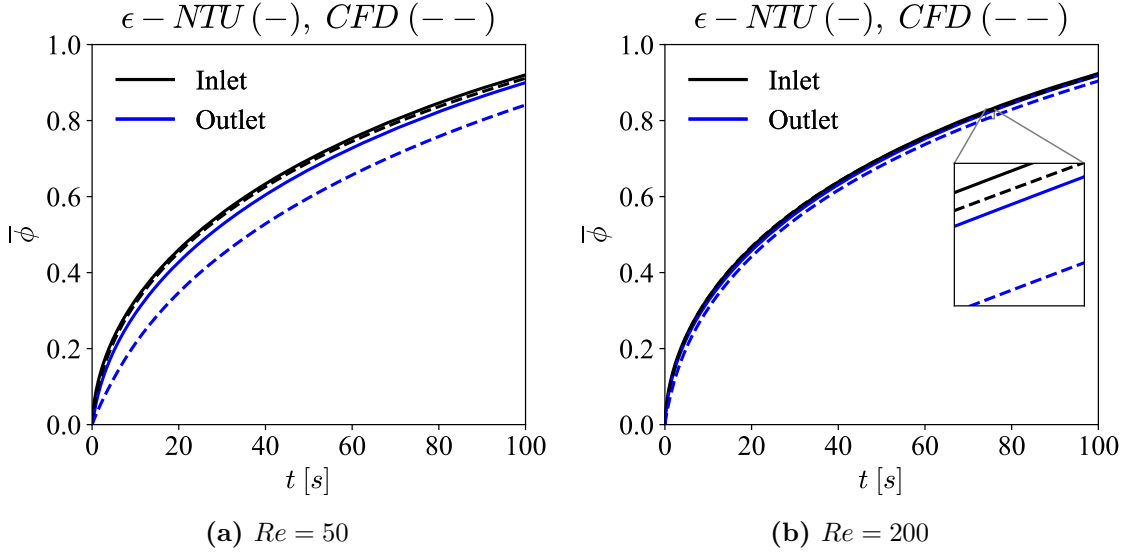


Figure 7.28: Comparison of the drop in the liquid fraction in the cavities over the length of the heat exchanger for the $\epsilon - NTU$ model (—) and the CFD results (---).

As the energy conservation model for the simple cavity model showed good agreement with CFD results, the divergence in results for the full heat exchanger is most likely attributed to the prediction of the temperature drop by the $\epsilon - NTU$ model. The fluid bulk temperature is higher for the heat exchanger towards the outlet in the $\epsilon - NTU$ model, such that the melting model predicts a higher amount of melted PCM. In the steady state formulation, the residence time of the fluid as well as the transient thermal response of the heat exchanger cannot be taken into account, which explains the difference in results. In previous iterations, where no fins were incorporated in the PCM layer, this only had a small impact on the melting of the PCM, due to the larger difference in the thermal resistances between the fluid and PCM layer.

The steady-state $\epsilon - NTU$ model for finned PCM-compact heat exchanger is therefore only recommended at larger Reynolds number, where the transient response of the heat exchanger is limited. For the remaining operating range of the heat exchanger, improving the $\epsilon - NTU$ relationship is required, so that it could capture the transient response of the heat exchanger.

7.5. Using the effectiveness-NTU model as a design tool

7.5.1. Motivation

The $\epsilon - NTU$ model that was developed in this chapter can significantly reduce the computational cost for determining appropriate dimensions of heat exchangers, when compared to CFD. In this section, two different test cases will be presented on how the model could be used for designing PCM compact heat exchanger geometries based on the aerospace loading case provided by BAE Systems, where 500 kJ of heat have to be stored in the RT18 HC PCM in 100 s, with a temperature difference between the PCM and the fluid inlet temperature of 10 K. For this, the 1/8-13.95 fin will be used on the fluid side, with water as the working fluid. As the model predominantly showed good agreement at high Reynolds number, a constant Reynolds number of $Re = 200$ will be kept throughout the different test cases.

For the first case, appropriate dimensions of the heat exchanger at constant fin spacing will be determined for various Reynolds numbers, while for the second case, optimal fin spacing will be determined to offset the decrease in melting over the length of the heat exchanger.

7.5.2. Determining the dimensions of PCM compact heat exchanger geometries

Finding the right dimensions for a heat exchanger is a crucial point of designing a heat exchanger for a specific test case. This section provides an overview on the different geometric parameters that can be changed within the model, that will have an impact on the size of the heat exchanger. For this, the $\epsilon - NTU$ model that has been implemented in MATLAB in Figure 7.25 is used, where the cumulative energy stored in the PCM at 100 s is tracked over the length of the heat exchanger to determine the optimal length of the heat exchanger.

The different test cases based on the parameters outlined in Table 7.2 were run and the amount of energy stored at the end of the melting cycle over the heat exchanger length lz was extracted from the $\epsilon - NTU$ model. Following this, the ideal length of the heat exchanger was interpolated at the point where an energy threshold of 500 kJ was reached was extracted.

Table 7.2: Simulation condition for example case 1 to illustrate how to find PCM heat exchanger dimensions

Simulation Properties	
Frontal Area per PCM cavity $A_{PCM,cav}$ [mm^2]	25
Aspect Ratio of PCM cavity AR	0.25 : 4
Number of PCM layers (n_{PCM})	10 : 2 : 30
Heat exchanger width (l_x) [mm]	100 : 25 : 300

Figure 7.29 shows results of the parametric study at $AR = 1$, indicating the required length and the corresponding volume of the heat exchanger for the various values of l_x and l_y . This indicates that to minimize the volume of the heat exchanger, it is necessary to decrease l_z , as this parameter will be influenced by the temperature drop of the fluid, where further along the length of the heat exchanger, less of the PCM will be melted.

Following this, the aspect ratio of the PCM cavities was varied and the effect on the heat exchanger volume can be observed in Figure 7.30. This indicates that as AR increases, the volume required for the heat exchanger also increases. This is partially linked to the fact that as AR increases, l_y decreases over the range of PCM layers that was studied and this leads to the need of a longer heat exchanger.

By looking at Figure 7.31, where the required length has been plotted for a constant

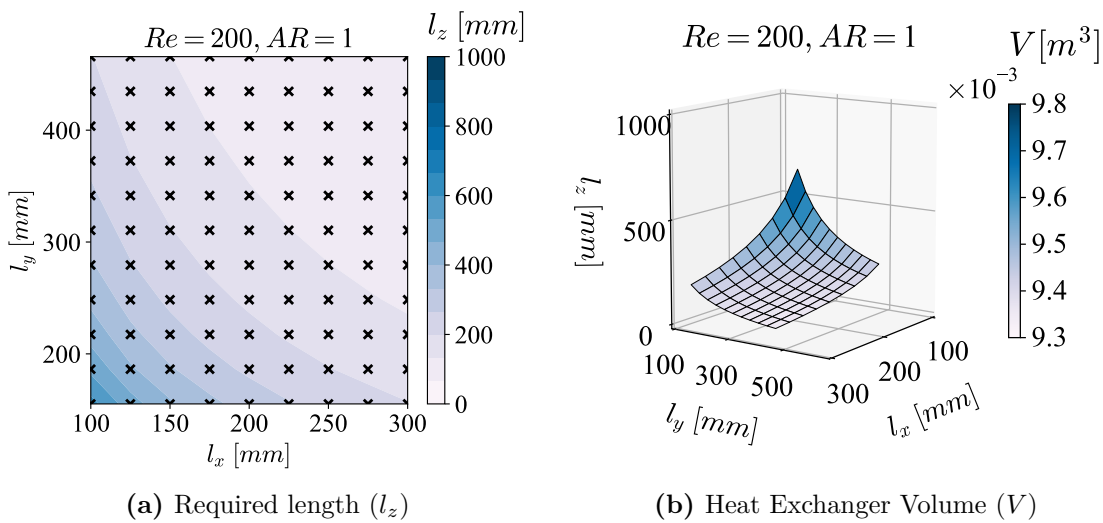


Figure 7.29: Effect of varying the dimensions of the heat exchanger on (a) the required length (l_z); and (b) Heat Exchanger Volume (V); to store 500 kJ of heat in 100 s in the PCM based on an operating Reynolds number of $Re=200$.

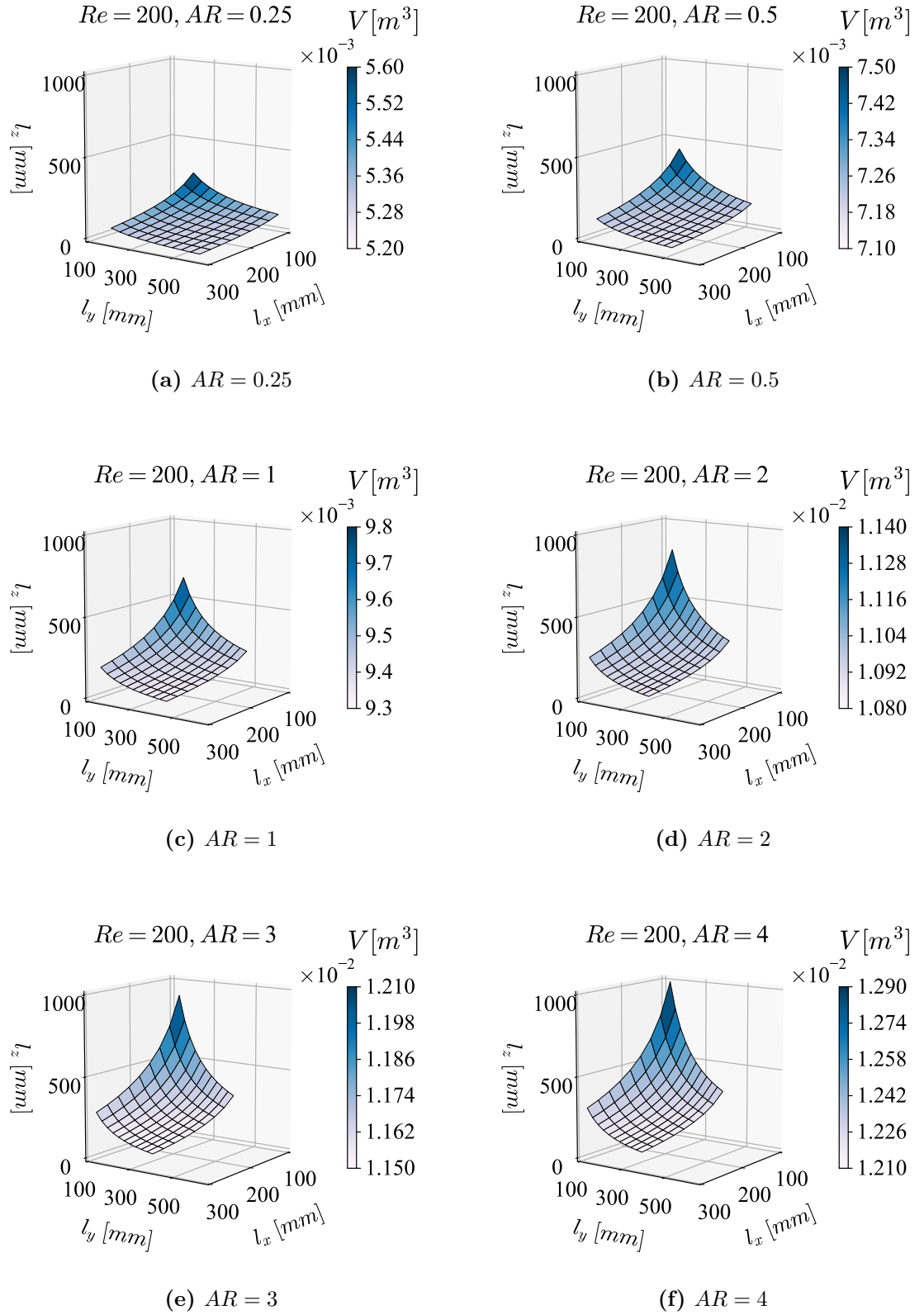


Figure 7.30: Effect of varying the dimensions of the heat exchanger on the Heat Exchanger Volume (V) for varying Aspect Ratio (AR).

