

Development and investigation of heat extraction concepts for metallic phase change thermal energy storage in mobile applications

Frank Nees

Deutsches Zentrum für Luft- und Raumfahrt
Institut für Fahrzeugkonzepte
Stuttgart



DLR

Deutsches Zentrum
für Luft- und Raumfahrt

Forschungsbericht 2026-16

Development and investigation of heat extraction concepts for metallic phase change thermal energy storage in mobile applications

Frank Nees

Deutsches Zentrum für Luft- und Raumfahrt
Institut für Fahrzeugkonzepte
Stuttgart

206 Seiten
96 Bilder
26 Tabellen
129 Literaturstellen



Deutsches Zentrum
DLR für Luft- und Raumfahrt



Herausgeber:

Deutsches Zentrum
für Luft- und Raumfahrt e. V.
Wissenschaftliche Information
Linder Höhe
D-51147 Köln

ISSN 1434-8454
ISRN DLR-FB-2026-16
Erscheinungsjahr 2026
DOI: [10.57676/6p4t-q518](https://doi.org/10.57676/6p4t-q518)

Erklärung des Herausgebers

Dieses Werk – ausgenommen anderweitig gekennzeichnete Teile – ist lizenziert unter den Bedingungen der Creative Commons Lizenz vom Typ Namensnennung 4.0 International (CC BY 4.0), abrufbar über <https://creativecommons.org/licenses/by/4.0/legalcode>

Lizenz



Creative Commons Attribution 4.0 International

Thermische Energiespeicher, Metallisches Phasenwechselmaterial, Wärmeaustrag, Thermischer Kontaktwiderstand, Siedekrise, Elektrische Fahrzeugsysteme

Frank NEES
DLR, Institut für Fahrzeugkonzepte, Stuttgart

Entwicklung und Untersuchung von Wärmeaustragskonzepten für metallische Latentwärmespeicher in mobilen Anwendungen
Universität Stuttgart

Die Speicherung von thermischer Energie mithilfe metallischer Phasenwechselmaterialien ist eine vielversprechende Technologie zur Verbesserung des Thermomanagements von Elektrofahrzeugen in kalten Umgebungen. Nach dem Stand der Technik und Forschung ist der Einsatz eines solchen Speichersystems in Fahrzeugen jedoch Neuland. Im Hinblick auf die thermische Entladung stellt der hohe Betriebstemperaturbereich des Speichersystems eine Herausforderung dar. Um diese Forschungslücke zu schließen, befasst sich die vorliegende Dissertation mit der Entwicklung und Untersuchung von Wärmeaustragskonzepten für metallische Latentwärmespeicher in mobilen Anwendungen. Auf der Grundlage einer systematischen Konzeptentwicklung wurden zwei vielversprechende Konzepte – ein direktes Luft- und ein neuartiges, mehrstufiges Verdampfungskonzept – für die weitere Funktionsmusterentwicklung, Simulation und experimentelle Untersuchung ausgewählt. Für beide Konzepte wurde ein Funktionsnachweis erreicht, der die Funktionalität, Leistungsfähigkeit und Effizienz des Wärmeaustrags demonstrierte. Die Ergebnisse erlauben eine erstmalige Charakterisierung der grundlegenden physikalischen Effekte der Entladung und zeigen wichtige technische Designaspekte auf.

Thermal energy storage, Metallic phase change material, Heat extraction, Thermal contact resistance, Boiling crisis, Electric vehicles

(Published in English)

Frank NEES
German Aerospace Center (DLR), Institute of Vehicle Concepts, Stuttgart

Development and investigation of heat extraction concepts for metallic phase change thermal energy storage in mobile applications

University of Stuttgart

Storing thermal energy using metallic phase change materials is a technology that shows promise for enhancing the thermal management of electric vehicles in cold environments. However, according to the current state of technology and research, the use of such a storage system in vehicles is uncharted territory. In terms of thermal discharge, the high operating temperature range of such systems poses challenges. To address this research gap, this thesis focuses on the development and investigation of heat extraction concepts for metallic phase change thermal energy storage in mobile applications. On the basis of conceptual development, two main concepts, a direct air and a novel multi-stage boiling discharge concept, were selected for further prototyping, simulation, and experimental investigation. A proof of concept was achieved for both concepts, demonstrating the functionality, performance, and efficiency of the heat extraction process. For the first time, fundamental physical effects of the discharge were characterized and important technical design aspects were revealed.

Development and investigation of heat extraction concepts for metallic phase change thermal energy storage in mobile applications

Von der Fakultät Konstruktions-, Produktions- und Fahrzeugtechnik
der Universität Stuttgart
zur Erlangung der Würde eines Doktor-Ingenieurs (Dr.-Ing.)
genehmigte Abhandlung

von
Dipl.-Ing. Frank Nees
aus Mainz

Hauptberichter: Prof. Dr.-Ing. Tjark Siefkes
Mitberichter: Prof. Dr.-Ing. Andreas Wagner

Tag der mündlichen Prüfung: 29.04.2026

Institut für Fahrzeugtechnik, Universität Stuttgart

Angefertigt am Institut für Fahrzeugkonzepte,
Deutsches Zentrum für Luft- und Raumfahrt (DLR) e.V. Stuttgart

Abstract

Storing thermal energy using metallic phase change materials is a technology that shows promise for enhancing the thermal management of electric vehicles in cold environments. These materials are energy-dense, highly thermally conductive, and easily available, making them ideal candidates for storing heat in a compact, lightweight, and cost-effective manner. However, according to the current state of technology and research, the use of such a storage system in vehicles is uncharted territory. In terms of thermal discharge, the high operating temperature range of such systems poses challenges, especially when transferring heat to the vehicle's coolant circuit.

In order to address this research gap, this thesis focuses on the development and investigation of heat extraction concepts for thermal energy storage based on metallic phase change materials in mobile applications. The objectives are therefore, on the one hand, to advance the technology and, on the other hand, to identify important physical effects and characteristics of the thermal discharge. This thesis considers the aluminum-silicon alloy AlSi_{12} , which has a phase change temperature of 577°C , as storage material. On the basis of conceptual development, two main concepts were selected for further prototyping, simulation, and experimental investigation. The first selected concept applies direct air discharge to the cabin and is considered to be the simplest approach. In addition to the experimental investigation of a bench- and application-scale prototype, a conjugate computational fluid dynamics simulation was performed to study the transient thermal discharge including the phase change of the storage material. The second concept developed is more sophisticated and is based on a novel multi-stage boiling approach. It addresses a practical engineering challenge, which is known as the boiling crisis, and has significant implications for energy efficiency and real-world applications. Heat extraction is achieved through a closed-loop two-stage boiling and condensing cycle using water as the working fluid, allowing heat to be transferred to the vehicle's cooling circuit.

The experimental investigation of the direct air discharge concept demonstrated the functionality, performance, and efficiency of the heat extraction process across an operating temperature range of $650\text{--}100^\circ\text{C}$. A proof of concept was achieved. The developed heat transfer calculation model and the conjugate numerical simulation showed good agreement with the experimental data, indicating the successful validation of the models. Based on the experiments and simulations, a strong effect of the phase change on the thermal contact resistance at the AlSi_{12} interface was observed. The contact resistance is lowest in the liquid state of the AlSi_{12} , increases significantly during the phase change, and continues to increase with decreasing temperature. The temperature-dependent values of the thermal contact resistance were determined.

Experiments with the novel multi-stage boiling discharge concept were conducted successfully across an operating temperature range of 650–100 °C and a proof of concept was achieved. High specific heat discharge rates were observed at high, medium, and low storage temperatures, with peak values exceeding 20 W/cm². The results demonstrate that the system effectively utilizes both latent and sensible heat of the storage material, resulting in high efficiency and low pumping power requirements. Under pressure-tight conditions, the absolute pressure in the discharge system remained below 1.4 bar. Compared to the direct air concept, the multi-stage boiling concept offers benefits at the vehicle system level, including reduced system volume and weight, and greater flexibility with regard to the vehicle heat distribution system.

The concepts developed and investigated in this thesis overcome the challenges of transferring high-temperature storage heat to a vehicle system. The results provide the first characterization of the complex physics of the heat extraction process. The novel concepts proposed in this research, along with the new physical insights gained, extend the state of the art and provide a basis for further development in this field.

Keywords: *Thermal energy storage, Metallic phase change material, Heat extraction, Thermal contact resistance, Boiling crisis, Electric vehicles*

Kurzfassung

Die Speicherung von thermischer Energie in metallischen Phasenwechselmaterialien ist eine vielversprechende Technologie, um das Thermomanagement von Elektrofahrzeugen in kalten Umgebungen zu verbessern. Aufgrund ihrer hohen Energiedichte und Wärmeleitfähigkeit eignen sich diese Materialien sehr gut für eine kompakte, leichte und kostengünstige Wärmespeicherung. Nach dem Stand der Technik und Forschung ist der Einsatz eines solchen Speichersystems in Fahrzeugen jedoch Neuland. Im Hinblick auf die thermische Entladung stellt der hohe Betriebstemperaturbereich des Speichersystems eine Herausforderung dar, insbesondere, wenn die Wärme an einen Flüssigkeitskreislauf des Fahrzeugs übertragen werden soll.

Um diese Forschungslücke zu schließen, befasst sich die vorliegende Dissertation mit der Entwicklung und Untersuchung von Wärmeaustragskonzepten für thermische Energiespeicher auf Basis metallischer Phasenwechselmaterialien in mobilen Anwendungen. Die Zielsetzung besteht zum einen in der Weiterentwicklung der Technologie und zum anderen in der Identifikation wichtiger physikalischer Effekte und Eigenschaften der thermischen Entladung. In dieser Arbeit wird die Aluminium-Silizium-Legierung AlSi_{12} als Speicher-material betrachtet. Diese weist eine Phasenwechseltemperatur von $577\text{ }^{\circ}\text{C}$ auf. Auf Basis einer systematischen Konzeptentwicklung wurden zwei vielversprechende Konzepte für eine weitere Funktionsmusterentwicklung, detaillierte Simulationen und experimentelle Untersuchung ausgewählt. Das erste ausgewählte Konzept nutzt die direkte Luftentladung und ist der am einfachsten umzusetzende Ansatz. Neben der experimentellen Untersuchung eines Funktionsmusters im Labor- und Anwendungsmaßstab wurden konjugierte numerische Strömungssimulationen durchgeführt, um die transiente Entladung einschließlich des Phasenwechsels des Speichermaterials zu analysieren. Das zweite entwickelte Konzept ist komplexer und basiert auf einem neuartigen, mehrstufigen Verdampfungsprinzip. Dieses adressiert ein zentrales technisches Problem, das Auftreten der Siedekrise, und hat erhebliche Auswirkungen auf die Leistungsfähigkeit realer Anwendungen. Die thermische Entladung erfolgt hierbei über einen geschlossenen zweistufigen Verdampfungs- und Kondensationskreislauf mit Wasser als Arbeitsmedium. Auf diese Weise kann die Wärme an den Wasser-Glykol-Kreislauf des Fahrzeugs übertragen werden.

Die experimentelle Untersuchung des Konzepts der direkten Luftentladung hat erstmals die Funktionalität, Leistungsfähigkeit und Effizienz der thermischen Entladung in einem Temperaturbereich von $650\text{--}100\text{ }^{\circ}\text{C}$ aufgezeigt und einen Funktionsnachweis erbracht. Das entwickelte Berechnungsmodell und die Ergebnisse der konjugierten numerischen Simulationen zeigten eine gute Übereinstimmung mit den experimentellen Daten, wodurch eine Validierung der Modelle erreicht wurde. Auf Basis der durchgeführten Experimente und

Simulationen ist ein großer Einfluss des Phasenwechsels auf den thermischen Kontaktwiderstand an der AlSi_{12} -Grenzfläche identifiziert worden. Dieser ist im flüssigen Zustand des AlSi_{12} am niedrigsten, steigt während des Phasenwechsels deutlich an und nimmt mit abnehmender Temperatur weiter zu. Die zugehörigen temperaturabhängigen Werte des thermischen Kontaktwiderstands wurden erstmals im Rahmen dieser Arbeit bestimmt.

Das mehrstufige Verdampfungskonzept wurde in einem Temperaturbereich von 650–100 °C experimentell untersucht und ein Funktionsnachweis erreicht. Es zeigten sich hohe spezifische Wärmeübertragungsraten bei hohen, mittleren und niedrigen Speichertemperaturen, mit Werten von bis zu über 20 W/cm². Die Ergebnisse zeigen, dass sowohl die latente als auch die sensible Wärme des Speichermaterials effektiv genutzt wird, bei gleichzeitig geringem Leistungsbedarf zur Förderung des Arbeitsmediums. Unter druckdichten Bedingungen blieb der gemessene absolute Druck im Entladesystem unter 1,4 bar. Im Vergleich zur direkten Luftentladung bietet das mehrstufige Verdampfungskonzept Vorteile auf Fahrzeugsystemebene, da dieses ein geringeres Systemvolumen und -gewicht aufweist und flexibler hinsichtlich der Einspeisung der Wärme ins Fahrzeug ist.

Die im Rahmen dieser Arbeit entwickelten und untersuchten Konzepte ermöglichen die Übertragung der in metallischen Phasenwechselmaterialien gespeicherten Hochtemperaturwärme in ein Fahrzeugsystem. Die Ergebnisse erlauben erstmals eine Charakterisierung der komplexen Physik des Wärmeextraktionsprozesses. Die neuartigen technischen Konzepte und die physikalischen Erkenntnisse, die im Zuge dieser Dissertation gewonnen wurden, erweitern den Stand der Technik und bilden die Grundlage für zukünftige Entwicklungen im Themenfeld.

Stichwörter: *Thermische Energiespeicher, Metallisches Phasenwechselmaterial, Wärmeaustrag, Thermischer Kontaktwiderstand, Siedekrise, Elektrische Fahrzeugsysteme*

Acknowledgments

This thesis was conducted as part of my work as a research associate at the Institute of Vehicle Concepts at the German Aerospace Center (DLR) in Stuttgart. First and foremost, I would like to thank my doctoral adviser, Professor Dr. Tjark Siefkes, Director of the DLR Institute of Vehicle Concepts and Professor at the University of Stuttgart, for giving me the opportunity to conduct this research at his institute and for his invaluable guidance and support throughout this journey.