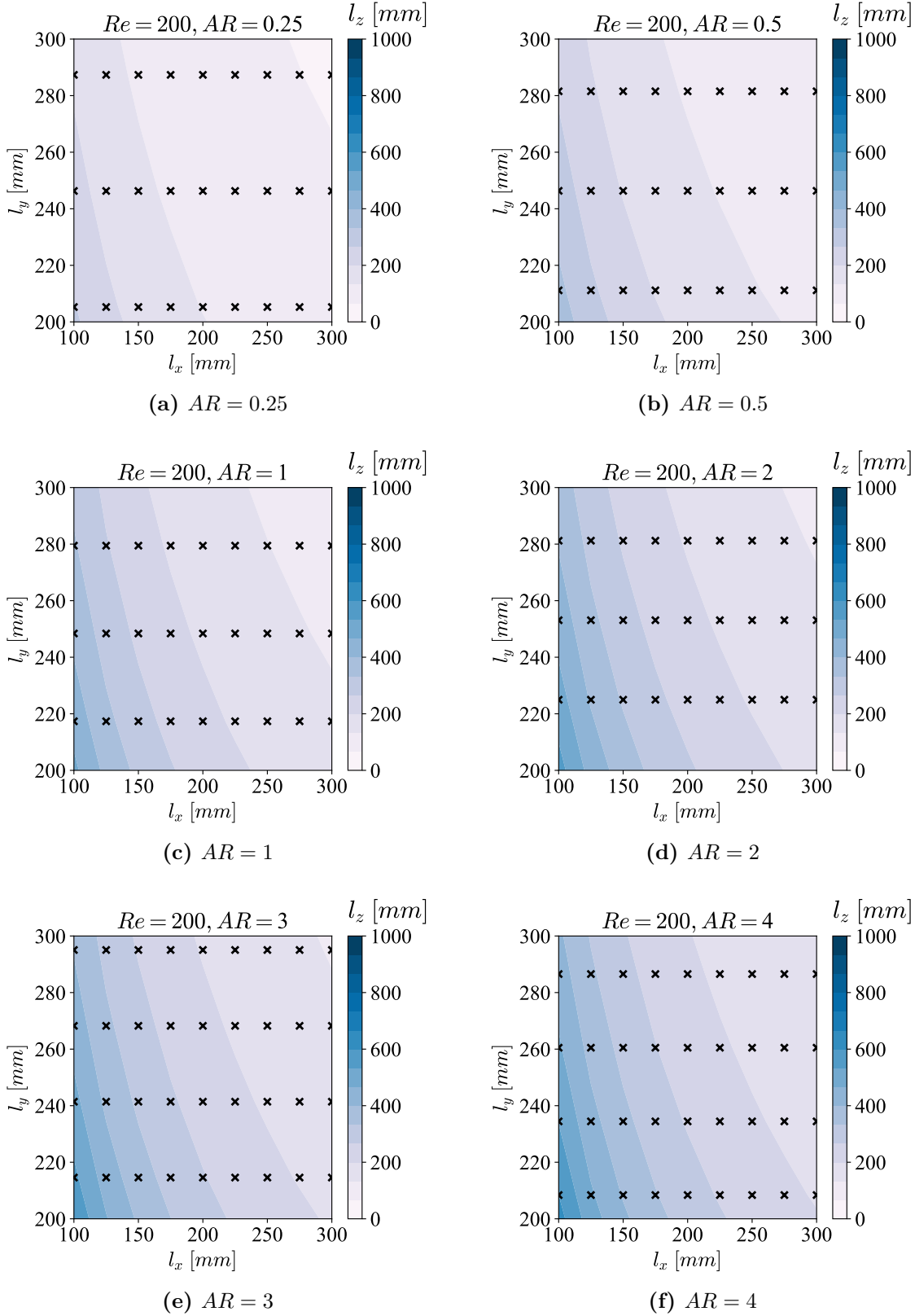


Figure 7.31: Effect of varying the dimensions of the heat exchanger on the length (l_z) of the heat exchanger for varying Aspect Ratio (AR).

range of heat exchanger height values, it is however possible to observe that a larger heat exchanger length is required for the same height and depth of the heat exchanger as the aspect ratio increases. This is linked to the fact that the heat exchanger has a greater proportion of PCM when compared to water, suggesting that to minimize the volume, it would be beneficial to reduce the size of the water channels. It is worthy to remember that as AR decreases, the volume of metal in the PCM layer also increases, which has a greater impact on the mass, than the PCM.

7.5.3. Determining optimal fin spacing for uniform melting of PCM over length of heat exchanger

To be able to use the available space more efficiently, the size of the PCM cavities could be adapted, based on the amount of energy that can be transferred to the PCM in that cavity. For this, the length of each cavity could be reduced over the length of the heat exchanger, where the volume in the cavity towards the outlet of the heat exchanger would be smaller than towards the inlet.

Due to the small computational cost of the PCM- ϵ – NTU model, this could be used as a basis for an algorithm, where the optimal size of the cavity is determined for each cavity individually. In the following section, this algorithm will first be described, before results will be presented.

7.5.3.1 Adapting the model to find optimal melting

Figure 7.32 indicates the general structure of the algorithm to determine optimal dimensions of the heat exchanger. For this, an optimal PCM cavity is added to the length of the heat exchanger at each iteration and the amount of heat that can be stored within the melting cycle is calculated. This is then compared to the heat threshold determined by the user and the algorithm is stopped once the heat threshold is reached.

To calculate the optimal dimensions of the cavity, a modified version of the bisection method is used as shown in Figure 7.33 that determines the cavity spacing (z_{opt}) for each cavity over the length of the heat exchanger. For this an initial interval of z dimensions is provided by the user and the liquid fraction at the end of the melting cycle is calculated for each value of the initial interval $[a, b]$. The interval of search is then updated until the calculated liquid fraction approaches ϕ_{opt} by calculating the mean value of the interval c and comparing the final liquid fraction for c to the

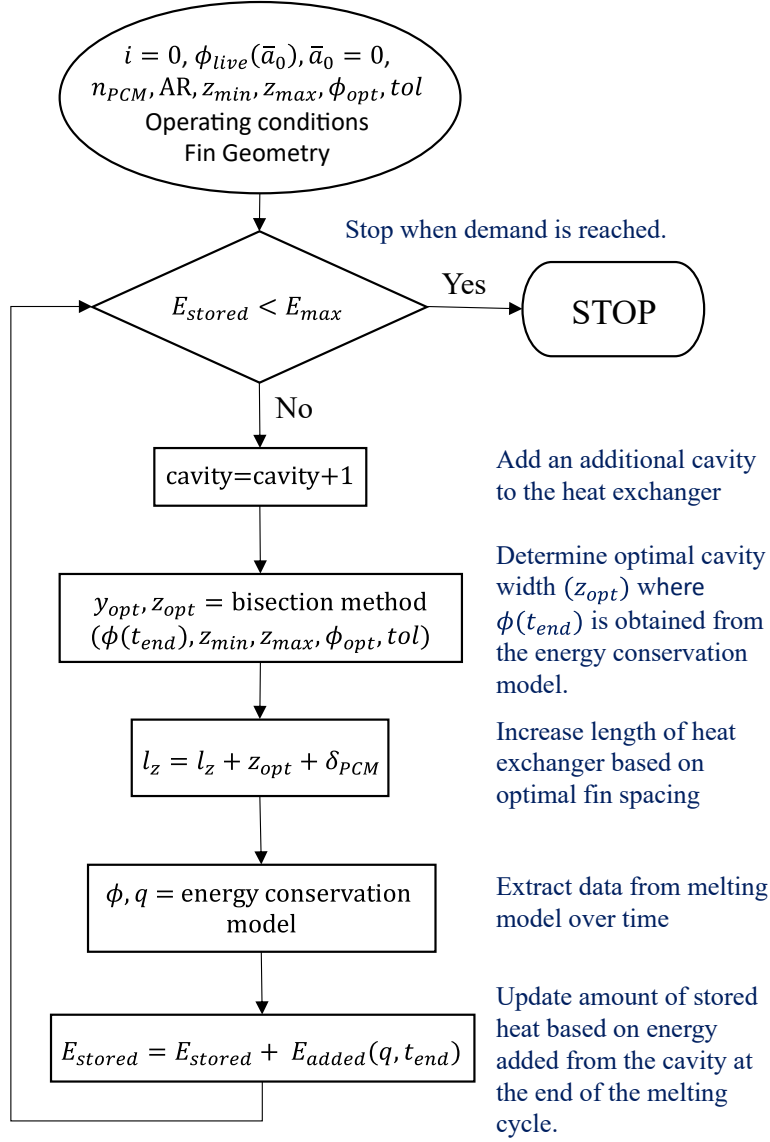


Figure 7.32: Flowchart of algorithm for finding optimal dimensions of heat exchanger based on an optimal melting criteria ϕ_{opt} . The optimal dimensions are determined by using a bisection algorithm as shown further down in Figure 7.33. To calculate the liquid fraction and heat flux, the energy conservation $\epsilon - NTU$ model described in Figure 7.25 is used.