I also want to thank Professor Dr. Andreas Wagner from the University of Stuttgart for his valuable contributions as co-referee of my thesis. Another thank goes to Professor Dr. Peter Pott from the University of Stuttgart for being a member of my dissertation committee.

Furthermore, I want to express my gratitude to Dr. Frank Rinderknecht, the head of my department, for enabling me to actively shape my own research and for the scientific freedom to work on my thesis project. Moreover, I want to thank all my colleagues at DLR, especially my research group leader, Werner Kraft, and my team members, Veronika Stahl, Peter Vetter, and Henrik Gröbbl, for their support in the lab and discussions provided.

I appreciate the great work of my students, Yogesh Sajikumar Pai, Lukas Ignatzi, Anastasios Katourtzidis, Frederik von Klitzing, and Julius Feine, as well as the good cooperation we had.

I am deeply grateful for my parents, Birgit and Gero, and their continuous support throughout my life. They made my career possible, and this work would never have been possible without them. Finally, I would like to thank Bettina for her unwavering love, patience, and loyalty over the years.

Contents

Abstract	i
Kurzfassung	iii
Acknowledgments	v
Contents	vii
List of Figures	xi
List of Tables	xvii
Nomenclature	xix
1 Introduction	1
1.1 Background and motivation	1
1.2 Problem definition and objectives	3
1.3 Methodological approach and chapter overview	5
2 Theoretical basis and state of the art	9
2.1 Thermal management in vehicles	9
2.2 Thermal energy storage technologies	10
2.2.1 Physical principle of thermal energy storage	10
2.2.2 Metallic phase change materials and AlSi ₁₂ as storage material	12
2.2.3 Thermal energy storage systems in stationary and mobile applications	15
2.2.4 Thermal discharge of high-temperature heat storages in literature	16
2.3 Physical principles of heat transfer	20
2.4 Heat transfer and pressure drop in single-phase flow	24
2.4.1 Convective heat transfer in internal ducts	24
2.4.2 Fins for heat transfer enhancement	25
2.4.3 Pressure drop in internal ducts and pumping power	27
2.5 Boiling and condensation	30
2.5.1 Boiling modes for pool and flow boiling	30
2.5.2 Heat transfer correlations for flow boiling	32
2.5.3 Critical heat flux and boiling crisis	34
2.5.4 Heat transfer in film boiling, dryout, and transition boiling regimes	36
2.5.5 Condensation	38
2.6 Thermal resistance and thermal contact resistance	39
2.7 Heat transfer fluids and current technical limitations	41

2.8	Conclusions from the state of the art	43
3	Development of concepts for thermal discharge	45
3.1	Storage system, boundary conditions, and requirements	45
3.2	Impact factors on discharge performance	48
3.3	Conceptual approach	50
3.4	Derivation of heat discharge concepts	52
3.5	Evaluation of concepts and concept selection	60
4	Direct air discharge – Development and investigation	63
4.1	Design philosophy and development	63
4.1.1	Design philosophy	63
4.1.2	Heat transfer calculation model	65
4.1.3	Parameter study and discussion	68
4.1.4	Prototype channel geometry	72
4.2	Experimental investigation of bench-scale prototype	73
4.2.1	Prototype design and test bench	73
4.2.2	Experimental procedure	77
4.2.3	Measurement uncertainty	78
4.2.4	Results and discussion	79
4.2.4.1	Constant fan power rate	79
4.2.4.2	Constant mass flow rate	80
4.2.4.3	Constant heat flow rate	81
4.2.4.4	Efficiency of heat extraction	83
4.3	Conjugate heat transfer analysis of bench-scale prototype	84
4.3.1	Model description	84
4.3.2	Results and discussion	88
4.4	Experimental investigation of application-scale prototype	92
4.4.1	Prototype design and test bench	92
4.4.2	Experimental procedure	95
4.4.3	Measurement uncertainty	96
4.4.4	Results and discussion	97
4.4.4.1	Constant mass flow rate	97
4.4.4.2	Design heat flow rate	98
4.4.4.3	Efficiency of heat extraction	99
5	Multi-stage boiling discharge – Development and investigation	101
5.1	Design philosophy and simulation	101
5.1.1	Multi-stage design philosophy	101
5.1.2	Simulation of 2D-temperature profile	104
5.2	Functional prototype and experimental methods	106
5.2.1	Experimental setup	106
5.2.2	Prototype design and construction	108
5.2.3	Experimental procedure	112

5.2.4	Measurement uncertainty	114
5.3	Experimental results and discussion	115
5.3.1	Thermal discharge characteristics of Stage 1	115
5.3.2	Thermal discharge characteristics of Stage 2	117
5.3.3	Evaluation of thermal discharge performance	120
5.3.4	Efficiency of heat extraction	121
5.3.5	Time-dependent temperature pattern	122
5.3.6	Closed-loop system pressure characteristics	123
5.4	Derivation of heat transfer calculation model	124
5.4.1	Development of a physical heat transfer model	125
5.4.2	Adjustment of the model and comparison with experimental data	129
5.4.2.1	Stage 1 model	130
5.4.2.2	Stage 2 model	132
5.4.3	Coupled operation of Stages 1 and 2	134
5.4.4	Upscaling of results to application scale	135
6	Comparison of concepts and evaluation at vehicle system level	139
6.1	Discharge level and storage utilization	139
6.2	Volume, weight, and electrical power requirements	143
6.2.1	Definition of scenarios and methodology	143
6.2.2	Results and discussion	147
6.3	Evaluation of results	150
7	Conclusions and outlook	151
7.1	Conclusions	151
7.2	Outlook	153
Appendix		155
A	Relevant fluid and material properties	155
B	Supplemental information on measurement uncertainties	158
C	Grid convergence study for numerical simulations	162
References		165
List of publications, patents, and awards		177

List of Figures

1.1	Operating energy density ranges for different thermal storage media and a typical range for batteries used in electric vehicles, reproduced from [88] . .	2
1.2	Temperatures of mPCM and cabin air versus operating thermal energy for AlSi ₁₂ storage material	4
1.3	Methodological approach of this thesis	6
2.1	Sensible and latent heat versus temperature and stored thermal energy . .	12
2.2	Gravimetric and volumetric energy densities versus melting point, adapted from [88]	14
2.3	Solid-medium TES concept for heat supply in electric vehicles and ceramic honeycombs used as storage material	17
2.4	Exemplary mPCM thermal storage discharge concepts from literature . . .	19
2.5	mPCM thermal storage discharge concepts from DLR Institute of Vehicle Concepts	19
2.6	Overview about different fin types, adapted from [129]	26
2.7	Characteristic boiling curve for pool and flow boiling, adapted from [70] . .	31
2.8	Schematic of flow regimes for flow boiling in horizontal tube and nucleate boiling dominant heat transfer, reproduced from [49]	33
2.9	Boiling crisis mechanisms. (a) Film boiling (DNB). (b) Dryout (Mist flow). Reproduced from [34] with permission from Springer Nature.	35
2.10	Critical heat flux versus vapor mass fraction for flow boiling of water . . .	36
2.11	Temperature drop due to thermal contact resistance, reproduced from [39] with permission from John Wiley & Sons	40
2.12	Operating temperature range of different high-temperature HTFs, adapted from [118]	42
3.1	Principle geometric arrangements of discharge unit within the mPCM . . .	46
3.2	Schematic of the basic storage system arrangement considered as a boundary condition in this thesis. Own illustration derived from [57, 60].	46
3.3	Exemplary temperature profile and thermal resistances for heat transfer from mPCM to the discharge fluid	49
3.4	Basic conceptual approach for thermal discharge, adapted from Nees et al. [77]	51
3.5	Schematic representation of the direct air discharge concept (Concept 1) .	52
3.6	Schematic representation of the compressed gas cycle concept (Concept 2), showing the components and the arrangement of the fluid channels	53

3.7	Schematic representation of the multi-stage concept (Concept 3) showing the components and the arrangement of the evaporator Stage 1 and Stage 2, reproduced from Nees et al. [80].	55
3.8	Schematic representation of start-up and shut-down process of Concept 3, reproduced from Nees et al. [80].	56
3.9	Schematic representation of the hybrid concept (Concept 6), showing the components and the arrangement of the evaporator and gas channels . . .	58
3.10	Schematic representation of start-up and shut-down process of Concept 6 .	59
4.1	Arrangement of the air heat transfer section and geometric details, based on the geometric boundary conditions discussed in Section 3.1	64
4.2	Schematic representation of the air heat transfer modeling approach	66
4.3	Lower limit temperature for $T_{w,AirHEX}$ versus normalized volume of the air heat transfer section in dependence of different parameters	70
4.4	Impact of material selection and fin thickness on lower limit temperature and channel height	71
4.5	Channel geometry for the bench-scale prototype (plain fin)	72
4.6	Channel geometry for the application-scale prototype (lazy ruffled wavy fin) .	73
4.7	Schematic representation of the bench-scale storage prototype and experimental setup, adapted from Nees et al. [79]	74
4.8	Test bench and prototype, adapted from Nees et al. [79]	75
4.9	Graphite container and storage material, adapted from Nees et al. [79] . .	75
4.10	Plate prototype for heating and heat extraction with integrated plain fin geometry, adapted from Nees et al. [79]	76
4.11	Experimental procedure and exemplary charging and discharging curve, adapted from Nees et al. [79]	77
4.12	Heat extraction characteristics for constant fan power rate, adapted from Nees et al. [79]	80
4.13	Heat extraction characteristics for constant mass flow rate, adapted from Nees et al. [79]	81
4.14	Heat extraction characteristics for constant heat output, adapted from Nees et al. [79]	82
4.15	Ratio $P_{el} / \dot{Q}_{ex}$ versus mPCM temperature for different fan power rates, reproduced from Nees et al. [79]	83
4.16	Three-dimensional numerical model with selected boundary conditions, reproduced from Nees et al. [76]	85
4.17	Visualizations of the mesh from a perspective view and a detailed view . .	85
4.18	Comparison of simulation results for temperature and heat flow with experimental data, adapted from Nees et al. [76]	88
4.19	Liquid mass fraction of mPCM cross-section at various stages of solidification quantifying the discharge of latent heat, reproduced from Nees et al. [76]	90
4.20	Temperature distribution at cross-section after $t = 1011$ s thermal discharge, reproduced from Nees et al. [76]	91

4.21	Temperature and velocity contours at channel outlet in the air heat transfer section at various time steps	91
4.22	Schematic representation of the application-scale prototype and experimental setup	93
4.23	Test bench with application-scale prototype, adapted from [87]	93
4.24	Base plate with connecting parts and air inlet line, adapted from [87]	94
4.25	Air heat transfer geometry used in prototype before and after brazing	94
4.26	Experimental heat flow characteristics for a constant air mass flow rate of 125 kg/h and comparison with calculation model	98
4.27	Experimental heat flow characteristics for design case of 10 kW and comparison with calculation model	99
4.28	Ratio $P_{el} / \dot{Q}_{ex}$ against temperature for design heat flow rate	100
5.1	Schematic representation of Stage 1 and Stage 2 channel concepts	102
5.2	Critical heat flux of water at 1 bar versus vapor mass fraction for various flow rates and an inner channel diameter of 3 mm	102
5.3	Resulting heat flux across air gap and inner tube wall for various gap widths versus temperature	103
5.4	Numerical model with selected boundary conditions	104
5.5	Temperature profile for coupled operation of Stage 1 and Stage 2 for high and low temperature regime	105
5.6	Schematic representation of the prototype test bench and experimental setup, reproduced from Nees et al. [80]	107
5.7	Simplified fluid and heat flow diagram of the experimental setup, reproduced from Nees et al. [80]	107
5.8	Test bench and prototype setup shown from different perspectives, illustrating the storage system and components within the test bench, reproduced from Nees et al. [80]	109
5.9	Base plate arrangement, reproduced from Nees et al. [80]	110
5.10	Storage material and graphite encapsulation, reproduced from Nees et al. [80]	111
5.11	Insulated prototype from different perspectives, reproduced from Nees et al. [80]	111
5.12	Experimental procedure and exemplary charging and discharging curve, reproduced from Nees et al. [80]	113
5.13	Heat discharge characteristics of Stage 1 (for 10 l/h, 64 kg/(m ² s)), reproduced from Nees et al. [80]	116
5.14	Heat discharge characteristics of Stage 1 for various flow rates, reproduced from Nees et al. [80]	117
5.15	Heat discharge characteristics of Stage 2 (for 10 l/h, 64 kg/(m ² s)), reproduced from Nees et al. [80]	118
5.16	Heat discharge characteristics of Stage 2 for various flow rates, reproduced from Nees et al. [80]	119

5.17	Heat flow versus state of charge (SoC) of mPCM storage material for Stages 1 and 2 (for 10 l/h, 64 kg/(m ² s)), reproduced from Nees et al. [80]	120
5.18	Ratio $P_{el}/\dot{Q}_{ex}$ against temperature (for 10 l/h, 64 kg/(m ² s)), reproduced from Nees et al. [80]	121
5.19	Time-dependent temperature pattern in base plate and mPCM for Stages 1 and 2, reproduced from Nees et al. [80]	123
5.20	Pressure and heat flow versus discharge time in pressure-tight condition (10 l/h, 64 kg/(m ² s)), reproduced from Nees et al. [80]	124
5.21	Schematic representation of heat transfer from base plate to fluid bulk with corresponding temperatures	125
5.22	Schematic representation of the modeling approach	126
5.23	Model criteria for selection of heat transfer regime	127
5.24	Definition of model parameters (modeled boiling curve)	129
5.25	Comparison of heat flow results of adapted and unadapted model with experimental data for Stage 1	130
5.26	Comparison of heat flow results of adapted and unadapted model with experimental data for Stage 2	132
5.27	Results for coupled operation of Stages 1 and 2, based on the adapted model (15 l/h corresponds to 97 kg/(m ² s))	134
5.28	Results for coupled operation of Stages 1 and 2, based on the upscaling of the adapted model results to the application scale (150 l/h corresponds to 97 kg/(m ² s))	137
6.1	Comparison of heat flow versus temperature for multi-stage boiling and direct air discharge concepts based on application-scale storage size and developed calculation models	140
6.2	Lower limit temperature versus heat flow rate for multi-stage boiling and direct air discharge concepts	141
6.3	Storage utilization factor (η_{SV}) versus heat flow rate for multi-stage boiling and direct air discharge concepts	142
6.4	Improvement in storage utilization (η_{SV}) versus heat flow rate for the multi-stage boiling concept in comparison to the direct air discharge concept	143
6.5	Schematic of the distribution systems	145
6.6	Component weight and volume proportions for Scenario 2	147
6.7	Normalized volume of direct air and multi-stage boiling discharge concepts including their distribution systems, based on 10 kW thermal design	148
6.8	Normalized weight of direct air and multi-stage boiling discharge concepts including their distribution systems, based on 10 kW thermal design	149
6.9	Ratio of estimated electrical power requirements for fans and pumps in relation to extracted heat flow at lower limit temperature	149
A.1	Selected material properties of the storage material AlSi ₁₂ , based on data from [87]	156
A.2	Selected fluid properties of water, based on data from [42]	156