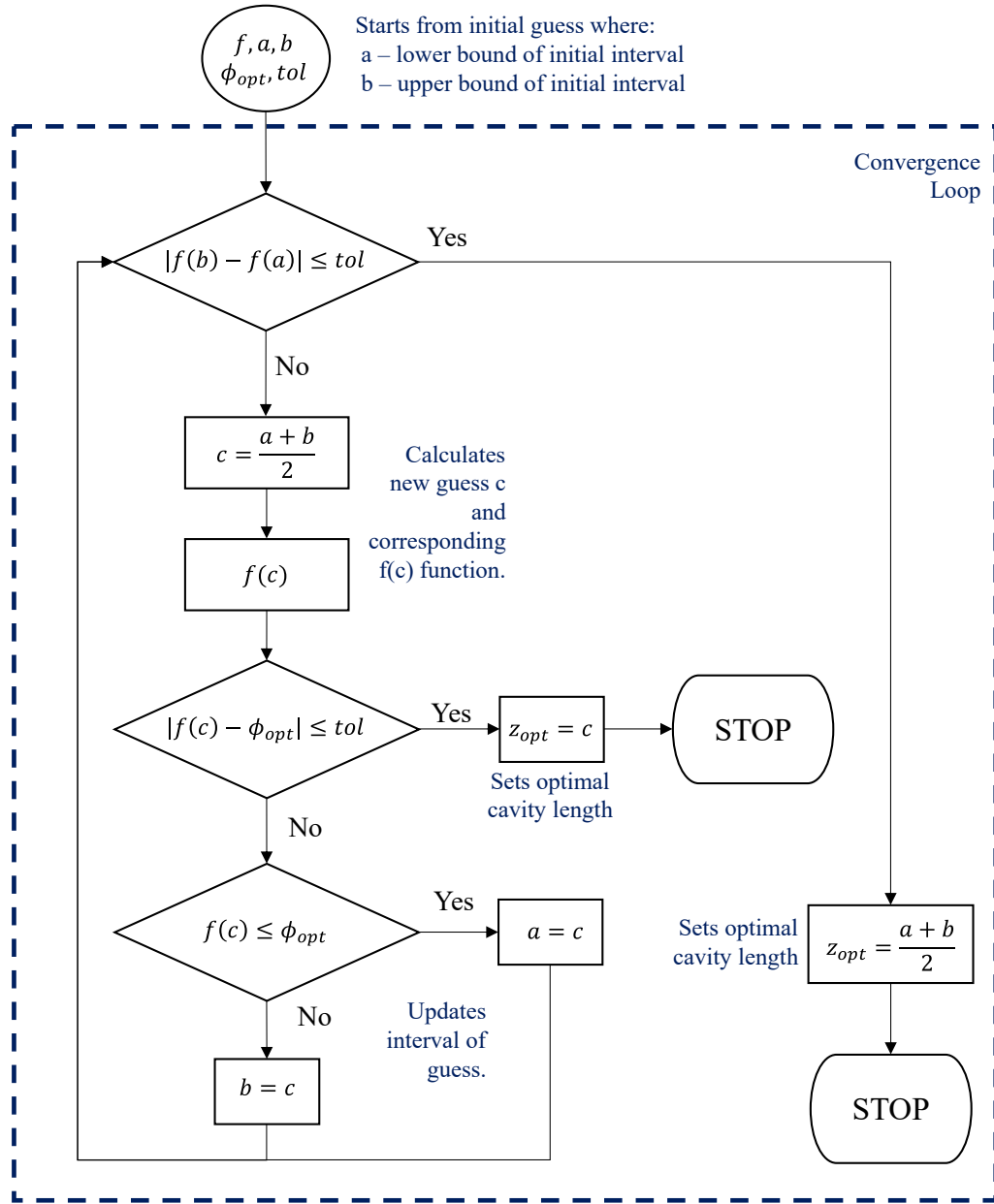


Figure 7.33: Algorithm used to find optimal length z_{opt} of fin cavity such that at the end of the melt cycle, the full cavity is melted.

optimum melt fraction.

For the first cavity, the optimal cavity height y_{opt} is calculated from the initial aspect ratio AR and z_{opt} , making it possible to determine the width and height of the heat exchanger. The liquid fraction and heat flux over time can also be extracted from the energy conservation model, making it possible to track the heat transfer performance of the heat exchanger over time.

7.5.3.2 Example Case

To illustrate the algorithm, an example case is presented, where the optimal fin spacing is determined for different values of the initial aspect ratio. For this, the additional simulation conditions set out in Table 7.3 are prescribed.

Table 7.3: Additional simulation condition for example case 2 to illustrate the optimum variable fin spacing algorithm

Simulation Properties	
Number of PCM layers (n_{PCM})	15
Heat exchanger width (l_x)	l_y
Optimal melt fraction ϕ_{opt}	0.95
Tolerance (tol)	0.001
Heat to be stored in PCM (E_{max}) [kJ]	500
Initial Interval for calculating z_{opt} [mm]	[0.5; 20]

Figure 7.34 shows the change in the macroproperties for different aspect ratio. At $t = 100$ s the liquid fraction reaches the optimal melt fraction of 0.95 and a similar temperature drop of the fluid is observed over time for all cases. This shows that the model manages to control the dimensions of the heat exchanger, such that the input parameters defined by the user are met by the algorithm. Interestingly, the PCM melts more rapidly at the beginning of the melt cycle at lower AR , but this does not have a significant impact on the temperature difference of the fluid layer.

Figure 7.35 shows the effect of the aspect ratio (AR) on the dimensions of the heat

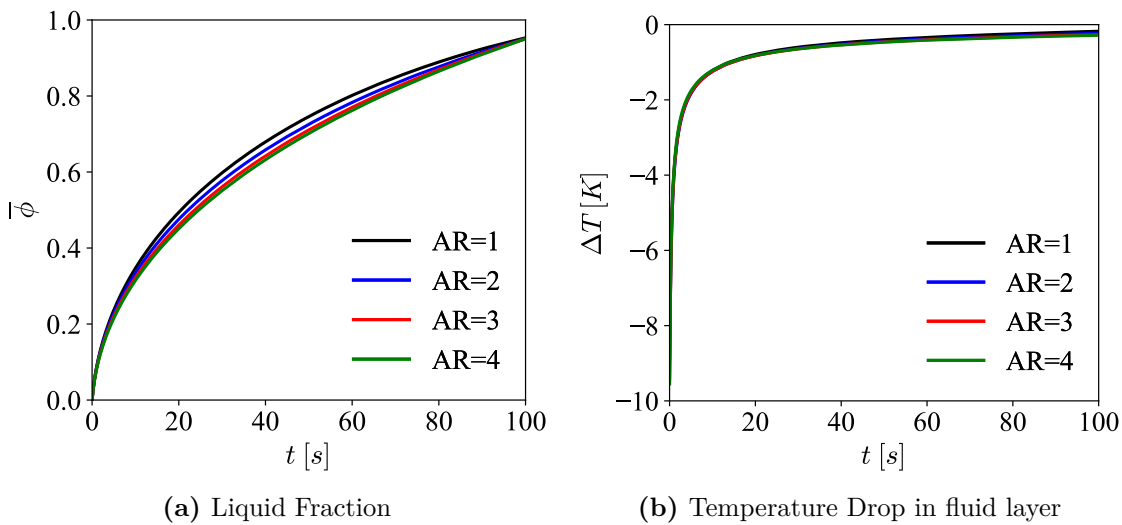
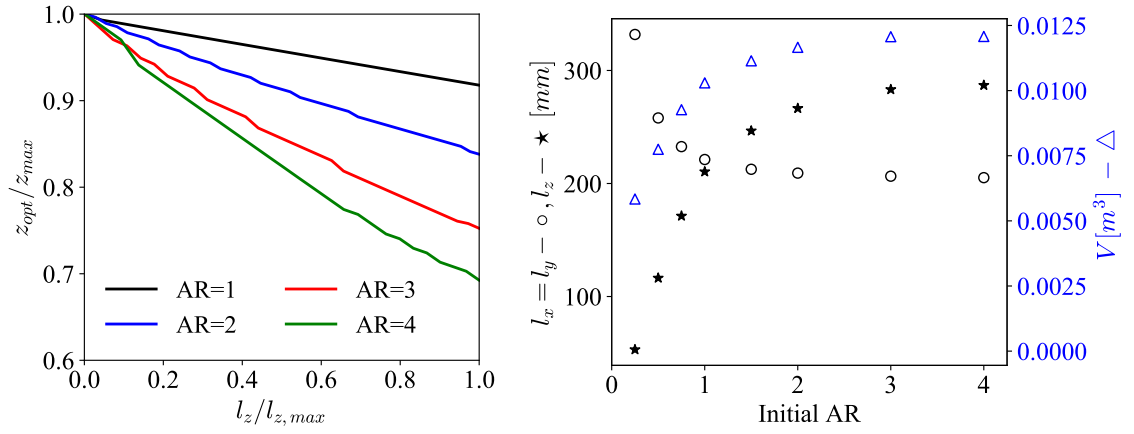


Figure 7.34: Comparison of different initial aspect ratio on average properties over time: (a) Liquid Fraction, (b) Temperature Drop in fluid layer.



(a) Change in the optimal fin spacing over length (b) Variation in the dimensions of the heat exchanger

Figure 7.35: Effect of Aspect ratio on the dimensions of the heat exchanger with varying fin spacing: (a) Change of optimal fin spacing over the length of the heat exchanger; (b) Variation in the dimensions of the heat exchanger.

exchanger. As AR increases, the change in the cavity width over the length of the heat exchanger becomes more significant. This translates into a sharper decrease in the fin spacing at higher AR . Similarly to what was observed in the previous example case, increasing AR leads to a larger heat exchanger volume as the fluid layer occupies a larger proportion of the heat exchanger volume.

A constant fin spacing heat exchanger geometry was created to compare against the results of the variable fin spacing geometry. For this, the cavity width and height were chosen such that they matched the first cavity of the variable heat exchanger and the length, height and width of the heat exchanger from the variable case was used. The case where the largest deviation in the fin spacing ($AR = 4$) was used to compare the melting for the two different geometries in Figure 7.36 on the following page. The variable geometry melts a small additional amount of PCM than the constant fin spacing, while the temperature drop is relatively consistent. This indicates that while the PCM in the cavities melted more, this did not lead to a more significant amount of stored heat. This is linked to the greater fin density towards the outlet as illustrated by Figure 7.37, such that a smaller volume of PCM is available in the heat exchanger. Additionally, as the simulation was run at $Re = 200$, the melting in the heat exchanger is relatively uniform, such that only a small improvement in the liquid fraction can be observed for the chosen fluid fin geometry.

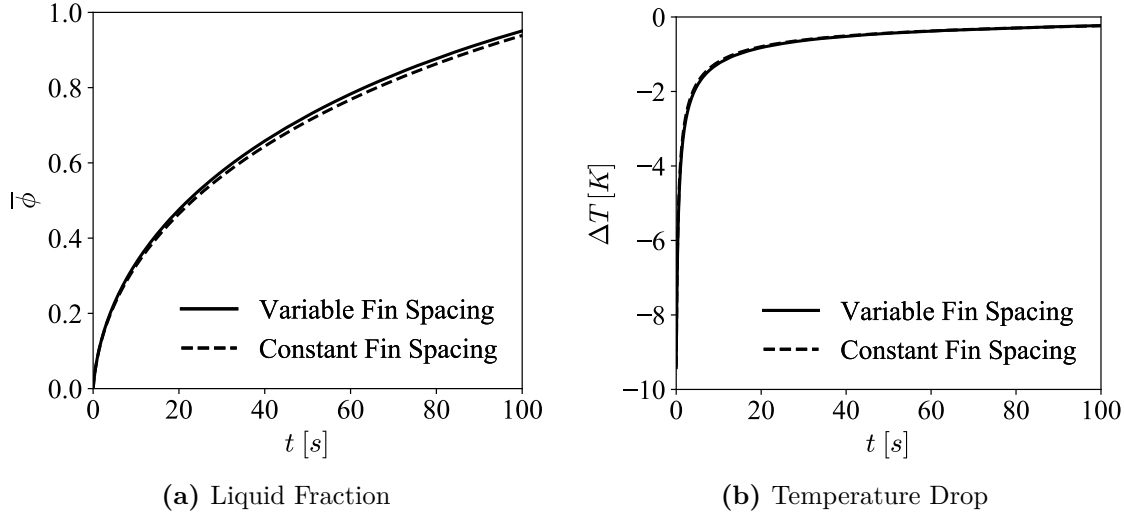


Figure 7.36: Comparison of the effect of using variable and constant fin spacing on the average liquid fraction over time for similar heat exchanger dimensions (l_x, l_y, l_z) at $AR = 4$.

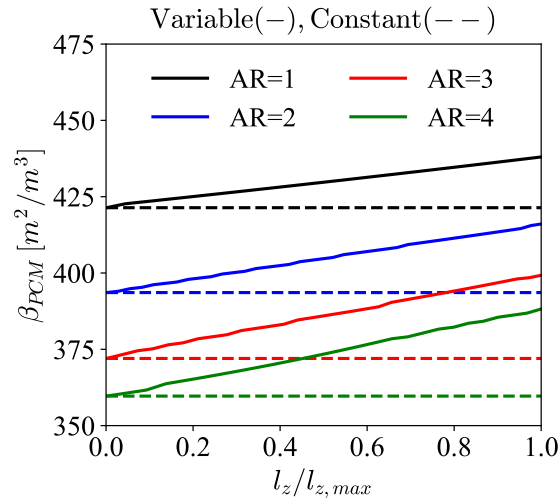


Figure 7.37: Variation of the compactness (β) over the normalized length of the heat exchanger ($l_z/l_{z,max}$) for both constant and variable fin spacing.

7.6. Summary

In this chapter, a model for designing PCM compact heat exchanger was developed and compared with CFD results from previous chapters. This started off with an ϵ -NTU for a heat exchanger with unfinned PCM layers, where the melting of the PCM was tracked by using a linear approximation of a 1D Stefan problem. The model showed good agreement with CFD results from Chapter 5. As the linearised Stefan model cannot easily be adapted to incorporate fins into the PCM layer, this was subsequently replaced by an energy conservation model, which showed equivalent results to the linear Stefan model.

Following this, a model was created to capture the melting in finned PCM cavities. This was achieved by adapting the surface effectiveness for fluids to PCM layers and by applying a shape factor to map the 2D heat transfer to the 1D heat transfer equations used by the $\epsilon - NTU$ model. When compared to CFD results for a single cavity, good agreement within a maximum difference of 7% was observed for rectangular cavities over a range of operating temperatures and cavity aspect ratios. However, for triangular fins, only equilateral fin cavities were found to appropriately predict the melting rate, where higher aspect ratios led to a significant underprediction of the melting rate.

The finned cavity model was incorporated in the $\epsilon - NTU$ for the whole heat exchanger. This found that the quasi steady state formulation of the model was capable of predicting the amount of melted PCM accurately only in cases where the Reynolds number was sufficiently large, i.e. in cases where the transient response of the fluid channel was negligible.

Finally, two example cases were presented to illustrate how the $\epsilon - NTU$ model could be used to design PCM compact heat exchanger based on the operating conditions obtained from the industrial partner at $Re = 200$, which had shown good agreement with the CFD results.

CHAPTER 8

Conclusion

Contents

8.1	Summary of results	227
8.2	Implication of results	231
8.3	Recommendations for future work	231

8.1. Summary of results

The purpose of this research was to develop novel heat exchangers based on latent heat storage to reduce the considerable load that will be placed on aircraft thermal management systems in the near future. Thermal management systems are characterised by highly transient head loads that vary over the course of the flight and leads to oversized heat exchanger systems. PCM could be a potential solution to reducing the size of these heat exchangers by managing the temperature fluctuations within the system. As PCM heat exchangers are mainly used within buildings, significant changes to the geometry of current LHTES have to be made to improve the melting rate and decrease the mass of the systems. This project focused on improving the understanding of the melting of PCM within plate-fin heat exchangers that are commonly used as part of current thermal management systems. This was achieved in this thesis, by numerically studying the effect of extended surfaces within plate-fin heat exchangers subjected to an aircraft loading profile over a range of mass flow rates. Based on the findings from the simulations, a semi-analytical model was developed. The main outcomes of the research are summarized for each results chapter and are related to the initial project aims and objectives that were

outlined in Section 1.3.2:

Chapter 4: Modelling of a PCM Shell and Tube Heat Exchanger

- This chapter focussed on developing a strong understanding of the melting behaviour of PCM to create a modelling framework (Aim 1, Obj. 1-2) that could be used throughout the remaining body of work. A study from the literature was recreated, where a commercial paraffin-based PCM was melted within a shell and tube heat exchangers by using the enthalpy-porosity model.
- Initially, a two dimensional model was created, where the inlet temperature was applied at the tube outer wall. The model was used to select the modelling inputs by testing the effect of different properties. This included the choice of modelling software, material property definition and choice of numerical schemes.
- When compared to the experimental results from the original study, the two dimensional model was found to be insufficient to fully capture the energy absorption from the PCM, as a significant over-prediction of the liquid fraction and average temperature over time was observed. The difference in results was attributed to the temperature drop within the fluid channel, where the temperature at the tube wall towards the outlet is lower than at the inlet.
- By incorporating the temperature drop into the model by using a conjugate heat transfer approach where the fluid channel was fully modelled, a better agreement with the original study was observed. The change in the temperature drop as the PCM melts was found to have a significant impact on the amount of melted PCM over the length of the heat exchanger, where a larger difference in the average liquid fraction was observed at the beginning of the simulation between the inlet and outlet. This speaks in favour of coupling the fluid domain and the PCM domain in future work.
- Additionally, a detailed three dimensional model of the shell was created for comparison with the two dimensional model. A similar sensible and latent heat addition was observed for the two models, although a disagreement was observed for the shape of the melting front. The three dimensional model led to the emergence of plumes in both the axial and lateral direction, while the two dimensional model neglected the velocity in the axial direction. This explained the difference in the shape of the melting front.

Chapter 5: Improving the melting of PCM with finned surfaces in fluid layer

- In this chapter the incorporation of fins on the fluid side within plate-fin heat exchanger geometries was studied numerically by introducing fins on the fluid side (Aim 2, Obj 2). For this, a heat exchanger with no fins, continuous fins and strip fins in the fluid layer was considered.
- The onset of convection was studied for a rectangular cavity based on the operating conditions for an aircraft, where the onset was observed to occur significantly after the end of the melting cycle. As a result, convection was ignored for the rest of this chapter.
- The addition of fins was shown to greatly improve the heat transfer performance of the heat exchanger, where both the effectiveness and amount of heat that was stored in the PCM increased. This was attributed to the increased heat transfer coefficient of the finned surfaces. A small increase in the heat transfer performance was noted by using strip fins, rather than continuous fins, while this significantly increased the pressure drop.
- The use of finned surfaces led to very uniform melting of the PCM at the low range of Reynolds number that was studied ($Re = [50; 200]$), where an increase of Reynolds number led to a marginal increase in the amount of stored heat. For the plate heat exchanger case, a significantly higher Reynolds number of $Re > 20,000$ was required for the melting to be uniform and the amount of stored heat to not change significantly.

Chapter 6: Enhancement of melting by using extended surfaces in PCM

- The incorporation of fins on the PCM side within plate-fin heat exchanger geometries was studied numerically to study the enhancement that can be obtained from the fins (Aim 2, Obj 3).
- The effect of changing the size and the aspect ratio of a two dimensional finned rectangular cavity was initially studied over a range of wall temperatures applied on the lower wall. This found that creating thin rectangular sections would lead to a faster melting of the PCM than square PCM for a similar volume of PCM. Contrary to what was observed in the previous chapter, the onset of convection occurred during the melting cycle. Multiple recirculation zones in the PCM were observed, which led to a change the shape of the melt front, but only marginally increased the amount of heat that could be stored

in the PCM.

- A three dimensional compact heat exchanger geometry with a finned PCM layer was created by choosing the dimensions based on the findings from the two dimensional study and the melting of PCM was studied over a range of Reynolds number. It was found that the amount of PCM that was melted only marginally increased past $Re = 100$, where the difference in melted PCM mostly changed towards the outlet due to the a lower melt uniformity at low Re .
- The addition of fins in the PCM layer significantly improved the heat transfer when compared to the heat exchanger without fins in the PCM, leading to an average increase of 8.7% of heat stored in the PCM.

Chapter 7: Development of a low cost model for PCM compact heat exchanger

- This chapter presented a cost-effective tool based on the ϵ -NTU model, incorporating various formulations to predict the melting rate for both plain and finned PCM layers.
- The ϵ -NTU model for unfinned PCM layers employing a linear 1D Stefan approximation to predict the phase change, demonstrated good agreement with CFD results. Replacing the linear Stefan model with an energy conservation approach maintained the accuracy of the model while increasing its flexibility.
- A model for finned cavities was developed that used adapted surface effectiveness and shape factors to map 2D heat transfer into 1D equations. This achieved good agreement within 7% with CFD results for rectangular cavities, although predictions for triangular fins were only accurate for equilateral geometries. The formulation was incorporated within the $\epsilon - NTU$ model to model compact heat exchanger geometries with finned PCM layers, which accurately predicted the melting of the PCM at high Reynolds numbers where transient effects were negligible.
- The model's utility was illustrated with two design examples under industrial conditions in the Reynolds range that previously showed good agreement with CFD results. This highlighted the ability of the model to be used within large parametric studies and optimization work due to its low computational cost.

8.2. Implication of results

This thesis explored innovative PCM heat exchanger geometries that have been largely overlooked in previous research, but are essential for achieving the high thermal loads and short melting cycles required in high-performance aircraft applications. By incorporating fins into both the PCM and fluid layers, this study demonstrated substantial improvements in the thermal performance when compared to plate heat exchangers. This advancement significantly increased the amount of PCM melted within the short melting cycles demanded by aerospace systems.

Computational Fluid Dynamics (CFD) proved invaluable for gaining detailed insights into the melting behaviour of PCM within compact heat exchangers. Its ability to provide detailed data on both thermo-hydraulic performance and phase change processes makes it an indispensable tool for exploring complex phenomena that can be challenging to capture experimentally for such geometries. The thesis also noted the considerable computational cost involved in accurately predicting melting behaviour, particularly for geometries with a large number of corrugations.

The high computational cost limits the suitability of CFD for conducting extensive studies on the initial sizing of the heat exchanger or optimization across geometries. To address these challenges, an ϵ -NTU model was developed that offers reliable sizing estimates for heat exchangers at a fraction of the cost associated with CFD or experimental approaches. By leveraging key inputs that can be reliably approximated using CFD, such as heat transfer coefficients, the simplified model achieves a balance between accuracy and efficiency. The proposed model provides exceptional flexibility, allowing for rapid adjustments to dimensions and operating conditions, capabilities that are difficult to achieve with alternative methods.

8.3. Recommendations for future work

Some of the key areas that were identified to need further research are summarized here:

- This work only considered constrained melting of PCM. The solidification process could be considered and the effect of unconstrained melting investigated in future work.
- Conducting experiments of plate-fin heat exchangers would allow for the CFD models to be validated and allow to further develop the computational model.

Additionally, a better value of mushy zone constant could be chosen based on experimental work, as only an approximation was used in this work justified by the small amount of convection in the PCM.

- Only small sections could be modelled with CFD in this thesis linked to the large computational cost of modelling the individual corrugations. Further work could involve developing a porous model of the heat exchanger corrugations, similar to work conducted by Greiciunas et al. (2019a) and McDermott & He (2024). This would make it possible to explore the effect of design challenges such as flow maldistribution on the melting of the PCM in compact geometries. As part of the numerical work, implementing an advanced time step control system based on transferred heat would be beneficial in reducing the computational cost of future simulations.
- The $\epsilon - NTU$ model that was developed in Chapter 7 could be improved to cover a larger spectrum of operating conditions and geometries. Currently, only large Reynolds numbers can be considered, due to the steady-state assumption that was used within the model. This could be achieved by expanding the model to account for the transient response of the fluid channel to the PCM channel, leading to a more accurate prediction at low Reynolds number. Additionally, a large discrepancy was found for the prediction of the melting rate of triangular finned surfaces in the PCM channel at $AR \neq 1$, attributed to the simplistic shape factor relationship. Further investigations are required in this area.
- The model could be used to conduct an optimization study that covers a wider range of parameters than what was used in this thesis. For instance, the study could include exploring different finned surfaces in the fluid layer, the use of various phase change materials and operating conditions. Additionally, the performance of the heat exchanger could be explored within the thermal management system.