A.3	Selected fluid properties of air, based on data from [42]	157
B.1	Absolute and relative uncertainty of experimental heat flow versus mPCM temperature for bench-scale prototype experiments	159
B.2	Absolute and relative uncertainty of experimental heat flow versus $T_{w,AirHEX}$ temperature for application-scale prototype experiments	160
B.3	Absolute and relative uncertainty of experimental heat flow for Stages 1 and 2 versus mPCM temperature for multi-stage prototype experiments (for 10 l/h, 64 kg/(m ² s))	161

List of Tables

2.1	Physical properties of AlSi_{12} storage material in the operating temperature range of this work, based on data from [88]	13
2.2	Overview of high-temperature mPCM thermal storage discharge concepts in literature	18
2.3	Nusselt numbers for fully developed laminar flow in circular and rectangular channels, based on data from literature [35]	24
2.4	Boiling regimes and corresponding temperature differences for water at atmospheric pressure (data derived from [39])	31
3.1	Evaluation matrix for concept selection	61
4.1	Basic data of the bench-scale storage prototype, numbers based on data from Nees et al. [79] and additional information	76
4.2	Detailed geometry of the air heat transfer section, based on data from Nees et al. [79]	76
4.3	Characteristics of the selected mesh for the transient CHT simulation	86
4.4	Thermal contact resistances at mPCM-graphite interface based on empirical study, numbers reproduced from Nees et al. [76]	89
4.5	Basic data of the application-scale storage prototype, numbers based on data from [87] and additional information	95
4.6	Detailed geometry of the air heat transfer section	95
5.1	Basic data of the storage prototype used in the experiments (without piping)	109
5.2	Prototype channel geometry of Stages 1 and 2	110
5.3	Dedication of heat transfer regimes to specific calculation methods for the heat transfer coefficient h	128
5.4	Definition of model parameters dT_{crit} and $\dot{q}_{crit}$, and dT_{min} and $\dot{q}_{min}$	128
5.5	Adjustment factors for the adapted model of Stage 1	131
5.6	Parameters for $\dot{q}_{crit}$, dT_{crit} , dT_{min} , and film boiling HTC factor for the adapted model of Stage 2	133
5.7	Upscaled channel geometry, considering base plate dimensions identical to that of the direct air discharge application-scale prototype	136
5.8	Ratios for upscaling of experimental prototype results to the application scale for the multi-stage boiling discharge concept	136
6.1	Definition of scenarios for further comparison	144

- A.1 Material description of bench-scale air discharge and multi-stage discharge
prototypes 155
- A.2 Material description of application-scale air discharge prototype 155
- B.1 Uncertainties of single temperature measurements at various positions for
bench-scale prototype experiments 159
- B.2 Uncertainties of single temperature measurements at various positions for
application-scale prototype experiments 160
- B.3 Uncertainties of single temperature measurements at various positions for
multi-stage prototype experiments 161
- C.1 Estimation of the discretization error of the conjugate numerical simulation
under steady state conditions and 15 kg/h air mass flow rate 163

Nomenclature

Latin symbols

A	Area	[m ²]
A_c	Cross sectional area of fin	[m ²]
A_f	Fluid cross section	[m ²]
A_{ref}	Reference area	[m ²]
A_s	Surface area	[m ²]
$A_{x,BP}$	Cylindrical area based on $d_{x,BP}$ and number of fluid channels	[m ²]
a	Thermal diffusivity	[m ² /s]
a	Channel width	[m]
a^*	Aspect ratio	[-]
b	Channel height	[m]
Bo	Boiling number	[-]
c_p	Specific heat capacity	[J/(kgK)]
d	Diameter	[m]
d_h	Hydraulic diameter	[m]
dT	Temperature difference	[K]
$d_{w,S1/S2}$	Drill hole diameter in base plate corresponding to Stages 1 and/or 2	[m]
$d_{x,BP}$	Diameter based on plate thickness	[m]
E_M	Energy associated with mass transport	[J]
e	Error in numerical simulation	[%]
f	Darcy friction factor	[-]
Fr	Froude number	[-]
G	Mass flux	[kg/(m ² s)]
H	Height	[m]
H	Enthalpy	[J/kg]
h	Global grid size	[m]
h	Heat transfer coefficient	[W/(m ² K)]
h_{cb}	Convective boiling heat transfer coefficient	[W/(m ² K)]
h_{fg}	Latent heat of vaporization	[J/kg]
h_{nb}	Nucleate boiling heat transfer coefficient	[W/(m ² K)]

$h_{sensible}$	Sensible heat	[J/kg]
h_{tp}	Two-phase heat transfer coefficient	[W/(m ² K)]
h_v	Specific enthalpy of vaporization	[J/kg]
j	Colburn j-factor	[-]
K_f	Empirical parameter	[-]
k	Thermal conductivity	[W/(mK)]
L, l	Length	[m]
m	Mass	[kg]
$\dot{m}$	Mass flow	[kg/s]
$\dot{m}_v$	Vapor mass flow	[kg/s]
Nu	Nusselt number	[-]
n_{ch}	Number of fluid channels	[-]
P	Power	[W]
P_{el}	Electric energy consumption of fan and/or pump	[W]
P_F	Wetted perimeter of channel	[m]
P_H	Heated perimeter of channel	[m]
P_R	Reduced pressure	[-]
p	Pressure	[Pa]
p_{crit}	Critical pressure	[bar]
Pr	Prandtl number	[-]
Q	Heat	[J]
Q_{ex}	Extracted thermal energy	[J]
$\dot{Q}$	Heat flow	[W]
$\dot{Q}_{ex}$	Extracted heat flow	[W]
$\dot{Q}_h$	Heat flow via convection	[W]
$\dot{Q}_{k+rad}$	Heat flow via conduction and radiation	[W]
$\dot{Q}_m$	Heat flow via mass transport	[W]
$\dot{Q}_{sub}$	Subcooled heat flow	[W]
$\dot{q}$	Heat flux	[W/m ²]
$\dot{q}_{crit}$	Critical heat flux	[W/m ²]
$\dot{q}_{ex}$	Specific extracted heat flow	[W/m ²]
$\dot{q}_h$	Heat flux in fluid channel	[W/m ²]
$\dot{q}_{min}$	Minimum heat flux	[W/m ²]
q_H''	Effective averaged heat flux	[W/m ²]
R	Thermal resistance	[m ² K/W]
R_c	Thermal contact resistance	[m ² K/W]
R_{cond}	Thermal resistance due to conduction	[m ² K/W]
R_h	Thermal resistance due to convection	[m ² K/W]
R_s	Specific gas constant	[J/(kgK)]
R_{th}	Overall thermal resistance	[m ² K/W]

Re	Reynolds number	[-]
r	Grid refinement factor	[-]
T	Temperature	[K]
T_{air}	Air temperature	[K]
T_{BP}	Average base plate temperature	[K]
T_{crit}	Temperature at critical heat flux	[K]
T_f	Fluid temperature	[K]
T_{min}	Temperature at Leidenfrost point	[K]
T_{mPCM}	mPCM temperature	[K]
T_s	Fluid saturation temperature	[K]
T_{sub}	Temperature of the subcooled liquid	[K]
T_w	Wall temperature	[K]
$T_{w,AirHEX}$	Wall temperature at air heat exchanger interface	[K]
$T_{w,S1}$	Base plate channel wall temperature of Stage 1	[K]
$T_{w,S2}$	Base plate channel wall temperature of Stage 2	[K]
t	Time	[s]
t_{fin}	Fin thickness	[m]
t_{plate}	Thickness of cover plate	[m]
U	Channel perimeter	[m]
U	Overall heat transfer coefficient	[W/(m ² K)]
u_c	Uncertainty	[-]
V	Volume	[m ³]
$\dot{V}$	Volume flow	[m ³ /s]
v	Velocity	[m/s]
W	Width	[m]
W	Work	[W]
We_{fo}	Liquid Weber number	[-]
X_{tt}	Lockhart-Martinelli parameter based on turbulent liquid-turbulent vapor flows	[-]
$\dot{x}, x$	Vapor mass fraction	[-]
x, y, z	Coordinates	[m]

Greek symbols

α	Heat transfer coefficient, German notation	[W/(m ² K)]
α_{film}	Film boiling heat transfer coefficient	[W/(m ² K)]
α_{sub}	Subcooled film boiling heat transfer coefficient	[W/(m ² K)]
Δ	Differential	[-]
ΔE	Change in internal energy	[J]

Δh_f	Latent heat of fusion	[J/kg]
Δp	Pressure drop	[Pa]
ΔT	Temperature difference	[K]
ε	Emission coefficient	[-]
ζ	Friction factor	[-]
η	Efficiency	[-]
η_{fin}	Fin efficiency	[-]
η_{SU}	Storage utilization	[-]
η_{TEF}	Thermal enhancement factor	[-]
Θ	Temperature difference	[K]
Θ_{amb}	Ambient temperature	[°C]
λ	Thermal conductivity, German notation	[W/(mK)]
μ	Dynamic viscosity	[kg/(ms)]
ν	Kinematic viscosity	[m ² /s]
ρ	Density	[kg/m ³]
σ	Surface tension of liquid	[J/m ²]
ϕ	Field variable	[W]

Indices

0	Reference
1, 2, 3, ...	Position, identifier
<i>A</i> , <i>B</i>	Identifier
<i>air</i>	Corresponding to air
<i>amb</i>	Ambient
<i>bulk</i>	Corresponding to fluid bulk
<i>ch</i>	Channel
<i>cond</i>	Conduction
<i>crit</i>	Corresponding to critical heat flux
<i>d</i>	Diameter
<i>ext</i>	Extrapolated value
<i>f</i>	Fluid
<i>fin</i>	Fin
<i>g</i>	Vapor
<i>in</i>	Corresponding to inlet
<i>i</i> , <i>n</i>	Numerical enumerator
<i>l</i>	Liquid
<i>latent</i>	Corresponding to latent energy
<i>m</i>	Mean value, average
<i>max</i>	Maximum

<i>min</i>	Minimum
<i>mix</i>	Corresponding to air mix temperature
<i>mPCM</i>	Corresponding to storage material
<i>OP</i>	Operation
<i>out</i>	Corresponding to outlet
$\dot{q}$	Heat flux boundary
<i>rad</i>	Radiation
<i>S1</i>	Corresponding to Stage 1
<i>S2</i>	Corresponding to Stage 2
<i>s</i>	Solid
<i>sensible</i>	Corresponding to sensible energy
<i>sub</i>	Subcooled liquid
<i>T</i>	Temperature boundary
<i>TB</i>	Transition boiling
<i>total</i>	Corresponding to total energy
<i>tube</i>	Tube
<i>v</i>	Vapor
<i>w</i>	Wall
∞	Ambient condition

Physical constants

σ	Stefan–Boltzmann constant, $5.670 \cdot 10^{-8} \text{ [W/(m}^2\text{K}^4\text{)]}$
g	Gravitational acceleration at sea level, $9.807 \text{ [m/s}^2\text{]}$

Abbreviations

0D	Zero-Dimensional
1D	One-Dimensional
2D	Two-Dimensional
3D	Three-Dimensional
AlSi ₁₂	Aluminium Silicon Alloy
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CHF	Critical Heat Flux
CHT	Conjugate Heat Transfer
CO ₂	Carbon dioxide
DLR	Deutsches Zentrum für Luft- und Raumfahrt
DNB	Departure from Nucleate Boiling
EV	Electric Vehicle

GCI	Grid Convergence Index
HEX	Heat Exchanger
HTC	Heat Transfer Coefficient
HTF	Heat Transfer Fluid
JCGM	Joint Committee for Guides in Metrology
LHTES	Latent Heat Thermal Energy Storage
LiOH	Lithium hydroxide
MGA	Miscibility Gap Alloy
mPCM	Metallic Phase Change Material
NaK	Sodium potassium alloy
ONB	Onset of Nucleate Boiling
PCM	Phase Change Material
PEM	Proton Exchange Membrane
RSME	Root Mean Square Error
SoC	State of Charge
TES	Thermal Energy Storage
VDI	Verein Deutscher Ingenieure

1 Introduction

In this chapter, the context of this thesis and the underlying motivation is described. This is followed by a definition of the problem, objectives, methodological approach, and chapter structure of this thesis.

1.1 Background and motivation

Today's battery-powered electric vehicles have drawbacks compared to conventional vehicles when it comes to heating in cold environments. Compared to internal combustion engines or fuel cells, battery-powered electric motors produce little waste heat that can be used for heating. In cold climates, the energy demand for cabin heating can exceed the energy demand for traction [29, 127]. In particular, battery-powered electric buses and trains are currently facing the problem of a limited range during winter months [12, 59]. The state of the art for heating electric vehicles comprises resistance heaters and heat pumps [86, 128]. As both devices are battery powered, the range is reduced when heating. In addition, heat pumps become less efficient as the ambient temperature decreases because there is less heat available in the outside air. At temperatures below $-10\text{ }^{\circ}\text{C}$, the heating capacity of conventional heat pumps is significantly reduced, requiring additional backup heaters to fulfill the vehicle's heating demand [59].

To address these issues, thermal energy storage (TES) for electric vehicles (EVs) has attracted increasing interest among scientists and engineers in recent years [124]. Compared to resistance heaters or heat pumps, TES can reduce the electrical energy demand, which can have a significant impact on the effective range. The basic idea is to charge the thermal energy storage in parallel with the charging of the electric vehicle's battery and provide the heat needed for cabin heating while driving. Thus, battery capacity is conserved and a loss in range is prevented, in those cases where TES is added additionally. The storage of heat enables time-decoupled operation and is of interest in many technical areas, including thermal buffering in solar thermal power plants and industrial processes, thermal management in spacecraft and electronics, and heating and cooling in buildings [73]. In literature, thermal energy storage systems are classified as sensible, latent or thermochemical heat storage [37, 73, 106]. The storage material and the operating temperature range strongly influence their application potential and characteristics. In mobile applications, the use of a TES system for cabin heating in a vehicle requires high energy density and high charge/discharge rates at low system cost to be competitive with conventional batteries.