Bibliography

Abdi, A. & Rastan, H. (2023), ‘Experimental comparative analysis of close-contact and constrained melting of n-eicosane in a finned rectangular cavity’, *Applied Thermal Engineering* **219**, 119677.

URL: <https://www.sciencedirect.com/science/article/pii/S1359431122016076>

Abdulateef, A. M., Jaszczur, M., Hassan, Q., Anish, R., Niyas, H., Sopian, K. & Abdulateef, J. (2021), ‘Enhancing the melting of phase change material using a finsnanoparticle combination in a triplex tube heat exchanger’, *Journal of Energy Storage* **35**, 102227.

URL: <https://www.sciencedirect.com/science/article/pii/S2352152X20320491>

Agyenim, F., Eames, P. & Smyth, M. (2009), ‘A comparison of heat transfer enhancement in a medium temperature thermal energy storage heat exchanger using fins’, *Solar Energy* **83**(9), 1509–1520.

URL: <https://www.sciencedirect.com/science/article/pii/S0038092X09000942>

Agyenim, F. & Hewitt, N. (2010), ‘The development of a finned phase change material (PCM) storage system to take advantage of off-peak electricity tariff for improvement in cost of heat pump operation’, *Energy and Buildings* **42**(9), 1552–1560.

URL: <https://www.sciencedirect.com/science/article/pii/S0378778810001192>

Agyenim, F., Hewitt, N., Eames, P. & Smyth, M. (2010), ‘A review of materials, heat transfer and phase change problem formulation for latent heat thermal energy storage systems (LHTESS)’, *Renewable and Sustainable Energy Reviews* **14**(2), 615–628.

URL: <https://www.sciencedirect.com/science/article/pii/S1364032109002469>

Ahlers, M. F. (2020), *Introduction to Aircraft Thermal Management*, SAE International.

URL: <https://app.knovel.com/hotlink/toc/id:kpIATM000M/introduction-aircraft/introduction-aircraft>

Al-Ketan, O., Lee, D.-W., Rowshan, R. & Abu Al-Rub, R. K. (2020), ‘Functionally graded and multi-morphology sheet TPMS lattices: Design, manufacturing, and mechanical properties’, *Journal of the Mechanical Behavior of Biomedical Materials* **102**, 103520.

URL: <https://www.sciencedirect.com/science/article/pii/S1751616119312482>

- Al-mudhafar, A. H. N. (2019), Innovative heat transfer enhancements for thermal energy storage systems based on phase change materials, phd, University of Sheffield.
URL: <https://etheses.whiterose.ac.uk/25192/>
- AL-Saadi, S. N. & Zhai, Z. J. (2013), ‘Modeling phase change materials embedded in building enclosure: A review’, *Renewable and Sustainable Energy Reviews* **21**, 659–673.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032113000555>
- Amano, R. (1987), ‘A Numerical Study of Turbulent Heat Transfer in a Channel with Bends Using Reynolds-Stress Model’, *Chemical Engineering Communications* **51**(1-6), 207–219. Publisher: Taylor & Francis _eprint: <https://doi.org/10.1080/00986448708911844>.
URL: <https://doi.org/10.1080/00986448708911844>
- Amin, N. A. M., Belusko, M. & Bruno, F. (2014), ‘An effectiveness-NTU model of a packed bed PCM thermal storage system’, *Applied Energy* **134**, 356–362.
URL: <https://www.sciencedirect.com/science/article/pii/S030626191400823X>
- Anibal, J. L. & Martins, J. R. R. A. (2024), ‘Adjoint-based shape optimization of a plate-fin heat exchanger using CFD’, *Applied Thermal Engineering* **252**, 123570.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431124012389>
- ANSYS FLUENT Theory Guide* (2009).
- Ardahaie, S. S., Hosseini, M. J., Ranjbar, A. A. & Rahimi, M. (2019), ‘Energy storage in latent heat storage of a solar thermal system using a novel flat spiral tube heat exchanger’, *Applied Thermal Engineering* **159**, 113900.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431118369904>
- Asli, M., König, P., Sharma, D., Pontika, E., Huete, J., Konda, K. R., Mathiazhagan, A., Xie, T., Höschler, K. & Laskaridis, P. (2024), ‘Thermal management challenges in hybrid-electric propulsion aircraft’, *Progress in Aerospace Sciences* **144**, 100967.
URL: <https://www.sciencedirect.com/science/article/pii/S0376042123000830>
- Attarzadeh, R., Attarzadeh-Niaki, S.-H. & Duwig, C. (2022), ‘Multi-objective optimization of TPMS-based heat exchangers for low-temperature waste heat recovery’, *Applied Thermal Engineering* **212**, 118448.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431122004033>
- Awais, M. & Bhuiyan, A. A. (2018), ‘Heat and mass transfer for compact heat exchanger (CHXs) design: A state-of-the-art review’, *International Journal of Heat and Mass Transfer* **127**, 359–380.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931018311499>
- Aziz, N. A., Amin, N. A. M., Majid, M. S. A., Bruno, F. & Belusko, M. (2022), ‘Optimising a packed bed phase change material of spheres using effectiveness-number of transfer unit method’, *Journal of Energy Storage* **49**, 104019.
URL: <https://www.sciencedirect.com/science/article/pii/S2352152X22000627>

- Barreneche, C., Navarro, H., Serrano, S., Cabeza, L. F. & Fernández, A. I. (2014), 'New Database on Phase Change Materials for Thermal Energy Storage in Buildings to Help PCM Selection', *Energy Procedia* **57**, 2408–2415.
URL: <https://www.sciencedirect.com/science/article/pii/S1876610214016166>
- Barz, T. & Emhofer, J. (2021), 'Paraffins as phase change material in a compact plate-fin heat exchanger - Part I: Experimental analysis and modeling of complete phase transitions', *Journal of Energy Storage* **33**, 102128.
URL: <https://www.sciencedirect.com/science/article/pii/S2352152X20319551>
- Baal, B. & Ünal, A. (2013), 'Numerical evaluation of a triple concentric-tube latent heat thermal energy storage', *Solar Energy* **92**, 196–205.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X13000996>
- Beck, A., Koller, M., Walter, H. & Hameter, M. (2015), Transient Numerical Analysis of Different Finned Tube Designs for Use in Latent Heat Thermal Energy Storage Devices, in 'Volume 2', American Society of Mechanical Engineers, San Diego, California, USA, p. V002T13A004.
URL: <https://doi.org/10.1115/ES2015-49145>
- Bejan, A. (1996), 'Entropy generation minimization: The new thermodynamics of finitesize devices and finitetime processes', *Journal of Applied Physics* **79**(3), 1191–1218.
URL: <https://doi.org/10.1063/1.362674>
- Bejarano, G., Suffo, J. J., Vargas, M. & Ortega, M. G. (2018), 'Novel scheme for a PCM-based cold energy storage system. Design, modelling, and simulation', *Applied Thermal Engineering* **132**, 256–274.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431117350809>
- Belusko, M., Halawa, E. & Bruno, F. (2012), 'Characterising PCM thermal storage systems using the effectiveness-NTU approach', *International Journal of Heat and Mass Transfer* **55**(13), 3359–3365.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931012001676>
- Belusko, M., Tay, N. H. S., Liu, M. & Bruno, F. (2016), 'Effective tube-in-tank PCM thermal storage for CSP applications, Part 2: Parametric assessment and impact of latent fraction', *Solar Energy* **139**, 744–756.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X15005204>
- Bhutta, A. M. M., Hayat, N., Bashir, M. H., Khan, A. R., Ahmad, K. N. & Khan, S. (2012), 'CFD applications in various heat exchangers design: A review', *Applied Thermal Engineering* **32**, 1–12.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431111004807>
- Borrajó-Peláez, R., Ortega-Casanova, J. & Cejudo-López, J. M. (2010), 'A three-dimensional numerical study and comparison between the air side model and the air/water side model of a plain fin-and-tube heat exchanger', *Applied Thermal Engineering* **30**(13), 1608–1615.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431110001286>

- Browne, M. W. & Bansal, P. K. (2001), 'An elemental NTU- model for vapour-compression liquid chillers', *International Journal of Refrigeration* **24**(7), 612–627.
URL: <https://www.sciencedirect.com/science/article/pii/S0140700700000918>
- Bruno, F., Belusko, M. & Tay, N. H. (2010), Design of PCM Thermal Storage Systems Using the Effectiveness-NTU Method, in 'Proceedings of the EuroSun 2010 Conference', International Solar Energy Society, Graz, Austria, pp. 1–8.
URL: <http://proceedings.ises.org/citation?doi=eurosun.2010.16.01>
- Bruno, F., Tay, N. H. S. & Belusko, M. (2014), 'Minimising energy usage for domestic cooling with off-peak PCM storage', *Energy and Buildings* **76**, 347–353.
URL: <https://www.sciencedirect.com/science/article/pii/S0378778814002059>
- Cao, Y. & Faghri, A. (1991), 'Performance characteristics of a thermal energy storage module: a transient PCM/forced convection conjugate analysis', *International Journal of Heat and Mass Transfer* **34**(1), 93–101.
URL: <https://www.sciencedirect.com/science/article/pii/001793109190177G>
- Cong, L., Zou, B., Palacios, A., Navarro, M. E., Qiao, G. & Ding, Y. (2022), 'Thickening and gelling agents for formulation of thermal energy storage materials A critical review', *Renewable and Sustainable Energy Reviews* **155**, 111906.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032121011722>
- de Gracia, A. & Cabeza, L. F. (2017), 'Numerical simulation of a PCM packed bed system: A review', *Renewable and Sustainable Energy Reviews* **69**, 1055–1063.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032116305822>
- Dekhil, M. A., Simo Tala, J. V., Bulliard-Sauret, O. & Bougeard, D. (2021), 'Numerical analysis of the performance enhancement of a latent heat storage shell and tube unit using finned tubes during melting and solidification', *Applied Thermal Engineering* **192**, 116866.
URL: <https://www.sciencedirect.com/science/article/pii/S135943112100315X>
- Deng, S., Nie, C., Jiang, H. & Ye, W.-B. (2019), 'Evaluation and optimization of thermal performance for a finned double tube latent heat thermal energy storage', *International Journal of Heat and Mass Transfer* **130**, 532–544.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931018324359>
- Diaconu, B. M., Cruceru, M. & Anghelescu, L. (2022), 'Phase Change Materials Applications and Systems Designs: A Literature Review', *Designs* **6**(6), 117. Number: 6 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/2411-9660/6/6/117>
- Diaconu, B. M., Cruceru, M. & Anghelescu, L. (2024), 'Phase change materials in space systems. Fundamental applications, materials and special requirements A review', *Acta Astronautica* **216**, 163–213.
URL: <https://www.sciencedirect.com/science/article/pii/S0094576523006707>
- Dixit, T., Al-Hajri, E., Paul, M. C., Nithiarasu, P. & Kumar, S. (2022), 'High

- performance, microarchitected, compact heat exchanger enabled by 3D printing', *Applied Thermal Engineering* **210**, 118339.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431122002976>
- Elbahjaoui, R. & El Qarnia, H. (2017), 'Transient behavior analysis of the melting of nanoparticle-enhanced phase change material inside a rectangular latent heat storage unit', *Applied Thermal Engineering* **112**, 720–738.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431116325662>
- Esapour, M., Hosseini, M. J., Ranjbar, A. A., Pahlavani, Y. & Bahrampoury, R. (2016), 'Phase change in multi-tube heat exchangers', *Renewable Energy* **85**, 1017–1025.
URL: <https://www.sciencedirect.com/science/article/pii/S0960148115301580>
- Esen, M., Durmu, A. & Durmu, A. (1998), 'Geometric design of solar-aided latent heat store depending on various parameters and phase change materials', *Solar Energy* **62**(1), 19–28.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X97001047>
- Faden, M., König-Haagen, A., Höhle, S. & Brüggemann, D. (2018), 'An implicit algorithm for melting and settling of phase change material inside macrocapsules', *International Journal of Heat and Mass Transfer* **117**, 757–767.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931017329241>
- Fallahi, A., Guldentops, G., Tao, M., Granados-Focil, S. & Van Dessel, S. (2017), 'Review on solid-solid phase change materials for thermal energy storage: Molecular structure and thermal properties', *Applied Thermal Engineering* **127**, 1427–1441.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431117329939>
- Gardner, K. A. (1945), 'Efficiency of Extended Surface', *Journal of Fluids Engineering* **67**(8), 621–628.
URL: <https://asmedigitalcollection.asme.org/fluidsengineering/article/67/8/621/1154082/Efficiency-of-Extended-Surface>
- Gartling, D. K. (1978), Finite element analysis of convective heat transfer problems with change of phase, Technical Report SAND-77-2037C; CONF-780712-2, Sandia Labs., Albuquerque, N.Mex. (USA).
URL: <https://www.osti.gov/biblio/6758731>
- Gharebaghi, M. & Sezai, I. (2007), 'Enhancement of Heat Transfer in Latent Heat Storage Modules with Internal Fins', *Numerical Heat Transfer, Part A: Applications* **53**(7), 749–765. Publisher: Taylor & Francis _eprint: <https://doi.org/10.1080/10407780701715786>.
URL: <https://doi.org/10.1080/10407780701715786>
- Greiciunas, E. (2019), Modelling Novel Heat Exchangers for Aircraft Thermal Management, phd, University of Leeds.
URL: <https://etheses.whiterose.ac.uk/24232/>

- Greiciunas, E., Borman, D., Summers, J. & Smith, S. J. (2019a), 'A multi-scale conjugate heat transfer modelling approach for corrugated heat exchangers', *International Journal of Heat and Mass Transfer* **139**, 928–937.
URL: <https://linkinghub.elsevier.com/retrieve/pii/S0017931019302455>
- Greiciunas, E., Borman, D., Summers, J. & Smith, S. J. (2019b), 'A numerical evaluation of next generation additive layer manufactured inter-layer channel heat exchanger', *Applied Thermal Engineering* **162**, 114304.
URL: <https://linkinghub.elsevier.com/retrieve/pii/S1359431119334209>
- Gürel, B. (2020), 'A numerical investigation of the melting heat transfer characteristics of phase change materials in different plate heat exchanger (latent heat thermal energy storage) systems', *International Journal of Heat and Mass Transfer* **148**, 119117.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931019319003>
- Hameter, M. & Walter, H. (2016), Influence of the Mushy Zone Constant on the Numerical Simulation of the Melting and Solidification Process of Phase Change Materials, Vol. 38A, pp. 439–444.
- Hamidi, E., Ganesan, P. B., Sharma, R. K. & Yong, K. W. (2023), 'Computational study of heat transfer enhancement using porous foams with phase change materials: A comparative review', *Renewable and Sustainable Energy Reviews* **176**, 113196.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032123000527>
- Hassani, A. V., Hollands, K. G. T. & Raithby, G. D. (1993), 'A close upper bound for the conduction shape factor of a uniform thickness, 2D layer', *International Journal of Heat and Mass Transfer* **36**(12), 3155–3158.
URL: <https://www.sciencedirect.com/science/article/pii/0017931093900447>
- Hayes, D., Lone, M., Whidborne, J. F., Camberos, J. & Coetzee, E. (2017), 'Adopting exergy analysis for use in aerospace', *Progress in Aerospace Sciences* **93**, 73–94.
URL: <https://www.sciencedirect.com/science/article/pii/S0376042117300623>
- Herbinger, F., Bhourri, M. & Groulx, D. (2018), 'Investigation of heat transfer inside a PCM-air heat exchanger: a numerical parametric study', *Heat and Mass Transfer* **54**(8), 2433–2442.
URL: <http://link.springer.com/10.1007/s00231-017-2101-9>
- Hesselgreaves, J. E., Law, R., Reay, D. A. & Heggs, P. J. (2017), *Compact heat exchangers : selection, design, and operation*, second edition. edn, Butterworth-Heinemann, Amsterdam, Netherlands. Publication Title: Compact heat exchangers : selection, design, and operation.
- Hiramatsu, M., Ishimaru, T. & Ohkouchi, T. (1992), 'Numerical Analysis of Innerfins for Intercoolers', *JSME international journal. Ser. 2, Fluids engineering, heat transfer, power, combustion, thermophysical properties* **35**(3), 406–412.
- Homitz, J., Tucker, B. P. & Messmer, J. M. (2012), Thermal Storage Device for

- High-Power-Density Systems, in 'Volume 2: Heat Transfer Enhancement for Practical Applications; Fire and Combustion; Multi-Phase Systems; Heat Transfer in Electronic Equipment; Low Temperature Heat Transfer; Computational Heat Transfer', American Society of Mechanical Engineers, Rio Grande, Puerto Rico, USA, pp. 465–472.
URL: <https://asmedigitalcollection.asme.org/HT/proceedings/HT2012/44786/465/245196>
- Hoseinzadeh, S., Ghasemiasl, R., Havaei, D. & Chamkha, A. J. (2018), 'Numerical investigation of rectangular thermal energy storage units with multiple phase change materials', *Journal of Molecular Liquids* **271**, 655–660.
URL: <https://www.sciencedirect.com/science/article/pii/S0167732218338054>
- Hosseini, M. J., Rahimi, M. & Bahrampoury, R. (2014), 'Experimental and computational evolution of a shell and tube heat exchanger as a PCM thermal storage system', *International Communications in Heat and Mass Transfer* **50**, 128–136.
URL: <https://www.sciencedirect.com/science/article/pii/S0735193313002194>
- Hosseini, M. J., Ranjbar, A. A., Sedighi, K. & Rahimi, M. (2012), 'A combined experimental and computational study on the melting behavior of a medium temperature phase change storage material inside shell and tube heat exchanger', *International Communications in Heat and Mass Transfer* **39**(9), 1416–1424.
URL: <https://www.sciencedirect.com/science/article/pii/S0735193312001881>
- Hou, F., Cao, S. & Wang, H. (2021), 'New Equivalent Thermal Conductivity Model for Size-Dependent Convection-Driven Melting of Spherically Encapsulated Phase Change Material', *Materials* **14**(16), 4752. Number: 16 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/1996-1944/14/16/4752>
- Hsieh, C.-T. & Jang, J.-Y. (2012), 'Parametric study and optimization of louver finned-tube heat exchangers by Taguchi method', *Applied Thermal Engineering* **42**, 101–110.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431112001585>
- Hu, H. & Argyropoulos, S. A. (1996), 'Mathematical modelling of solidification and melting: a review', *Modelling and Simulation in Materials Science and Engineering* **4**(4), 371–396.
URL: <https://iopscience.iop.org/article/10.1088/0965-0393/4/4/004>
- Ismail, L. S. & Velraj, R. (2009), 'Studies on Fanning Friction (f) and Colburn (j) Factors of Offset and Wavy Fins Compact Plate Fin Heat Exchanger A CFD Approach', *Numerical Heat Transfer, Part A: Applications* **56**(12), 987–1005.
URL: <http://www.tandfonline.com/doi/abs/10.1080/10407780903507957>
- Jaguemont, J., Omar, N., Van den Bossche, P. & Mierlo, J. (2018), 'Phase-change materials (PCM) for automotive applications: A review', *Applied Thermal Engineering* **132**, 308–320.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431117319762>
- Jany, P. & Bejan, A. (1988), 'Scaling theory of melting with natural convection in

- an enclosure', *International Journal of Heat and Mass Transfer* **31**(6), 1221–1235.
URL: <https://www.sciencedirect.com/science/article/pii/0017931088900658>
- Jennions, I., Ali, F., Miguez, M. E. & Escobar, I. C. (2020), 'Simulation of an aircraft environmental control system', *Applied Thermal Engineering* **172**, 114925.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431119350811>
- Ji, C., Qin, Z., Low, Z., Dubey, S., Choo, F. H. & Duan, F. (2018), 'Non-uniform heat transfer suppression to enhance PCM melting by angled fins', *Applied Thermal Engineering* **129**, 269–279.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431117356806>
- Kalapala, L. & Devanuri, J. K. (2018), 'Influence of operational and design parameters on the performance of a PCM based heat exchanger for thermal energy storage A review', *Journal of Energy Storage* **20**, 497–519.
URL: <https://www.sciencedirect.com/science/article/pii/S2352152X18304523>
- Kays, W. M. & London, A. L. (1964), *Compact Heat Exchangers*, 2 edn, McGraw-Hill Book Company.
- Khan, Z. & Khan, Z. A. (2017), 'Experimental investigations of charging/melting cycles of paraffin in a novel shell and tube with longitudinal fins based heat storage design solution for domestic and industrial applications', *Applied Energy* **206**, 1158–1168.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261917314563>
- Khare, S., DellAmico, M., Knight, C. & McGarry, S. (2012), 'Selection of materials for high temperature latent heat energy storage', *Solar Energy Materials and Solar Cells* **107**, 20–27.
URL: <https://www.sciencedirect.com/science/article/pii/S0927024812003832>
- Kim, D., Abdallah, S., Bosi, G. & Hales, A. (2023), 'A Numerical Study of the Suitability of Phase-Change Materials for Battery Thermal Management in Flight Applications', *World Electric Vehicle Journal* **14**(1), 15. Number: 1 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/2032-6653/14/1/15>
- Kobayashi, H., Yaji, K., Yamasaki, S. & Fujita, K. (2019), 'Freeform winglet design of fin-and-tube heat exchangers guided by topology optimization', *Applied Thermal Engineering* **161**, 114020.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431118377135>
- Koyuncu, A. (2023), Analysis of Phase Change Material Heat Sink, PhD thesis, Middle East Technical University.
- Lacroix, M. (1993), 'Numerical simulation of a shell-and-tube latent heat thermal energy storage unit', *Solar Energy* **50**(4), 357–367.
URL: <https://www.sciencedirect.com/science/article/pii/0038092X9390029N>
- Lacroix, M. & Benmadda, M. (1997), 'Numerical Simulation of Natural Convection-Dominated Melting and Solidification from a Finned Vertical Wall', *Numerical*