Several TES technologies have been proposed in the literature for use in EVs, including

sensible, latent, and thermochemical energy storage [124]. Compared to sensible TES, latent TES based on phase change materials (PCM) have the potential to offer higher energy densities. A subgroup of PCMs are metallic phase change materials (mPCMs). These materials are energy dense, highly thermally conductive and easily available, making them ideal candidates for storing heat in a compact, lightweight and cost-effective manner. Unlike conventional PCM materials, metallic phase change materials combine the advantages of both high energy density and high thermal conductivity [40, 88, 98]. These characteristics enable high effective storage densities and high thermal discharge rates, which are important features for mobile applications.

Figure 1.1 shows the operating energy density ranges for different thermal storage media per volume, including mPCMs, and a typical range for batteries used in electric vehicles. These ranges are based on the storage material and the battery cell itself, rather than the entire system. The operating energy density is useful for comparing different storage materials because the temperature range of actual use is taken into account. In the graph, sensible heat only implementations are colored green, the battery range is colored red, and latent heat implementations are colored blue. Clearly, mPCMs have the highest volumetric energy densities of any thermal storage medium shown, and significantly higher than battery storage densities. The same is true when considering the gravimetric energy densities, which are not shown in the graph and account for energy density per weight. Therefore, metallic latent heat thermal energy storage (LHTES) is a promising storage technology.

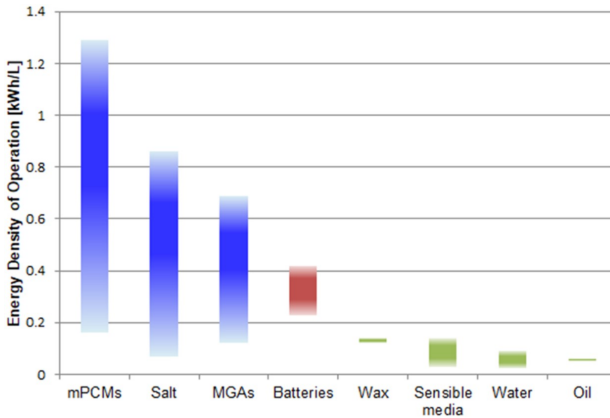


Figure 1.1: Operating energy density ranges for different thermal storage media and a typical range for batteries used in electric vehicles, reproduced from [88]

Regarding latent heat of fusion, energy density and cost, the eutectic aluminum-silicon alloy (AlSi_{12}) is an ideal candidate as storage material according to Rawson et al. [88]. As discussed further in Subsection 2.2.2, this material is considered as storage material in this thesis. However, disadvantages of this material are the high operating temperature

and the fact that the liquid aluminum-silicon alloy is highly corrosive to metals such as steel substrates [21].

For applications in solar thermal power plants, the use of mPCMs has been the subject of extensive research, simulation, and early experimental prototypes [22, 56, 83, 89]. All of these studies have in common that they use only the high-temperature latent heat of the storage material. Thus, the operation is nearly isothermal with respect to thermal discharge.

For cabin heating in electric vehicles, a first experimental study of AlSi_{12} as storage material was published by Kraft et al. using ambient air for thermal discharge [58]. Luo et al. performed another experimental study with the same storage material for heating in EVs, using also air as the discharge fluid [66]. In this study, a storage density of 225 Wh/kg and a thermal output of 1.5 kW for 3 hours has been reported [66]. Both studies did not consider corrosion protection against the highly corrosive liquid metal and were limited to the demonstration of an initial experimental prototype without addressing in-depth heat transfer issues.

1.2 Problem definition and objectives

Despite its promising potential, the use of metallic LHTES in mobile applications is still uncharted territory according to the current state of technology and research, with the exception of a few initial studies. Even stationary applications, for which there are more publications and technical concepts in the literature, are still in the early stages of development. In general, and especially in mobile applications, one of the main challenges of the technology is the thermal discharge of the storage unit.

To this day, there is a lack of concepts, simulations, and experimental studies on the subject, and no satisfactory technical solutions exist for mobile applications. Consequently, the current state of the art is too weak to build upon. Therefore, the first problem is to design a heat discharge system for a new application, with a large number of degrees of freedom, which is a complex task. These degrees of freedom include the heat transfer fluid and its parameters (e.g., mass flow, pressure, and temperature), the heat transfer mechanism (e.g., free or forced convection and single-phase or two-phase flow with phase change), the detailed geometry of the heat transfer structure, and the control of the heat flow.

The mobile application results in numerous requirements for a thermal discharge system. These include a compact system design with low weight and volume, a high storage density, a low electrical power requirement for pumping the heat transfer fluid, low system complexity and low costs. In addition, there is the problem of heat extraction only on demand.

In order to increase the effective storage density, one way to do this is to maximize the energy output of both the latent heat and the sensible heat. This leads to the next

challenge, a wide operating temperature range for thermal discharge, which is in contrast to near isothermal operation where only latent heat is extracted. Previous studies on mPCM heat extraction in literature relate to stationary applications in energy and process technology [56, 73, 89]. These are mostly limited to a small operating range close to the phase change temperature and therefore cannot be transferred as approach to this thesis.

Figure 1.2 shows the temperatures of the mPCM and the cabin air temperature versus the operating thermal energy for the AlSi_{12} storage material considered. The phase change temperature of the storage material is about 577 °C, as shown by the plateau in the temperature of the AlSi_{12} in Figure 1.2. The discharge is highly transient with a wide operating temperature range. This results in large temperature differences between the storage material, adjacent components, and the heat transfer fluid.

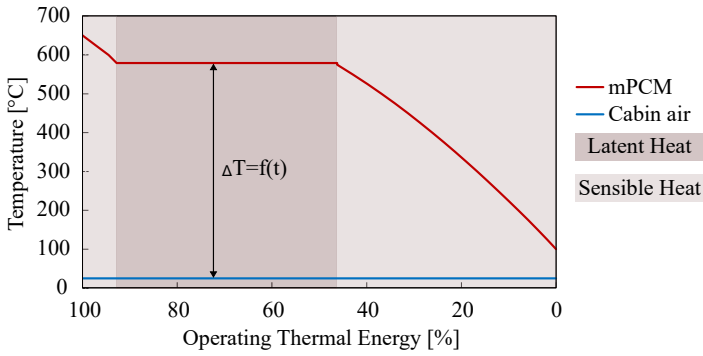


Figure 1.2: Temperatures of mPCM and cabin air versus operating thermal energy for AlSi_{12} storage material

Based on the above, the following problems arise for common types of heat transfer: First of all, the selection of potential liquid heat transfer fluids is very limited in the considered temperature range. Due to the high operating temperatures of up to 650–700 °C, the solid heat transfer structure selection is also limited. Secondly, due to the large temperature differences, massive effects due to the boiling crisis are expected on the performance of a boiling heat transfer concept. High temperature differences between the wall and the saturation temperature of the working fluid lead to the boiling crisis [34, 39], which significantly reduces the boiling heat transfer performance in both flow and pool boiling. Third, the heat transfer performance of gaseous media such as air, is severely limited at small temperature differences. Fourth, a direct heat transfer to the vehicle coolant circuit, such as water-glycol, is not possible, due to the high temperatures of the storage material.

Another problem for development, investigation, and analysis is the complex physics involved. In fact, the thermal discharge of the storage is a complex and transient multiphysics problem. On the one hand, there is the complex solidification process of the

storage material, with time-dependent thermal contact resistance at the surrounding interfaces. On the other hand, there is the complex heat transfer of the working fluid, in particular when boiling and two-phase flow are involved. Both result in a time-dependent and fast changing temperature profile in all components involved and are also critical in terms of thermomechanics.

The main objectives of this thesis can therefore be summarized by the following research questions:

How can a high-temperature metallic phase change thermal energy storage be discharged in the context of a mobile application?

What are important physical effects and performance characteristics of the thermal discharge of such a storage system?

Which functional and technical design characteristics lead to an efficient thermal discharge system?

To address these research questions, this thesis focuses on the development and investigation of heat extraction concepts for thermal energy storage based on metallic phase change materials in mobile applications. The objectives of the present work are therefore, on the one hand, to advance the technology and, on the other hand, to identify important physical effects and characteristics of the thermal discharge. In this work, the aluminum-silicon alloy AlSi_{12} , which has a phase change temperature of around $577\text{ }^{\circ}\text{C}$, is considered as storage material. It is housed in a box-shaped container, while heat extraction takes place at the bottom. Two main heat extraction concepts were selected for further prototyping, simulation, and experimental investigation based on conceptual development and several selection criteria. The first selected concept applies direct air discharge to the cabin and is considered to be the simplest approach. In addition to the experimental investigation of a bench- and application-scale prototype, a conjugate computational fluid dynamics simulation was performed to study the transient thermal discharge including the phase change of the storage material. The second concept developed is more sophisticated and is based on a novel multi-stage boiling approach. Heat extraction is achieved through a closed-loop boiling and condensing cycle using water as the working fluid, allowing heat to be transferred to the vehicle's cooling circuit. The heat discharge characteristics of both concepts were investigated in a temperature range from $650\text{--}100\text{ }^{\circ}\text{C}$. The objective is to demonstrate the functionality, performance, and efficiency of thermal discharge and to characterize the complex physics of the heat extraction process, thereby contributing to the state of the art.

1.3 Methodological approach and chapter overview

The overall methodological approach chosen for this work is shown in Figure 1.3. It is based on the V-model according to the VDI/VDE guideline 2206 [117]. The aim of this

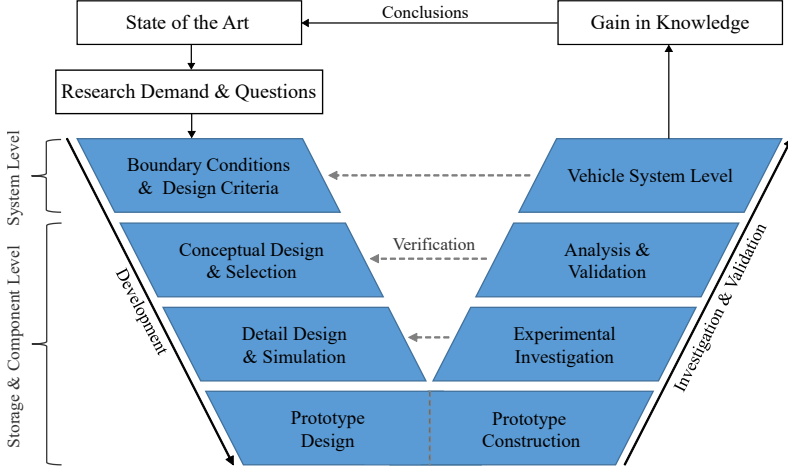


Figure 1.3: Methodological approach of this thesis

approach is to achieve a sufficiently large gain in knowledge through a structured process in order to be able to answer the research questions stated in the previous Section 1.2.

The methodological approach is divided into two paths, a descending development path and an ascending investigation and validation path, according to Figure 1.3. The development path starts with the definition of boundary conditions and design criteria, followed by conceptual design and concept selection. This is followed by detail design combined with simulations, which leads to a prototype design. The investigation path starts with the prototype construction process, followed by experimental investigations and in-depth analysis, and ends with an evaluation at the vehicle system level. The V-model is applied to each of the different prototypes developed in this thesis individually.

The development process is focused on a deductive approach, where new concepts and solutions are derived based on the state of the art and existing computational methods found in the literature. In this context, important correlations from literature are identified for the further design process. The subsequent experimental investigation, in-depth analysis and vehicle system level evaluation, on the other hand, follow an inductive approach, as new findings are brought to light, which are further analyzed and synthesized. The new findings will be used to validate the results and the design process. In the end, the knowledge gained based on all the results leads to conclusions helping to answer the research questions.

Following this introduction, Chapter 2 deals with the theoretical basis and the state of the art relevant to this work. The focus here is mainly in the areas of heat transfer and heat storage, in particular on single-phase and boiling heat transfer as well as on latent heat thermal energy storage. Chapter 3 deals with the conceptual development and concept selection, which forms the basis for all further investigations in the following chapters.

Chapter 4 further develops and investigates the first selected concept, the direct air discharge concept. The detail design is built on a developed correlation-based calculation design model. Two prototypes, one bench-scale and one application-scale, were constructed and experimentally investigated. Additionally, a conjugate CFD simulation model was developed to analyze the heat extraction process in more detail. The model was validated based on the experimental data obtained from the bench-scale prototype. Chapter 5 focuses on the second selected concept, the multi-stage boiling concept. Therefore, a functional prototype was developed, constructed, and experimentally investigated, followed by the derivation of a heat transfer calculation model. This is followed by a comparison and evaluation of the two concepts at vehicle system level in Chapter 6. The level of discharge and the degree of storage utilization are discussed, and a comparison of the volume, weight, and electrical power requirements for different scenarios is carried out. Finally, the results of this thesis are concluded in Chapter 7, extended by an outlook on future optimization, investigation, and application potentials.

2 Theoretical basis and state of the art

In this chapter, the theoretical basis and the state of the art are reviewed and discussed. This includes thermal management in vehicles and the principle of thermal energy storage, along with relevant storage concepts. Furthermore, the physical principles of heat transfer, single-phase and boiling heat transfer, as well as limitations of common heat transfer fluids and thermal resistances are described. In addition, the most important equations and correlations applied in this thesis are presented in the following sections.

2.1 Thermal management in vehicles

In the thermal management of vehicles, essential heat transfer concepts include air cooling via forced convection (e.g., front radiator), liquid circulation cooling using a water-glycol mixture (e.g., engine cooling), and the evaporation and condensation of a coolant (e.g., air conditioning). Großmann [29] and Tschöke [111] provide an extensive overview of the current state of the art of thermal management in vehicles.

For combustion engines, numerous studies have been carried out on single-phase liquid cooling and evaporative cooling [72, 81]. However, evaporative cooling has not yet prevailed over liquid cooling [72, 91].

For Proton Exchange Membrane (PEM) fuel cell electric vehicles, there has also been extensive research into single-phase liquid and evaporative cooling [91]. Compared to conventional combustion engines, fuel cell cooling is an even more challenging task. This is because the wall temperatures are lower (90 °C compared to about 200 °C) and there's more heat that needs to be removed by the cooling system [91]. However, waste heat from the engine or fuel cell can be used for heating purposes, which is advantageous for thermal management because no additional heat has to be provided in most cases.