- Heat Transfer, Part A: Applications* **31**(1), 71–86. Publisher: Taylor & Francis
 _eprint: <https://doi.org/10.1080/10407789708914026>.
URL: <https://doi.org/10.1080/10407789708914026>
- Lacroix, M. & Benmadda, M. (1998), ‘Analysis of natural convection melting from a heated wall with vertically oriented fins’, *International Journal of Numerical Methods for Heat & Fluid Flow* **8**(4), 465–478. Num Pages: 14
 Place: Bradford, United Kingdom Publisher: Emerald Group Publishing Limited.
URL: <https://www.proquest.com/docview/215206870/abstract/DF3303DEC928475APQ/1>
- Ladekar, C. L., Chaudhary, S. K. & Khandare, S. S. (2017), ‘Experimental Investigate for Optimization of Heat Pipe Performance in Latent Heat Thermal Energy Storage’, *Materials Today: Proceedings* **4**(8), 8149–8157.
URL: <https://www.sciencedirect.com/science/article/pii/S2214785317314530>
- Launder, B. E. & Sharma, B. I. (1974), ‘Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc’, *Letters in Heat and Mass Transfer* **1**(2), 131–137.
URL: <https://www.sciencedirect.com/science/article/pii/0094454874901507>
- Lee, K. O., Medina, M. A., Sun, X. & Jin, X. (2018), ‘Thermal performance of phase change materials (PCM)-enhanced cellulose insulation in passive solar residential building walls’, *Solar Energy* **163**, 113–121.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X18301075>
- Li, Y., Kong, B., Qiu, C., Li, Y. & Jiang, Y. (2025), ‘Numerical study on air-cooled battery thermal management system considering the sheer altitude effect’, *Applied Thermal Engineering* **258**, 124707.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431124023755>
- Liang, H., Niu, J. & Gan, Y. (2020), ‘Performance optimization for shell-and-tube PCM thermal energy storage’, *Journal of Energy Storage* **30**, 101421.
URL: <https://www.sciencedirect.com/science/article/pii/S2352152X19316640>
- Liu, M., Bruno, F. & Saman, W. (2011), ‘Thermal performance analysis of a flat slab phase change thermal storage unit with liquid-based heat transfer fluid for cooling applications’, *Solar Energy* **85**(11), 3017–3027.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X11003240>
- Liu, Q., He, Y. & Li, Q. (2017), ‘Enthalpy-based multiple-relaxation-time lattice Boltzmann method for solid-liquid phase-change heat transfer in metal foams’, *PHYSICAL REVIEW E* **96**(2).
- Lv, Y.-G., Zhang, G.-P., Wang, Q.-W. & Chu, W.-X. (2022), ‘Thermal Management Technologies Used for High Heat Flux Automobiles and Aircraft: A Review’, *Energies* **15**(21), 8316. Number: 21 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/1996-1073/15/21/8316>
- Ma, Z. & Zhang, Y. (2006), ‘Solid velocity correction schemes for a temperature

- transforming model for convection phase change', *International Journal of Numerical Methods for Heat & Fluid Flow* **16**(2), 204–225. Publisher: Emerald Group Publishing Limited.
URL: <https://doi.org/10.1108/09615530610644271>
- Mahani, R. B., Mohammed, H. I., Mahdi, J. M., Alamshahi, F., Ghalambaz, M., Talebizadehsardari, P. & Yaïci, W. (2020), 'Phase Change Process in a Zigzag Plate Latent Heat Storage System during Melting and Solidification', *Molecules* **25**(20), 4643. Number: 20 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/1420-3049/25/20/4643>
- Mahdi, J., Najim, F., Aljubury, I., Mohammed, H., Ben Khedher, N., Alshammari, N., Cairns, A. & Talebizadehsardari, P. (2022), 'Intensifying the thermal response of PCM via fin-assisted foam strips in the shell-and-tube heat storage system', *JOURNAL OF ENERGY STORAGE* **45**.
- Marcinkowski, M., Taler, D., Sacharczuk, J., Wglarz, K. & Taler, J. (2024), 'Innovative analysis of local and average air-side heat transfer coefficients in fin-and-tube heat exchangers using CFD and experimental method', *Energy* **309**, 133084.
URL: <https://www.sciencedirect.com/science/article/pii/S0360544224028597>
- McDermott, M. & He, S. (2024), 'An open-source porous media modelling approach to investigate thermohydraulic features of compact printed circuit heat exchangers', *Nuclear Engineering and Design* **421**, 113084.
URL: <https://www.sciencedirect.com/science/article/pii/S0029549324001845>
- Mehling, H., Brütting, M. & Haussmann, T. (2022), 'PCM products and their fields of application - An overview of the state in 2020/2021', *Journal of Energy Storage* **51**, 104354.
URL: <https://www.sciencedirect.com/science/article/pii/S2352152X22003784>
- Menter, F. R. (1994), 'Two-equation eddy-viscosity turbulence models for engineering applications', *AIAA Journal* **32**(8), 1598–1605.
URL: <https://arc.aiaa.org/doi/10.2514/3.12149>
- Menéndez Pérez, A., Fernández-Aballí Altamirano, C. & Borrajo Pérez, R. (2022), 'Parametric analysis of the influence of geometric variables of vortex generators on compact louver fin heat exchangers', *Thermal Science and Engineering Progress* **27**, 101151.
URL: <https://www.sciencedirect.com/science/article/pii/S2451904921003103>
- Mesalhy, O., Lafdi, K., Elgafy, A. & Bowman, K. (2005), 'Numerical study for enhancing the thermal conductivity of phase change material (PCM) storage using high thermal conductivity porous matrix', *ENERGY CONVERSION AND MANAGEMENT* **46**(6), 847–867.
- Morovat, N., Athienitis, A. K., Candanedo, J. A. & Dermardiros, V. (2019), 'Simulation and performance analysis of an active PCM-heat exchanger intended for

- building operation optimization', *Energy and Buildings* **199**, 47–61.
URL: <https://www.sciencedirect.com/science/article/pii/S037877881930578X>
- Motahar, S. & Khodabandeh, R. (2016), 'Experimental study on the melting and solidification of a phase change material enhanced by heat pipe', *International Communications in Heat and Mass Transfer* **73**, 1–6.
URL: <https://www.sciencedirect.com/science/article/pii/S0735193316300495>
- Nazir, H., Batool, M., Bolivar Osorio, F. J., Isaza-Ruiz, M., Xu, X., Vignarooban, K., Phelan, P., Inamuddin & Kannan, A. M. (2019), 'Recent developments in phase change materials for energy storage applications: A review', *International Journal of Heat and Mass Transfer* **129**, 491–523.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931018324578>
- Nazzi Ehms, J. H., De Césaró Oliveski, R., Oliveira Rocha, L. A., Biserni, C. & Garai, M. (2019), 'Fixed Grid Numerical Models for Solidification and Melting of Phase Change Materials (PCMs)', *Applied Sciences* **9**(20), 4334. Number: 20 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/2076-3417/9/20/4334>
- Nemati, H., Souriaee, V., Habibi, M. & Soharbi, N. (2023), 'How to simulate phase change material melting? A critical look at different methods with different levels of accuracy and difficulty', *Journal of the Brazilian Society of Mechanical Sciences and Engineering* **46**(1), 49.
URL: <https://doi.org/10.1007/s40430-023-04620-0>
- Ning, J., Wang, X., Sun, Y., Zheng, C., Zhang, S., Zhao, X., Liu, C. & Yan, W. (2022), 'Experimental and numerical investigation of additively manufactured novel compact plate-fin heat exchanger', *International Journal of Heat and Mass Transfer* **190**, 122818.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931022003003>
- Oró, E., de Gracia, A., Castell, A., Farid, M. M. & Cabeza, L. F. (2012), 'Review on phase change materials (PCMs) for cold thermal energy storage applications', *Applied Energy* **99**, 513–533.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261912002784>
- Parisher, R. A. & Rhea, R. A. (2012), Chapter 6 - Mechanical Equipment, in R. A. Parisher & R. A. Rhea, eds, 'Pipe Drafting and Design (Third Edition)', Gulf Professional Publishing, Boston, pp. 112–133.
URL: <https://www.sciencedirect.com/science/article/pii/B9780123847003000062>
- Patankar, S. V., Liu, C. H. & Sparrow, E. M. (1977), 'Fully Developed Flow and Heat Transfer in Ducts Having Streamwise-Periodic Variations of Cross-Sectional Area', *Journal of Heat Transfer* **99**(2), 180–186.
URL: <https://doi.org/10.1115/1.3450666>
- Patel, V. P., Koerner, M. & Loeffelholz, D. (2010), Thermal Management and Power Generation for Directed Energy Weapons, SAE Technical Paper 2010-01-1781, SAE International, Warrendale, PA. ISSN: 0148-7191, 2688-3627.

- URL:** <https://www.sae.org/publications/technical-papers/content/2010-01-1781/>
- Peiró, G., Gasia, J., Miró, L. & Cabeza, L. F. (2015), ‘Experimental evaluation at pilot plant scale of multiple PCMs (cascaded) vs. single PCM configuration for thermal energy storage’, *Renewable Energy* **83**, 729–736.
URL: <https://www.sciencedirect.com/science/article/pii/S0960148115004085>
- Pizzolato, A., Sharma, A., Maute, K., Sciacovelli, A. & Verda, V. (2017), ‘Design of effective fins for fast PCM melting and solidification in shell-and-tube latent heat thermal energy storage through topology optimization’, *Applied Energy* **208**, 210–227.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261917314642>
- Poirier, D. & Salcudean, M. (1988), ‘On Numerical Methods Used in Mathematical Modeling of Phase Change in Liquid Metals’, *Journal of Heat Transfer* **110**(3), 562–570.
URL: <https://doi.org/10.1115/1.3250529>
- Price, D. (2003), ‘A review of selected thermal management solutions for military electronic systems’, *IEEE Transactions on Components and Packaging Technologies* **26**(1), 26–39. Conference Name: IEEE Transactions on Components and Packaging Technologies.
URL: <https://ieeexplore.ieee.org/document/1202900>
- Promoppatum, P., Yao, S.-C., Hultz, T. & Agee, D. (2017), ‘Experimental and numerical investigation of the cross-flow PCM heat exchanger for the energy saving of building HVAC’, *Energy and Buildings* **138**, 468–478.
URL: <https://www.sciencedirect.com/science/article/pii/S037877881631876X>
- Pu, L., Zhang, S., Xu, L. & Li, Y. (2020), ‘Thermal performance optimization and evaluation of a radial finned shell-and-tube latent heat thermal energy storage unit’, *Applied Thermal Engineering* **166**, 114753.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431119349762>
- Qasem, N. A. A. & Zubair, S. M. (2018), ‘Compact and microchannel heat exchangers: A comprehensive review of air-side friction factor and heat transfer correlations’, *Energy Conversion and Management* **173**, 555–601.
URL: <https://www.sciencedirect.com/science/article/pii/S0196890418307131>
- Qureshi, Z. A., Al-Omari, S. A. B., Elnajjar, E., Al-Ketan, O. & Al-Rub, R. A. (2021), ‘Using triply periodic minimal surfaces (TPMS)-based metal foams structures as skeleton for metal-foam-PCM composites for thermal energy storage and energy management applications’, *International Communications in Heat and Mass Transfer* **124**, 105265.
URL: <https://www.sciencedirect.com/science/article/pii/S0735193321001597>
- Rohsenow, W. M., Hartnett, J. P. & Cho, Y. I., eds (1998), *Handbook of heat transfer*, McGraw-Hill handbooks, 3rd ed edn, McGraw-Hill, New York.