Battery-powered electric motors, in contrast, produce little waste heat that can be used for heating, in comparison to internal combustion engines or fuel cells. This is a drawback for thermal management when it comes to heating in cold environments. Therefore, the heating requirement that arises at cold ambient temperatures must be provided additionally. As a consequence, the energy required for cabin heating can exceed the energy required for traction in cold climates [29, 127]. Resistance heaters and heat pumps are considered to be state of the art for cabin heating in electric vehicles [86, 128]. As both devices are powered by the battery, the range is reduced when heating. At temperatures below -10 °C, the heating capacity of conventional heat pumps is significantly reduced and additional backup heaters are required to fulfill the vehicle's heating demand [59]. This

is why battery-driven electric vehicles, in particular buses and trains, experience limited range during the winter months [12, 59].

The energy demand for cabin heating in a vehicle depends on many parameters, including the ambient temperature, vehicle size, and vehicle type (e.g., car, bus, train, airplane, etc.). A rough estimation of the heating demand of a 12-meter mid-size city bus as a function of the ambient temperature (Θ_{amb}) is given by the equation of Stahl et al. [104]:

$$\dot{Q}_{12m} = (11920 - 574 \cdot \Theta_{amb}/^{\circ}\text{C})W \quad (2.1)$$

The equation is based on a linear model of several literature references, including investigations by Vehviläinen et al. [115]. Using the equation yields a heating demand of 11,920 W at an ambient temperature of 0 °C and 20,530 W at -15 °C.

According to state-of-the-art technology, heat is transferred into the vehicle’s water-glycol cooling circuit for further distribution, or directly into the cabin air, depending on the vehicle type. The water-glycol distribution system consists of piping, a liquid pump, and an air-water/glycol heat exchanger with associated air fans. Aluminum fin heat exchangers are commonly used for this purpose in vehicles.

For the sake of completeness, it should be noted that thermal management of lithium-ion batteries in electric vehicles is an active area of research with numerous publications in the literature. However, the operating temperature range of 15–35 °C [125] is much lower than that of the mPCM thermal storage discussed in this thesis.

2.2 Thermal energy storage technologies

2.2.1 Physical principle of thermal energy storage

Thermal energy storage (TES) enables the storage of thermal energy in suitable media and the discharge of the stored energy when required. TES is a rapidly developing and growing field of research with a wide range of real-world applications. Thermal energy storage systems can be classified into three types: sensible, latent, and thermochemical [37, 73, 106]. These types are described below:

Sensible heat storage

Energy can be added to a material by heating it to a higher temperature. The energy involved in this process is called “sensible heat” [37]. Sensible heat ($Q_{sensible}$) is the product of the mass m and the specific heat c_p of the material involved, as well as the temperature change (T_1 to T_2):

$$Q_{sensible} = \int_{T_1}^{T_2} mc_p(T)dT \quad (2.2)$$

Storage media for sensible heat include liquid media, such as water in hot water tanks and molten salts, as well as solid media, including alloys, ceramics, and rocks. In this work, sensible heat is relevant to all components of the developed storage system.

Latent heat storage

Another mechanism for storing thermal energy involves the phase transition of the storage material. In latent heat storage, thermal energy is stored through the phase change of the storage material. Latent heat is absorbed or supplied at a constant temperature and is considered as an isothermal process. This is in contrast to sensible heat, which involves a temperature range. Unlike thermochemical energy storage, there is no change in chemical composition. The phase change is usually solid-liquid, but it could also be solid-gas or liquid-gas. The storage material associated with latent heat storage is commonly called phase change material (PCM).

The latent heat of fusion (Q_{latent}) is the product of the mass of the PCM storage material involved and its specific phase change enthalpy Δh_f :

$$Q_{latent} = m\Delta h_f \quad (2.3)$$

As latent heat of fusion is stored in addition to sensible heat, the total stored energy (Q_{total}) and, consequently, the energy density can be increased when sensible and latent heat are combined:

$$Q_{total} = Q_{sensible} + Q_{latent} \quad (2.4)$$

Figure 2.1 shows the sensible and latent heat versus temperature as well as the stored thermal energy.

Storage of latent heat offers advantages over sensible heat such as higher energy storage capacities and the ability to maintain a constant operating temperature. In many applications, operating at a constant temperature is crucial to the process.

Storage materials can be classified as either ordinary phase change materials (PCM), such as paraffins and salt nitrates, or metallic phase change materials (mPCM), such as metal alloys. In this work, latent heat is relevant for the used mPCM storage material.

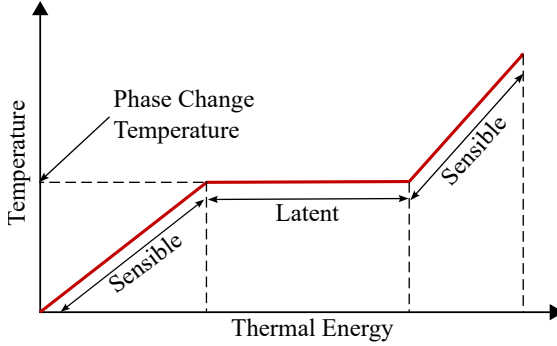


Figure 2.1: Sensible and latent heat versus temperature and stored thermal energy

Thermochemical energy storage

In thermochemical energy storage, thermal energy is stored in reversible chemical reactions. In general, chemical reactions result in the absorption or creation of heat, which is the underlying physical principle of thermochemical energy storage. The related process is also described in terms of quasi-latent heat [37]. In this work, energy storage in the mPCM material and adjacent storage system components involves only sensible and latent heat, not reversible chemical reactions. Therefore, thermochemical energy storage is not described in more detail. Additional information can be found in the related literature [37, 106].

2.2.2 Metallic phase change materials and AlSi_{12} as storage material

Metallic phase change materials (mPCMs) are metals or metal alloys that store and release thermal energy involving a phase change. Therefore, they are related to latent heat thermal energy storage. The phase change typically occurs from solid to liquid and back, at the specific phase change temperature of the material. They are known for their high energy densities and high thermal conductivities [40, 88, 98], which allow for compact energy storage and high thermal charge/discharge rates. This makes mPCMs suitable for mobile applications, such as electric vehicles, as well as for applications requiring efficient thermal energy storage and management. Figure 1.1 in Chapter 1 shows the operating energy density ranges for mPCMs compared with different thermal storage media and the typical range for batteries used in electric vehicles.

Selecting an appropriate storage material is an important step in developing a thermal energy storage system. The selection of a suitable mPCM depends on the process temperatures and the material properties of the storage medium. The selected mPCM should have high gravimetric and volumetric energy densities, high thermal conductivity, a melting point within the operating temperature range, chemical stability, and the ability to maintain its thermal properties after a large number of phase change cycles. Additionally, the mPCM should be cost-effective, easy to obtain and handle, non-hazardous, and

recyclable.

Eutectic compositions are beneficial for alloys, which are mixtures of two or more pure metal components, because the phase change occurs at a single temperature rather than over a large range from the solidus to the liquidus temperature [88]. This minimizes the effects of phase separation.

A number of publications are related to the selection process of metallic phase change material (mPCM) for thermal energy storage, including those by Rawson et al. [88], Shamberger et al. [98], and Rea et al. [90]. According to Rawson et al., the eutectic aluminum-silicon alloy (AlSi_{12}) is an ideal storage material candidate due to its latent heat of fusion, energy density, and cost [88]. Several researchers have also focused on AlSi_{12} for storage applications, including Kraft et al. [58] and Luo et al. [66] for cabin heating in vehicles, and Kotze [56] and Rea et al. [89] for solar thermal energy. The physical properties of AlSi_{12} are detailed in Table 2.1. Since AlSi_{12} was chosen as the storage material for this thesis, additional relevant temperature-dependent material properties are provided in Appendix A.

mPCM	Composition	Phase change temperature	Heat of fusion	Heat capacity Solid	Heat capacity Liquid
	[Wt.%]	[°C]	[kJ/kg]	[kJ/(kgK)]	[kJ/(kgK)]
AlSi_{12}	Al-12.5 wt.% Si	577	504	1.10	1.11

Table 2.1: Physical properties of AlSi_{12} storage material in the operating temperature range of this work, based on data from [88]

Figures 2.2(a) and 2.2(b) illustrate the gravimetric and volumetric energy densities within an operating temperature range of 160 °C to 660 °C, compared to the melting points of pure metals and eutectic alloys. According to Rawson et al. [88], this operating temperature range was chosen for space heating in vehicles. As can be seen, AlSi_{12} has the highest gravimetric energy density and one of the highest volumetric energy densities of all the plotted metals and alloys within the operating temperature range. Its gravimetric energy density is 293 Wh/kg, and its volumetric energy density is 708 Wh/L [88].

In contrast, the energy density of battery systems currently used in electric buses is much lower. The gravimetric energy density of today’s commercial lithium-ion batteries ranges from 106 Wh/kg for high-power systems to 183 Wh/kg for high-energy systems per battery pack [59]. The corresponding volumetric energy density ranges from 141 Wh/L for high-power systems to 235 Wh/L for high-energy systems per battery pack [59]. On a battery cell level and not the entire battery pack, energy densities of 230 to 350 Wh/L have been reported for batteries in electric vehicles under development or in use [108].

According to Figure A.1 in Appendix A, the thermal conductivity of AlSi_{12} depends on temperature and state. In its liquid state at 600 °C, its thermal conductivity is

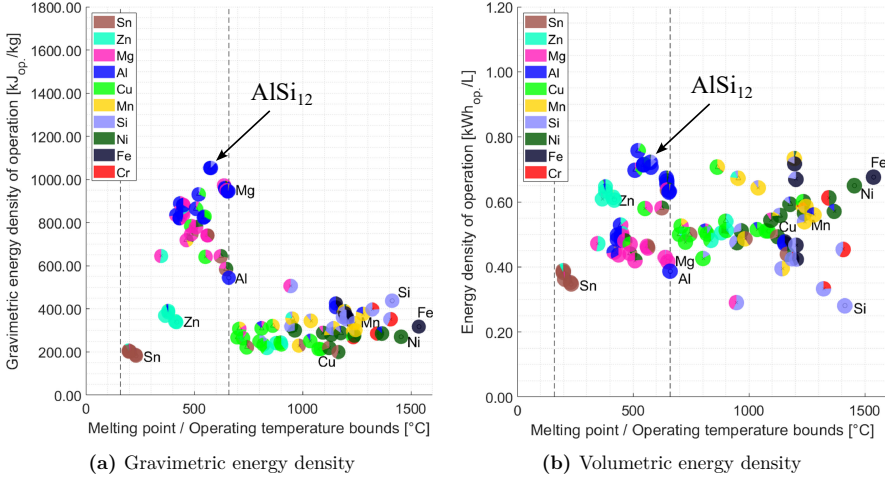


Figure 2.2: Gravimetric and volumetric energy densities versus melting point. The colors in each pie chart represent the elemental components and their respective mass fractions. The operating temperature range from 160 °C to 660 °C is marked with dashed lines. Adapted from [88].

66.5 W/(mK), and in its solid state at 300 °C, it is 170.4 W/(mK). Compared to ordinary phase-change materials (PCMs), such as salt compositions, which have low thermal conductivities in the range of 1 W/(mK) [48], the thermal conductivity of AlSi₁₂ is more than 100 times higher, which improves the storage system's charge and discharge capabilities. AlSi₁₂'s thermal conductivity is even higher than that of many pure metals and metal alloys, including stainless steel.

However, utilizing metallic phase change materials has several disadvantages and poses numerous challenges. First, the liquid state of mPCMs requires an appropriate enclosure to prevent the storage material from effusing. Second, liquid aluminum-silicon alloys are highly corrosive to metals such as steel substrates [21]. So far, the commercial application of mPCM is limited due to their corrosion behavior with conventional container materials. Therefore, careful selection of the container material is required. Fukahori et al. and Rawson et al. list several compatible container materials [21, 88]. These include graphite and ceramics such as aluminum oxide (Al₂O₃), magnesium oxide (MgO), and silicon carbide (SiC) [88]. Stahl et al. reported that carbon fiber-reinforced silicon carbide (C/C-SiC), a type of ceramic matrix composite, has potential applications as a container material [101].

Furthermore, the disadvantages of mPCM include shrinkage effects and the formation of shrinkage cavities during solidification and cool down process. Shrinkage effects result in a reduction of volume during the solidification process and as the temperature is cooled down further. Shrinkage in molten metal can lead to void defects, also known as shrinkage

cavities. These defects can form inside the material or on its outer surface [84]. These effects can reduce the effective contact area with the surrounding container and deteriorate the heat transfer capability.

2.2.3 Thermal energy storage systems in stationary and mobile applications

Thermal energy storage (TES) allows time-decoupled operation and is of interest in many technical fields, such as thermal buffering in solar thermal power plants and industrial processes, thermal management in spacecraft and electronics, heating and cooling in buildings, and vehicle applications. Notable examples for stationary and mobile applications are given below.

Stationary applications

Examples of thermal energy storage in stationary applications can be found in buildings, such as thermal buffering for heating and cooling (sensible storage with water), cold storage (ice), night storage heaters (heating bricks) [106]. One of the most active research areas is in the field of solar thermal energy. Of particular interest are storage systems based on salts for sensitive and latent storage in solar thermal power plants [9, 107]. The use of mPCM is also being investigated for this purpose [73, 89]. For utilizing industrial waste heat, approaches with mPCM as a storage material aim to generate process steam [54]. For space applications, considerations for storing sensible heat in lunar and martian rock are also worth mentioning [107].

Mobile applications

The investigation of thermal energy storage for the application in vehicles has already been conducted during the past decades. These studies have focused on preheating combustion engines [30] and catalytic converters [55] or optimizing the use of thermoelectric generators [51] using phase-change materials (PCMs). For the use in electric vehicles, several different TES technologies have been proposed in the literature, including sensible, latent, and thermochemical heat storage systems [124]. In the field of spacecraft, PCM-based heat storage systems using lithium hydroxide (LiOH) have been developed for space stations to generate electricity via an organic Rankine cycle [62]. Current research focuses on PCMs for temperature equalization in satellites, which are integrated into components using additive manufacturing processes [15]. A similar approach is being pursued for temperature control of lithium-ion batteries in electric vehicles. In this application, a PCM is used as a passive thermal management system [36].

A relatively new research field is the use of a TES system for cabin heating in electric vehicles in winter months, which requires rather large amounts of energy. Compared

to stationary applications, the requirements of such as a TES system for cabin heating are high energy density, high charge and discharge rates, and low costs to compete with conventional batteries. Therefore, concepts based on sensible heat storage using high-temperature solid-media ceramic honeycombs were proposed [17, 18, 19], as well as concepts based on mPCM using AlSi_{12} as storage material [58, 66]. These concepts are further described in the next section.

2.2.4 Thermal discharge of high-temperature heat storages in literature

The following literature review focuses on high-temperature thermal storages with a maximum operating temperature exceeding 300 °C.

High-temperature solid-media heat storages

In the literature, solid-media heat storages are often referred to as "packed-bed" heat storages. They store sensible heat without a phase change of the storage material and typically have a high maximum operating temperature and a wide operating temperature range. Air is typically used as the heat transfer fluid for thermal discharge [2, 110]. In rare cases, liquids are used. Bruch et al. investigated a thermal oil for thermal discharge at a maximum operating temperature of 300 °C [11].

The advantages of solid-media heat storage for heat transfer include direct contact between the storage material and the heat transfer fluid, as well as the ability to provide a high specific surface area through geometric design. However, the disadvantage is its lower energy density compared to thermal energy storage involving phase change when the same operating temperature range is considered.

Dreißgacker et al. proposed a high-temperature, solid-medium thermal energy storage (TES) concept for heat supply in electric vehicles based on ceramic honeycombs [17, 18, 19]. The storage material was charged to over 800 °C by electric heating wires and discharged by forced air convection [19]. Previously reported storage densities were 155 Wh/kg, and thermal outputs were 5 kW [19]. Figure 2.3 shows the TES concept and the ceramic honeycombs used. Using a honeycomb geometry for the storage material is not practical for this thesis because the mPCM requires containment in its liquid state.

High-temperature PCM storages based on salts

Salts are the most frequently used and analyzed storage medium for high-temperature applications. There are two types of salt storage systems: molten salt and latent heat.

On the one hand, molten salt storage systems are the state-of-the-art in solar thermal power plants [9, 89]. Molten salt is pumped from a hot reservoir to a cold reservoir, passing through a heat exchanger for thermal discharge. This process transfers heat,

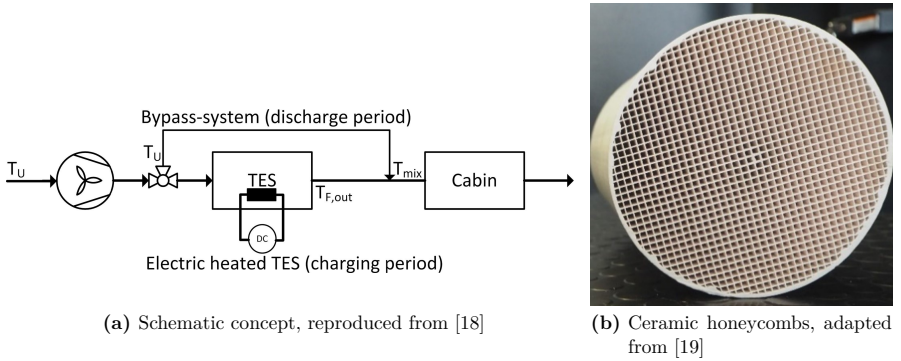


Figure 2.3: Solid-medium TES concept for heat supply in electric vehicles and ceramic honeycombs used as storage material

which is typically used to vaporize a working fluid, extracting sensible heat from the molten salt. Therefore, the molten salt acts as both a storage medium and a working fluid at the same time.

On the other hand, salt-based latent heat storage systems utilize phase change energy from solid to liquid and back. Because the temperature remains constant during the phase change, these systems are of interest for generating process steam. The heat transfer fluid used is typically high-pressure water/steam.

A major disadvantage of using salts for heat transfer is their low thermal conductivity, especially in the solid phase. As a result, achieving fast charging and discharging rates as required in vehicle applications is difficult. Numerous developments and investigations have been conducted to intensify heat transfer by using fin structures [38, 112]. Another method of improving thermal discharge is the use of flat-plate heat exchangers, which consist of several superimposed plates alternately filled with PCM and the heat transfer fluid [120]. Microencapsulated PCMs are a further improvement on this principle. Another developed and experimentally proven concept to improve heat transfer in salt-based latent storage systems is the rotating drum heat exchanger, which allows for constant, fully controllable heat transfer during the discharge process [109]. These technical concepts are not considered further in this thesis because mPCMs have a significant higher thermal conductivity than salts, making heat transfer enhancement on the storage material side needless.

High-temperature mPCM heat storages

The number of references in the literature on thermal storage based on mPCMs is much lower than the number of references for non-metallic PCMs. The high thermal conductivity of the metallic storage material is a major advantage for the thermal discharge of

a mPCM system compared to non-metallic PCMs. This makes complex fin structures or other concepts on the storage material side, as with salt-based PCMs, unnecessary. However, mPCMs always require an enclosure, which limits direct contact heat transfer and a high specific surface area compared to solid media heat storage.

Table 2.2 provides an overview of high-temperature phase-change material (mPCM) thermal storage discharge concepts found in the literature. The use of mPCMs for solar thermal power plant applications has been the subject of extensive research, simulations, and early experimental prototypes. Other applications include process steam supply, compressed air energy storage, and cabin heating in vehicles. With the exception of vehicle applications, these studies all share the characteristic of utilizing only the high-temperature latent heat of the storage material, thus operating nearly isothermally with respect to thermal discharge.

Reference	Year	Heat transfer fluid (HTF)	mPCM	Heat extraction	Application
Luo et al. [66]	2022	Ambient air	AlSi ₁₂	Latent+sensible	Vehicle (lab-scale)
Heim et al. [33]	2021	High pressure steam	Various mPCMs	Latent heat	Process steam supply
Lanz et al. [63]	2019	Water/Steam	AlSi ₁₂	Latent+sensible	Vehicle (lab-scale)
Becattini et al. [6]	2018	Compressed air	AlCuSi	Latent heat	Compressed air energy storage
Oshman et al. [83]	2017	Sodium (heat pipe)	AlSi ₁₂	Latent heat	Solar thermal energy
Giovanelli et al. [22]	2017	Compressed air	Mg56-Si44	Latent heat	Solar thermal energy
Ma and Zhang [67]	2017	Air	AlSi ₂₅	Latent heat	Solar thermal energy
Kraft et al. [58]	2016	Ambient air	AlSi ₁₂	Latent+sensible	Vehicle (lab-scale)
Andraka [3]	2014	Sodium (heat pipe)	CuSi	Latent heat	Solar thermal energy
Kotzé [56]	2014	NaK/High pressure steam	AlSi ₁₂	Latent heat	Solar thermal energy

Table 2.2: Overview of high-temperature mPCM thermal storage discharge concepts in literature

One of the first comprehensive studies of AlSi₁₂ as a storage material was conducted by Kotzé in 2014 [56]. He presented a concept that uses NaK and high-pressure steam for thermal charging and discharging in solar thermal power plants [56]. Oshman et al. designed, and Rea et al. constructed, a dispatchable LHTEs system with AlSi₁₂ as the storage material for integration into a solar power tower for electricity generation [83, 89]. They presented a thermal valve thermosiphon for on/off heat flow control using sodium as the working fluid [83, 89], as shown in Figure 2.4(a). Giovanelli et al. used a 3D CFD simulation to investigate the charging and discharging behavior of a mPCM storage system with a phase change temperature greater than 900 °C and compressed air as the

heat transfer fluid for use in solar thermal power plants [22]. The concept is shown in Figure 2.4(b).

The first experimental study on AlSi_{12} as a storage material for cabin heating in electric vehicles was conducted by Kraft et al. at the DLR Institute of Vehicle Concepts, who used ambient air for thermal discharge [58], as shown in Figure 2.5(a).

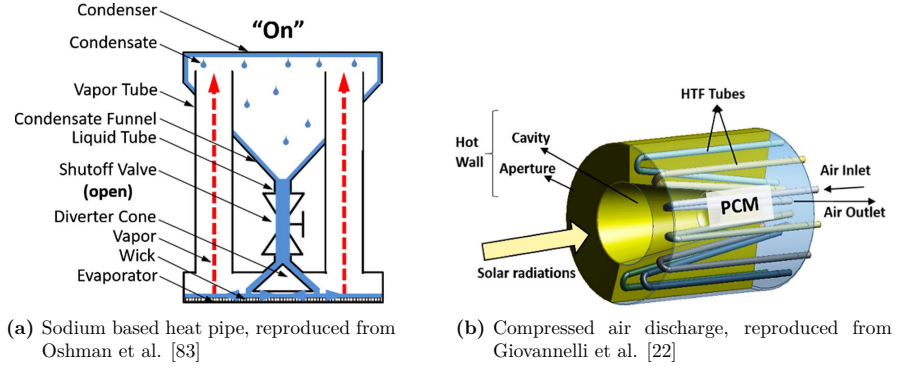


Figure 2.4: Exemplary mPCM thermal storage discharge concepts from literature

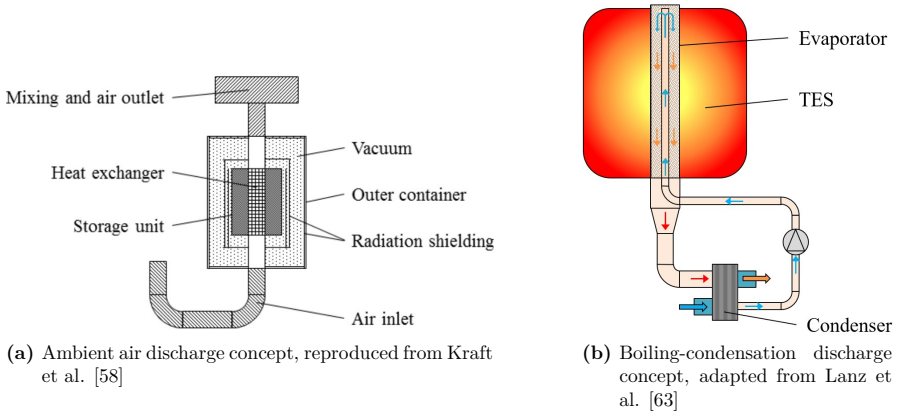


Figure 2.5: mPCM thermal storage discharge concepts from DLR Institute of Vehicle Concepts

From the same research group at DLR, Dicke experimentally investigated the impact of different storage component arrangements on the solidification process [16]. Lanz et al. proposed to link the storage to the vehicle coolant system by discharging the storage using a boiling-condensation cycle [63], as shown in Figure 2.5(b). However, experimental results have shown a low functionality of the concept for high temperatures and low mass

flows, most likely due to the boiling crisis. Furthermore, experiments were conducted without the mPCM storage material by replacing the AlSi_{12} and setting a temperature boundary condition with electric heaters.

Luo et al. performed an experimental study in which they used AlSi_{12} as the storage material and air as the discharge fluid for cabin heating in electric vehicles. They reported a storage density of 225 Wh/kg and a thermal output of 1.5 kW for three hours [66].

However, mPCM storage systems for cabin heating in electric vehicles are still in the early stages of development and are only available on a laboratory scale. Furthermore, neither prototype study addressed corrosion protection against highly corrosive liquid metals. The functional requirements on which the conceptual development of the prototypes in this work is based are outlined in Section 3.1.

2.3 Physical principles of heat transfer

Heat transfer is the process by which thermal energy is transported due to spatial temperature differences. There are three basic heat transfer mechanisms: Conduction, Convection, and Radiation. All three modes play a role in the thermal discharge of the storage system and are therefore described in the following.

Conduction

Heat conduction describes the transport of energy as a result of atomic and molecular interactions. Heat conduction does not involve any mass transfer. Heat transport by pure thermal conduction occurs predominantly in solids. The relationship between heat flux $\dot{q}$ and the temperature field ∇T is:

$$\dot{q} = -k\nabla T \quad (2.5)$$

where k is the thermal conductivity of the solid or fluid.

For a one-dimensional plane wall, having a temperature distribution $T(x)$, the equation can be simplified to [39]:

$$\dot{q}_x = -k \frac{dT}{dx} \quad (2.6)$$

The minus sign indicates that heat is always transported from the higher temperature level to the lower temperature level. The thermal conductivity is strongly dependent on the material and on the temperature. At room temperature, it is 220 W/(mK) for aluminum and 15 W/(mK) for stainless steel, whereas it is only 0.026 W/(mK) for air [61]. This comparison shows that the thermal conductivity of metals is, in general, much

higher than the thermal conductivity of gases such as air. The thermal conductivity is the dominant heat transfer mechanism in the solid parts of the storage system and in the storage material itself.

Convection

In convective heat transfer, the thermal energy is transferred by the macroscopic movement of a fluid in combination with diffusion. In the presence of a temperature gradient, such fluid motion contributes to heat transfer. Convection is classified into natural convection and forced convection. In natural convection, the temperature dependent density variations in a fluid induce buoyancy forces under the presence of gravity, causing a motion of the fluid molecules. In forced convection, the fluid flow is induced by external forces, such as pumping devices. The convective flow can be classified as internal or external flow, and as single-phase or two-phase flow.

The convective heat transfer rate in an internal flow is given by the equation:

$$\dot{Q} = hA(T_w - T_f) \quad (2.7)$$

where $\dot{Q}$ is the heat transfer rate, h is the heat transfer coefficient, A is the surface area for heat transfer, T_w is the solid wall temperature and T_f is the temperature of the fluid bulk.

The convective heat transfer coefficient h is highly dependent on the fluid properties, fluid pressure and velocity, and whether the flow is single-phase or two-phase. For boiling and condensation, h is usually significantly higher than for single-phase gas flow. The heat transfer coefficient is typically related to the Nusselt number Nu , which relates the convective heat transfer to the conductive heat transfer:

$$Nu = \frac{hL}{k_f} = \frac{hd_h}{k_f} \quad (2.8)$$

For an internal flow, the characteristic length L can be expressed by the channel hydraulic diameter, which is defined as $d_h = 4A_f/U$ for noncircular channels. A_f is the channel cross flow area and U the channel perimeter. For forced convection flow, the Nusselt number is specified as a function of the Reynolds number Re and Prandtl number Pr :

$$Nu = f(Re, Pr) \quad (2.9)$$

with

$$Re = \frac{\rho v d_h}{\mu} = \frac{v d_h}{\nu} \quad (2.10)$$

and

$$Pr = \frac{\nu}{a} \quad (2.11)$$

Here, ρ is the density of the fluid, v is the velocity, μ is the dynamic viscosity, ν is the kinematic viscosity, and a is the thermal diffusivity.