- Rösler, F. (2014), *Modellierung und Simulation der Phasenwechselvorgänge in makroverkapselten latenten thermischen Speichern*, Logos Verlag Berlin GmbH. Google-Books-ID: OGjTBgAAQBAJ.
- Saeed, R. M., Schlegel, J. P., Sawafta, R. & Kalra, V. (2019), 'Plate type heat exchanger for thermal energy storage and load shifting using phase change material', *Energy Conversion and Management* **181**, 120–132.
URL: <https://www.sciencedirect.com/science/article/pii/S0196890418313475>
- Salunkhe, P. B. & Shembekar, P. S. (2012), 'A review on effect of phase change material encapsulation on the thermal performance of a system', *Renewable and Sustainable Energy Reviews* **16**(8), 5603–5616.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032112003711>
- Santos, T., Wrobel, L. C., Hopper, N. & Kolokotroni, M. (2021), 'Numerical Design and Laboratory Testing of Encapsulated PCM Panels for PCM-Air Heat Exchangers', *Applied Sciences* **11**(2), 676. Number: 2 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/2076-3417/11/2/676>
- Sciacovelli, A., Gagliardi, F. & Verda, V. (2015), 'Maximization of performance of a PCM latent heat storage system with innovative fins', *Applied Energy* **137**, 707–715.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261914006825>
- Seddegh, S., Wang, X. & Henderson, A. D. (2016), 'A comparative study of thermal behaviour of a horizontal and vertical shell-and-tube energy storage using phase change materials', *Applied Thermal Engineering* **93**, 348–358.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431115010340>
- Seddegh, S., Wang, X., Joybari, M. M. & Haghighat, F. (2017), 'Investigation of the effect of geometric and operating parameters on thermal behavior of vertical shell-and-tube latent heat energy storage systems', *Energy* **137**, 69–82.
URL: <https://www.sciencedirect.com/science/article/pii/S036054421731188X>
- Shah, R. K. (2003), *Fundamentals of Heat Exchanger Design*, Wiley Interscience Imprint.
- Shah, R. K. (2006), 'Advances in Science and Technology of Compact Heat Exchangers', *Heat Transfer Engineering* **27**(5), 3–22. Publisher: Taylor & Francis
_eprint: <https://doi.org/10.1080/01457630600559462>.
URL: <https://doi.org/10.1080/01457630600559462>
- Shahsavari, A., Majidzadeh, A. H., Mahani, R. B. & Talebizadehsardari, P. (2021), 'Entropy and thermal performance analysis of PCM melting and solidification mechanisms in a wavy channel triplex-tube heat exchanger', *Renewable Energy* **165**, 52–72.
URL: <https://www.sciencedirect.com/science/article/pii/S0960148120318231>
- Sharma, A., Tyagi, V. V., Chen, C. R. & Buddhi, D. (2009), 'Review on thermal

- energy storage with phase change materials and applications', *Renewable and Sustainable Energy Reviews* **13**(2), 318–345.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032107001402>
- Sheik Ismail, L., Velraj, R. & Ranganayakulu, C. (2010), 'Studies on pumping power in terms of pressure drop and heat transfer characteristics of compact plate-fin heat exchangersA review', *Renewable and Sustainable Energy Reviews* **14**(1), 478–485.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032109001543>
- Singh, S. K., Mishra, M. & Jha, P. K. (2014), 'Nonuniformities in compact heat exchangersscope for better energy utilization: A review', *Renewable and Sustainable Energy Reviews* **40**, 583–596.
URL: <https://www.sciencedirect.com/science/article/pii/S1364032114006674>
- Sinnett, M. (2007), 'Saving Fuel and Enhancing Operational Efficiencies', *Aero* **7**(4), 6–11.
URL: http://www.boeing.ch/commercial/aeromagazine/articles/qtr_4_07/AERO_Q407_a
- Socaciu, L., Giurgiu, O., Banyai, D. & Simion, M. (2016), 'PCM Selection Using AHP Method to Maintain Thermal Comfort of the Vehicle Occupants', *Energy Procedia* **85**, 489–497.
URL: <https://www.sciencedirect.com/science/article/pii/S1876610215028970>
- Srinath, A. N., Pena López, ., Miran Fashandi, S. A., Lechat, S., di Legge, G., Nabavi, S. A., Nikolaidis, T. & Jafari, S. (2022), 'Thermal Management System Architecture for Hydrogen-Powered Propulsion Technologies: Practices, Thematic Clusters, System Architectures, Future Challenges, and Opportunities', *Energies* **15**(1), 304. Number: 1 Publisher: Multidisciplinary Digital Publishing Institute.
URL: <https://www.mdpi.com/1996-1073/15/1/304>
- Stefan, J. & Akad, M. (1889), *Ueber einige Probleme der Theorie der Waer-meileitung*, Vienna (Austria).
- Talebizadehsardari, P., Mahdi, J. M., Mohammed, H. I., Moghimi, M. A., Hossein Eisapour, A. & Ghalambaz, M. (2021), 'Consecutive charging and discharging of a PCM-based plate heat exchanger with zigzag configuration', *Applied Thermal Engineering* **193**, 116970.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431121004178>
- Tay, N. H. S., Belusko, M. & Bruno, F. (2012), 'An effectiveness-NTU technique for characterising tube-in-tank phase change thermal energy storage systems', *Applied Energy* **91**(1), 309–319.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261911006441>
- Tay, N. H. S., Belusko, M., Castell, A., Cabeza, L. F. & Bruno, F. (2014), 'An effectiveness-NTU technique for characterising a finned tubes PCM system using a CFD model', *Applied Energy* **131**, 377–385.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261914006230>

- Tian, S., Bai, H., Hui, N., Gao, H., Shao, S., Sun, Z. & Liu, S. (2021), 'A dynamic analytical model to predict and parametrically analyze charging and discharging process of tube-in-tank PCM systems', *Thermal Science and Engineering Progress* **22**, 100866.
URL: <https://www.sciencedirect.com/science/article/pii/S2451904921000287>
- van Heerden, A. S. J., Judt, D. M., Jafari, S., Lawson, C. P., Nikolaidis, T. & Bosak, D. (2022), 'Aircraft thermal management: Practices, technology, system architectures, future challenges, and opportunities', *Progress in Aerospace Sciences* **128**, 100767.
URL: <https://www.sciencedirect.com/science/article/pii/S0376042121000701>
- Vargas, J. V. C. & Bejan, A. (2001), 'Integrative thermodynamic optimization of the environmental control system of an aircraft', *International Journal of Heat and Mass Transfer* **44**(20), 3907–3917.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931001000333>
- Voller, V. R., Cross, M. & Markatos, N. C. (1987), 'An enthalpy method for convection/diffusion phase change', *International Journal for Numerical Methods in Engineering* **24**(1), 271–284. _eprint: <https://onlinelibrary.wiley.com/doi/pdf/10.1002/nme.1620240119>.
URL: <https://onlinelibrary.wiley.com/doi/abs/10.1002/nme.1620240119>
- Voller, V. R. & Prakash, C. (1987), 'A fixed grid numerical modelling methodology for convection-diffusion mushy region phase-change problems', *International Journal of Heat and Mass Transfer* **30**(8), 1709–1719.
URL: <https://www.sciencedirect.com/science/article/pii/0017931087903176>
- Voller, V. R., Swaminathan, C. R. & Thomas, B. G. (1990), 'Fixed grid techniques for phase change problems: A review', *International Journal for Numerical Methods in Engineering* **30**(4), 875–898. _eprint: <https://onlinelibrary.wiley.com/doi/pdf/10.1002/nme.1620300419>.
URL: <https://onlinelibrary.wiley.com/doi/abs/10.1002/nme.1620300419>
- Wang, Y.-Q., Dong, Q. W., Liu, M. S. & Wang, D. (2009), 'Numerical Study on Plate-Fin Heat Exchangers with Plain Fins and Serrated Fins at Low Reynolds Number', *Chemical Engineering & Technology* **32**(8), 1219–1226. _eprint: <https://onlinelibrary.wiley.com/doi/pdf/10.1002/ceat.200900079>.
URL: <https://onlinelibrary.wiley.com/doi/abs/10.1002/ceat.200900079>
- Wen, J., Li, Y., Zhou, A. & Zhang, K. (2006), 'An experimental and numerical investigation of flow patterns in the entrance of plate-fin heat exchanger', *International Journal of Heat and Mass Transfer* **49**(9), 1667–1678.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931005006587>
- Wilcox, D. C. (1988), 'Reassessment of the scale-determining equation for advanced turbulence models', *AIAA Journal* **26**(11), 1299–1310.
URL: <https://doi.org/10.2514/3.10041>
- Xie, J., Wang, W., Liu, J. & Pan, S. (2018), 'Thermal performance analysis of PCM

- wallboards for building application based on numerical simulation', *Solar Energy* **162**, 533–540.
URL: <https://www.sciencedirect.com/science/article/pii/S0038092X18300902>
- Xu, B., Li, P. & Chan, C. (2015), 'Application of phase change materials for thermal energy storage in concentrated solar thermal power plants: A review to recent developments', *Applied Energy* **160**, 286–307.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261915010867>
- Xu, H., Wang, Y. & Han, X. (2020), 'Analytical considerations of thermal storage and interface evolution of a PCM with/without porous media', *INTERNATIONAL JOURNAL OF NUMERICAL METHODS FOR HEAT & FLUID FLOW* **30**(1), 373–400.
- Xu, Y., Ren, Q., Zheng, Z.-J. & He, Y.-L. (2017), 'Evaluation and optimization of melting performance for a latent heat thermal energy storage unit partially filled with porous media', *Applied Energy* **193**, 84–95.
URL: <https://www.sciencedirect.com/science/article/pii/S0306261917301435>
- Yang, B., Raza, A., Bai, F., Zhang, T. & Wang, Z. (2019), 'Microstructural evolution within mushy zone during paraffins melting and solidification', *International Journal of Heat and Mass Transfer* **141**, 769–778.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931019321921>
- Yuan, J., Yang, K., Huang, B., Li, J., Qiu, C. & Jiang, Y. (2024), 'Transient thermal management of laser systems using Plate-Fin phase change heat Exchangers: Experimental and computational study', *Applied Thermal Engineering* **255**, 123994.
URL: <https://www.sciencedirect.com/science/article/pii/S1359431124016624>
- Yuan, X., Zhou, X., Pan, Y., Kosonen, R., Cai, H., Gao, Y. & Wang, Y. (2021), 'Phase change cooling in data centers: A review', *Energy and Buildings* **236**, 110764.
URL: <https://www.sciencedirect.com/science/article/pii/S0378778821000487>
- Zhang, J., Kundu, J. & Manglik, R. M. (2004), 'Effect of fin waviness and spacing on the lateral vortex structure and laminar heat transfer in wavy-plate-fin cores', *International Journal of Heat and Mass Transfer* **47**(8), 1719–1730.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931003005891>
- Zhang, Z. & Li, Y. (2003), 'CFD simulation on inlet configuration of plate-fin heat exchangers', *Cryogenics* **43**(12), 673–678.
URL: <https://www.sciencedirect.com/science/article/pii/S0011227503001796>
- Zheng, Z., Fletcher, D. F. & Haynes, B. S. (2014), 'Transient laminar heat transfer simulations in periodic zigzag channels', *International Journal of Heat and Mass Transfer* **71**, 758–768.
URL: <https://www.sciencedirect.com/science/article/pii/S0017931013011010>
- Zonouzi, S. A. & Dadvar, A. (2022), 'Numerical investigation of using helical fins for the enhancement of the charging process of a latent heat thermal energy storage

system', *Journal of Energy Storage* **49**, 104157.

URL: <https://www.sciencedirect.com/science/article/pii/S2352152X22001918>