Convective heat transfer is the dominant heat transfer mechanism in the air discharge channels in Chapter 4 and in the evaporator channels in Chapter 5, and therefore plays an important role in this thesis. The single-phase and boiling heat transfer mechanisms are further described in Sections 2.4 and 2.5.

Radiation

In radiative heat transfer, all objects with a temperature above absolute zero emit electromagnetic waves to the surrounding media such as air or even vacuum, which are then absorbed by objects with a colder temperature. Radiative heat transfer is described by the Stefan-Boltzmann law, which states that the rate of radiation emitted per unit area of a black body is proportional to the fourth power of its absolute temperature:

$$P = \sigma T^4 \quad (2.12)$$

where σ is the Stefan-Boltzmann constant ($\sigma = 5.67 \cdot 10^{-8} [W/(m^2 K^4)]$ [85]).

The heat transfer from one object with a hotter surface to another object with a colder surface, assuming black body characteristics and that all radiation emitted from object 1 reached object 2, is defined as:

$$\dot{Q}_{12} = \sigma A (T_1^4 - T_2^4) \quad (2.13)$$

A black body is an ideal body with an emissivity of $\varepsilon = 1$, meaning that it absorbs all of the incident radiation. For the majority of real objects, the actual emission factors ε are between 0 and 1 and the objects are described as grey bodies rather than black bodies. For a concentric pipe situation of grey objects, it follows:

$$\dot{Q}_{12} = \frac{\sigma}{\frac{1}{\varepsilon_1} + \frac{A_1}{A_2}(\frac{1}{\varepsilon_2} - 1)} A_1 (T_1^4 - T_2^4) \quad (2.14)$$

Radiative heat transfer plays a minor role in the heat extraction process of the thermal storage in this thesis compared to conduction and convection. However, radiation effects are considered in certain sections of this thesis.

Mass transport and conservation of energy

In the presence of a working fluid, such as air or water, heat can be transferred to the working fluid by convection and can be transported to other locations in the system for further heat transfer. As defined by the first law of thermodynamics, heat is the energy transported across the boundary of a thermodynamic system due to a temperature difference between the system and its surroundings. The first law of thermodynamics is a formulation of the law of conservation of energy and is described in a general form as follows [105]:

$$\Delta E = W + Q + E_M \quad (2.15)$$

The internal energy ΔE changes due to energy transport across the system boundary in the form of heat Q , work W , and energy associated with mass transport E_M .

From the first law of thermodynamics for an open system, under steady-state and steady flow conditions, with negligible potential and kinetic energy changes, and single-phase flow with no changes in latent energy, the energy equation can be simplified to:

$$\dot{Q} = \dot{m}c_p(T_{out} - T_{in}) \quad (2.16)$$

where $\dot{m}$ is the mass flow rate of the fluid, c_p is the specific heat capacity of the fluid at constant pressure at the bulk temperature ($T_{bulk} = (T_{in} + T_{out})/2$) of the fluid and T_{in} and T_{out} are the fluid temperatures at the inlet and outlet of the channel.

The fluid mass flow rate of the fluid can be expressed as:

$$\dot{m} = \rho A_f v = const. \quad (2.17)$$

where ρ is the fluid density, A_f is the cross-sectional area of the fluid channel, and v the fluid velocity. According to the conservation of mass, the mass flow entering the system is equal to the mass flow leaving the system.

Equation 2.16 is of practical importance in this thesis when it comes to calculating the heat flow from the experimental prototype data in Chapters 4 and 5. In addition, the combination of Equations 2.7 and 2.16 is important in the modeling process.

2.4 Heat transfer and pressure drop in single-phase flow

2.4.1 Convective heat transfer in internal ducts

Convective heat transfer is the main heat transfer mechanism in the air channels of the direct air discharge concept. For forced convection flow in internal ducts, a large number of correlations and calculation methods to derive the Nusselt number have been published in literature. A widely known equation goes back to Dittus–Boelter [39, 123]:

$$Nu = 0.023Re^{0.8}Pr^{0.4} \quad (2.18)$$

It accounts for thermally fully developed turbulent flow in smooth pipes, $Re \geq 10000$, and $0.7 \leq Pr \leq 160$.

The Gnielinski correlation gives a more accurate calculation of Nu and is valid over a large Reynolds number range including the transition region [23, 39]:

$$Nu = \frac{(f/8)(Re - 1000)Pr}{1 + 12.7(f/8)^{1/2}(Pr^{2/3} - 1)} \quad (2.19)$$

The correlation is valid for $3000 \leq Re \leq 5 \cdot 10^6$ and for $0.5 \leq Pr \leq 2000$. The Darcy friction factor f used in the correlation can be calculated using Equation 2.30.

As the Reynolds number decreases as hydraulic diameter decreases, the flow tends towards laminar flow condition for small channels. The critical Reynolds number for a laminar condition to prevail applies for about $Re < 2300$ [100]. For higher Re , turbulent flow is present. For laminar flow in smooth pipes, Nu is a constant value and only dependent on the geometry of the channel, while independent from Re and Pr . This is in contrast to turbulent flow.

Table 2.3 shows the Nusselt numbers for fully developed laminar flow in circular and rectangular channels for the boundary conditions of a uniform temperature and a uniform heat flux. The given numbers indicate that for rectangular channels, the higher the aspect ratio b/a , the higher the Nusselt number.

Boundary	Circular	Rect $b/a = 2$	Rect $b/a = 4$	Rect $b/a = 8$
Nu_T (Uniform T_w)	3.66	3.39	4.44	5.60
$Nu_{\dot{q}}$ (Uniform $T_{\dot{q}}$)	4.36	4.12	5.33	6.49

Table 2.3: Nusselt numbers for fully developed laminar flow in circular and rectangular channels, based on data from literature [35]

For rectangular channels with other aspect ratios as given in Table 2.3, fully developed laminar flow, and uniform temperature boundary condition, the Nusselt number can be calculated using the analytical equation [35]:

$$Nu_T = 7.541(1 - 2.61a^* + 4.97a^{*2} - 5.119a^{*3} + 2.702a^{*4} - 0.548a^{*5}) \quad (2.20)$$

with the aspect ratio $a^* = 2b/2a$, while a and b being width and height of the channel. This equation is only valid for laminar flow condition. For turbulent flow condition, the Gnielinski correlation 2.19 is recommended also for rectangular channels [35].

In this thesis, both laminar and turbulent flow play a role and have therefore to be taken into account. The laminar flow condition prevails especially in the fin channels of the air heat transfer section in Chapter 4.

2.4.2 Fins for heat transfer enhancement

According to Equation 2.7, the heat transfer rate for convection is directly proportional to the heat transfer coefficient, the heat transfer surface area, and the effective temperature difference. A common way to enhance the heat transfer rate is to use extended surfaces, called fins, which are attached to the main object. By the use of fins, the heat transfer surface area can be significantly increased, resulting in a higher heat transfer rate. The geometry and type of fin can be customized based on the application and to provide different heat transfer and pressure drop characteristics. Among the most common types of fin geometries are plain, serrated, perforated, and wavy fins. Figure 2.6 gives an overview about the different types of fins.

The temperature distribution in the fin is not uniform as the thermal conductivity in the fin is not infinite and heat is removed by convection. If an adiabatic boundary condition is assumed at the fin tip, the temperature distribution along the fin is given by [39]:

$$\frac{T(x) - T_\infty}{T_w - T_\infty} = \frac{\Theta}{\Theta_w} = \frac{\cosh m(L - x)}{\cosh mL} \quad (2.21)$$

with L the length, respectively height of the fin. The factor m is defined as:

$$m = \sqrt{\frac{hU}{kA_c}} \quad (2.22)$$

Here, h is the heat transfer coefficient, U the perimeter, k the thermal conductivity of the fin material and A_c the cross-sectional area of the fin.

The heat transfer rate associated with the fin $\dot{Q}_{fin}$ is then:

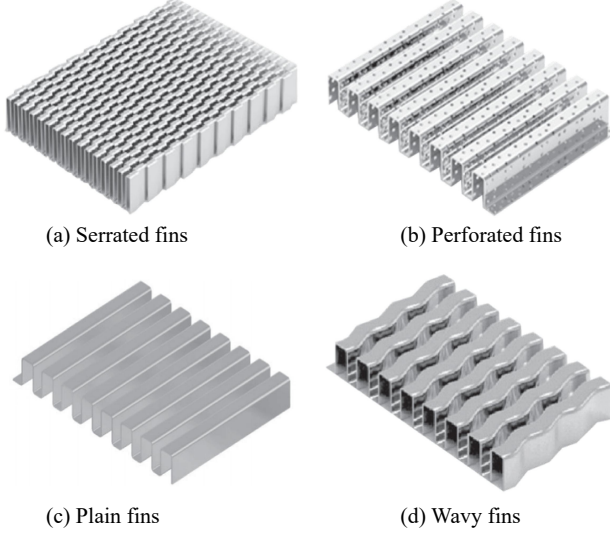


Figure 2.6: Overview about different fin types, adapted from [129]

$$\dot{Q}_{fin} = \sqrt{hkUA_c\Theta_w \tanh mL} \quad (2.23)$$

The performance of a fin can be quantified by the fin efficiency η_{fin} , which represents the actual heat transfer rate to an ideal heat transfer rate (which is an isothermal condition) for the same fin and it is given by the equation [39]:

$$\eta_{fin} = \frac{\dot{Q}_{fin}}{\dot{Q}_{max}} = \frac{\tanh mL}{mL} \quad (2.24)$$

The fin efficiency decreases with the increase in fin length due to the decrease in fin temperature. The shorter the fin length and the higher the thermal conductivity, the higher the fin efficiency.

As described above, the increase in heat transfer surface area through fins features an important enhancement of the heat transfer rate. In addition, there is another aspect which further enhances the heat transfer rate by the use of fins. As the flow pattern is affected by the physical presence of the fins, the heat transfer coefficient can be enhanced depending on the fin type and specific geometry selected. However, this comes at the price of increased pressure drop characteristics.

For plain fins, the Nusselt number can be derived using the calculation method for the rectangular fluid channel described in the previous Subsection 2.4.1.

For other geometries than plain fins, such as serrated or wavy fins, heat transfer correlations in literature are usually expressed in terms of the dimensionless j -factor defined by:

$$j = \frac{Nu}{RePr^{1/3}} \quad (2.25)$$

Serrated fins are typically the highest performer in terms of heat transfer of common fin types, however, go along with high pressure drop on the other hand. A simplified correlation for estimation purposes for the j -factor of this fin type is [35]:

$$j = 0.6Re^{-0.5} \left(\frac{d_h}{l} \right)^{0.5} \quad (2.26)$$

Where l is the length of the strip and f is the friction factor. This equation is valid for $400 < Re < 3000$ and thin fins with high fin/total area.

Wavy fins typically fall somewhere between plain and serrated fins in terms of j -factor and f -factor. According to Hesselgrave [35], there is a lack of general correlations for this fin type in literature. Kays and London recommend the following equation for a high-performance wavy fin surface [35, 47]:

$$j = 0.24Re^{-0.425} \quad (2.27)$$

This equation is valid for $400 < Re < 3000$ and can be regarded as a simplified correlation for estimation purposes only, as it does not contain specific parameters such as the wave amplitude or height.

Fins are in particular effective for single-phase gas flows with their relatively low heat transfer coefficients. In two-phase flows, there is less benefit from fins, as the heat transfer coefficients are already high. The enhancement of the heat transfer rate by means of fins plays an important role in the development, design, and analysis of the direct air discharge concept described in Chapter 4.

2.4.3 Pressure drop in internal ducts and pumping power

The pressure drop in an internal duct represents a decrease in absolute pressure along the flow path caused mainly by viscous effects and friction due to flow resistance imposed by the no-slip wall boundary condition. The determination of the total pressure drop in an internal flow system is important as it determines the pump or fan power requirements. The pressure drop Δp in pipe flow is defined by [45]:

$$\Delta p = f \frac{L}{d_h} \frac{\rho v^2}{2} \quad (2.28)$$

With the dimensionless friction factor f , the length L , the hydraulic diameter d_h , the fluid velocity v , and the fluid density ρ . In this work, the dimensionless friction factor f according to Darcy is used. By rearranging the equation, the Darcy friction factor f is defined by the equation:

$$f = \frac{\Delta p}{L} \frac{d_h}{\frac{1}{2}\rho v^2} \quad (2.29)$$

The Darcy friction factor is not to be confused with the Fanning friction coefficient, often used in literature. The Darcy friction factor is four times the Fanning friction factor ($f_D = 4f_f$).

According to Blasius, for fully developed turbulent flow in a smooth circular pipe, the friction factor f depends on Re and is given for $3000 \leq Re \leq 100000$ by [39, 45]:

$$f = \frac{0.3164}{\sqrt[4]{Re}} \quad (2.30)$$

In the laminar regime for $Re < 2300$, the friction factor f in a circular pipe can be described according to the Hagen-Poiseuille law [39, 45]:

$$f = \frac{64}{Re} \quad (2.31)$$

For rectangular channels and fully developed laminar flow, the friction factor f can be calculated using the analytical equation [35]:

$$f = \frac{4}{Re} 24(1 - 1.3553a^* + 1.19467a^{*2} - 1.7012a^{*3} + 0.95664a^{*4} - 0.2537a^{*5}) \quad (2.32)$$

With the aspect ratio $a^* = 2b/2a$.

The friction factor is multiplied by a factor of 4 different from the original literature source [35] because the literature source considers the Fanning friction factor instead of the Darcy friction factor used in this work. The rectangular channel correlation is valid for the plain fin design.

For serrated fins and wavy fins, a simplified correlation for estimation purposes for the f-factor in the range $400 < Re < 3000$ is defined by [35]:

$$f = 4(4j) \quad (\text{serrated fin}) \quad (2.33)$$

$$f = 4(1.08Re^{-0.425}) \quad (\text{wavy fin}) \quad (2.34)$$

Again, the friction factor is multiplied by a factor of 4 different from the original literature source because the literature source considers the Fanning friction factor.

Based on the reported correlations, the serrated fin produces a higher pressure drop than the wavy fin and the wavy fin produces a higher pressure drop than the plain fin. The implications for the selection of an appropriate fin for air discharge are discussed in Chapter 4.

In addition to the pressure drop defined in Equation 2.28, which increases continuously with the length of the flow path, abrupt changes in cross section of the flow path add additional pressure drop to the overall system. Such abrupt changes in cross section are caused by the inflow and outflow of fluid channels, bends and elbows in the pipe system, fittings, mixing of streams, and more. The pressure drop for all such types is determined by the following equation:

$$\Delta p = \zeta \frac{\rho v^2}{2} \quad (2.35)$$

The ζ friction factor differs from the Darcy friction factor f and is dependent on the characteristic of the abrupt change in cross section. For many common types of elements in flow systems, ζ is given in the literature in tabular form or in diagrams for engineering purposes [42, 96].

The pump or fan power P required to overcome the resistance to flow associated with the overall pressure drop can be expressed as:

$$P = v A_f \Delta p \quad (2.36)$$

With the velocity v and the fluid cross section A_f .

The electric power consumption of the pump or fan P_{el} is dependent on the efficiency η of the pumping device and therefore higher than the required power to overcome the pressure drop resistances:

$$P_{el} = \frac{P}{\eta} \quad (2.37)$$

The estimation of the pressure drop and of the electric pumping power consumption is part of Chapter 4 and Chapter 6. The power consumption of the pump plays a role in the evaluation of the efficiency of the discharge concept.

2.5 Boiling and condensation

2.5.1 Boiling modes for pool and flow boiling

Boiling is a fundamental mechanism used in many technical applications characterized by high heat fluxes. The high heat transfer coefficients associated with boiling make it an attractive method for heat extraction from thermal energy storage systems. In comparison to single-phase heat transfer systems, it is possible to reach heat transfer rates orders of magnitude higher. However, the complex two-phase nature of boiling poses a number of challenges for real-world applications.

In general, boiling heat transfer can be classified into pool boiling and flow boiling. Pool boiling works with natural convection and is often used in industrial processes, such as kettles and reboilers. Flow boiling, on the other hand, works with forced convection and is used more frequently in technical processes, such as in nuclear reactors. The advantages of flow boiling over pool boiling are higher heat transfer rates and better controllability of the process. For the thermal discharge developed and investigated in this work, flow boiling was considered instead of pool boiling.

A further distinction must be made between saturated boiling and subcooled boiling. In saturated boiling, the fluid is at boiling temperature, whereas in subcooled boiling, the fluid bulk temperature is below the saturation temperature. Both saturated boiling and subcooled boiling were considered in the development and investigation of the thermal discharge concept in this thesis.

According to Nukiyama's boiling curve, different boiling regimes apply in dependence on the wall superheat. Nukiyama's pioneering work on boiling in 1934 was the first study to identify different boiling regimes [82]. Figure 2.7 shows the characteristic boiling curve for both pool and flow boiling. Here, the heat flux is plotted against the wall superheat (also known as the excess temperature), which is defined as the temperature difference ΔT_w between the wall T_w and the saturation temperature of the fluid T_s .

With low wall superheat up to point A, the natural convection regime dominates with single-phase convective heat transfer. At point A, the onset of nucleate boiling (ONB) occurs, leading to a sharp increase in heat flux up to point B, where fully developed nucleate boiling occurs. The heat flux continues to increase until the critical heat flux (CHF) is reached at point C. After passing point C, a decrease in heat flux is observed until point D, which is known as the transition boiling or partial film boiling regime. In this regime, the vapor forms an insulating film layer over the wall surface, deteriorating the heat transfer coefficient. Yet, the conditions oscillate between film and nucleate boiling. At point D, the minimum heat flux is reached, as the surface is completely covered by

a vapor blanket, which is referred to as the Leidenfrost point. Leidenfrost was the first to discover this phenomenon in 1756 [65]. As the excess temperature increases, the heat flux increases as radiation through the vapor film becomes more significant. Passing point C, respectively CHF, towards higher wall superheat is also referred to the boiling crisis. In engineering applications, most evaporators are designed to keep the heat flux below the CHF, operating within the nucleate boiling regime. Table 2.4 summarizes the boiling regimes and their corresponding temperature differences for water at atmospheric pressure. Here, the excess temperature for the critical heat flux corresponds to dT_{crit} and the excess temperature for the minimum heat flux corresponds to dT_{min} .

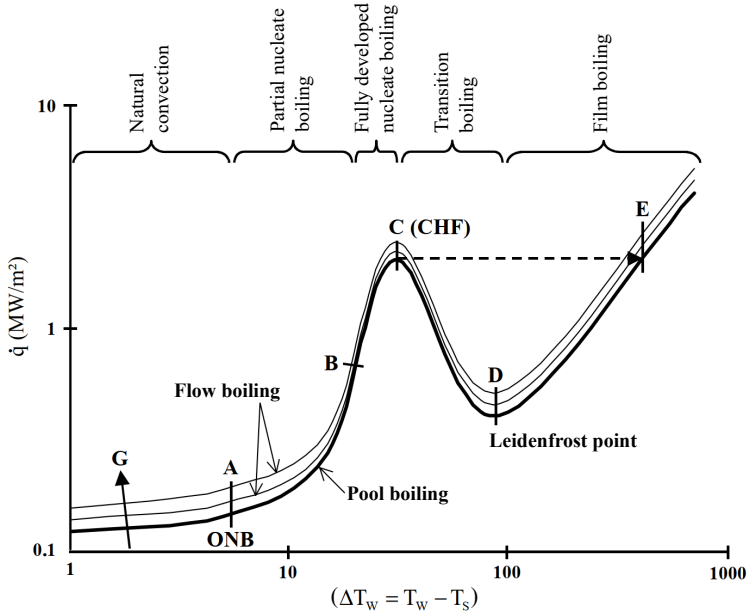


Figure 2.7: Characteristic boiling curve for pool and flow boiling, adapted from [70]

Boiling regime	$\Delta T_w = T_w - T_s$
Natural convection	$\lesssim 5$ K
Nucleate boiling	$\approx 5\text{--}30$ K ($dT_{crit} \approx 30$ K)
Transition boiling	$\approx 30\text{--}120$ K ($dT_{min} \approx 120$ K)
Film boiling	$\gtrsim 120$ K

Table 2.4: Boiling regimes and corresponding temperature differences for water at atmospheric pressure (data derived from [39])

In this work, the thermal energy storage is discharged over a temperature range of several hundred Kelvins, with start-up wall temperatures exceeding 600 °C. Considering the

discussed boiling characteristics, significant challenges for a boiling discharge system are revealed, as the system passes through different boiling regimes, including nucleate boiling, transition boiling, and film boiling. Therefore, the boiling curve characteristics and the corresponding wall superheats for each regime play an important role in the development of the multi-stage boiling discharge concept described in Chapters 3 and 5.

2.5.2 Heat transfer correlations for flow boiling

Flow boiling processes are widely used in many technical applications due to their high heat transfer rates and good controllability. However, the underlying physical mechanisms are complex, involving two-phase flow and the simultaneous transfer of heat and mass. Consequently, there has been intense research on the subject during the past few decades, which is still ongoing. Basic literature on the subject includes the books by Collier and Thome [14], Kandlikar [43], Incropera [39], and VDI Heat Atlas [42].

The majority of studies in the literature focus on nucleate boiling due to its high relevance to industrial applications. Common correlations distinguish between flow boiling in vertical and horizontal tubes, as well as between saturated and subcooled flow boiling.

Recently, there has been a lot of ongoing research related to heat transfer correlations for flow boiling in mini and microchannels. Comprehensive research and review papers on the subject are given, for example, by Karayannis et al. [44] and Kim et al. [49], offering an overview and evaluation of the latest heat transfer correlations available. Further research activities focus on heat transfer enhancement in boiling over modified surfaces. Vlachou et al. [119] provide a critical review of this topic.

The complexity of boiling heat transfer is linked to the different flow regimes that occur depending on the vapor mass fraction, flow rate, heat flux, and other factors. Figure 2.8 shows a schematic of the flow regimes for flow boiling in a horizontal tube and for nucleate boiling-dominant heat transfer. As the flow direction increases and the vapor mass fraction increases, the following flow regimes occur: Bubbly, slug, annular, and mist flow. Each flow regime has its own heat transfer characteristics. At lower mass fluxes, the bubbly regime is predominant, whereas at higher mass fluxes, the annular regime becomes predominant.

Vapor mass fraction $\dot{x}$ is an important factor in flow boiling. It is defined as the ratio of vapor mass flow $\dot{m}_v$ to total mass flow $\dot{m}$.

$$\dot{x} = \dot{m}_v / \dot{m} \quad (2.38)$$

The mass flux G is a widely used parameter in flow boiling. It is defined as the ratio of the mass flow rate $\dot{m}$ to the fluid crossflow area A_f .

$$G = \dot{m} / A_f \quad (2.39)$$

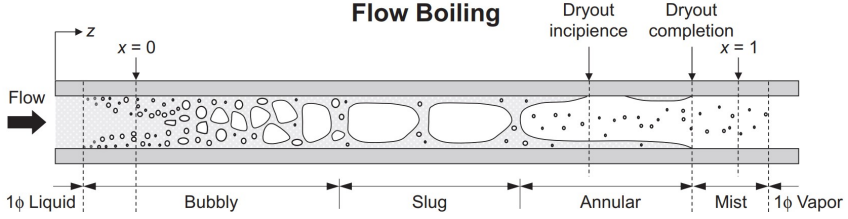


Figure 2.8: Schematic of flow regimes for flow boiling in a horizontal tube and nucleate boiling dominant heat transfer, reproduced from [49]

For estimating the nucleate boiling heat transfer coefficient, the Kim and Mudawar method [49] was found to be suitable and was selected for the calculations in Chapter 5. This method is specifically designed for flow boiling in mini channels and horizontal tubes, which are used in this work. The correlation is recommended for water and hydraulic diameters with an application range of $0.19 < d_h < 6.5$ mm, and mass flux range of $19 < G < 1608$ kg/(m²s). The pressure and liquid-only Reynolds number application range is suitable for this work. The method was developed for saturated flow boiling with $0 < x < 1$, and the correlation is based on a large database of 10,805 saturated boiling heat transfer coefficient data points from 37 sources [49].

Kim and Mudawar's correlation of the flow boiling heat transfer coefficient (h_{tp}) is based on the superposition approach of the heat transfer coefficients of the nucleate boiling (h_{nb}) and convective boiling (h_{cb}) dominant heat transfer regimes. Depending on the flow regime and heat flux, different heat transfer characteristics dominate in the channels. Superposition allows both areas to be covered with one calculation method [49]:

$$h_{tp} = (h_{nb}^2 + h_{cb}^2)^{0.5} \quad (2.40)$$

$$h_{nb} = \left[2345 \left(Bo \frac{P_H}{P_F} \right)^{0.7} P_R^{0.38} (1-x)^{-0.51} \right] \left(0.023 Re_f^{0.8} Pr_f^{0.4} \frac{k_f}{D_h} \right) \quad (2.41)$$

$$h_{cb} = \left[5.2 \left(Bo \frac{P_H}{P_F} \right)^{0.08} We_{fo}^{-0.54} + 3.5 \left(\frac{1}{X_{tt}} \right)^{0.94} \left(\frac{\rho_g}{\rho_f} \right)^{0.25} \right] \left(0.023 Re_f^{0.8} Pr_f^{0.4} \frac{k_f}{D_h} \right) \quad (2.42)$$

$$\text{with } Bo = \frac{q''_H}{G h_{fg}}, \quad P_R = \frac{p}{p_{crit}}, \quad Re_f = \frac{G(1-x)D_h}{\mu_f}$$

$$\text{and } We_{fo} = \frac{G^2 D_h}{\rho_f \sigma}, \quad X_{tt} = \left(\frac{\mu_f}{\mu_g} \right)^{0.1} \left(\frac{1-x}{x} \right)^{0.9} \left(\frac{\rho_g}{\rho_f} \right)^{0.5}$$

In the equations, Bo is the boiling number, We_{fo} is the Weber number, and X_{tt} is the Lockhart-Martinelli parameter. The Reynolds number Re and the Prandtl number Pr are defined as in Section 2.3. The vapor mass fraction x and the mass flux G are as defined by Equations 2.38 and 2.39. The heated perimeter of the channel is represented by P_H , and the wetted perimeter is represented by P_F . ρ , μ , and k are the density, dynamic viscosity, and thermal conductivity of the fluid, respectively. The index f indicates the liquid phase, and the index g indicates the vapor phase. The latent heat of vaporization is h_{fg} . The surface tension of the liquid is σ . p_R is the reduced pressure, which is related to the critical pressure p_{crit} . The heat flux $\dot{q}$ is named q_H'' in the equations and is averaged over the channel perimeter.

Unlike saturated boiling, subcooled boiling occurs when the temperature of most of the liquid is below the saturation temperature. The heat transfer rate of subcooled flow boiling may be higher than that of saturated flow boiling due to the greater temperature difference between the fluid bulk and the wall, as well as the condensation of bubbles formed at the surface in the liquid. Correlations for the subcooled flow boiling region are available in the literature [5, 43, 50].

2.5.3 Critical heat flux and boiling crisis

During the last decades, critical heat flux and boiling crisis have been intensively studied, especially in nuclear engineering. With the development and installation of the first nuclear reactors in the 1950s and 1960s, the technical importance of the topic increased, as large amounts of heat had to be transferred from the reactor core. Today, there are a large number of different CHF models, which differ strongly in type [7]. Comprehensive reviews on the topic have been provided by Kandlikar [43], Hall and Mudawar [31], Groenefeld [27], and Herbst [34]. This section provides only a brief overview and the most important findings for the further development of the technical concepts in this thesis.

Two different types of critical heat flux have been identified for flow boiling. Which type of boiling crisis mechanism is predominant depends mainly on the vapor mass fraction. At low vapor mass fractions, film boiling occurs, denoted as departure from nucleate boiling (DNB). At high vapor mass fractions, dryout occurs, which is also associated with mist flow and is considered less severe than film boiling. Both types of critical heat flow are expected to occur in the boiling heat discharge system of a high-temperature thermal storage system such as the one in this thesis. Figure 2.9 illustrates the boiling crisis mechanisms for film boiling and dryout.

The critical heat flux and the onset of the boiling crisis have been shown to depend on a large variety of parameters. The most common are the flow rates, vapor mass fraction,

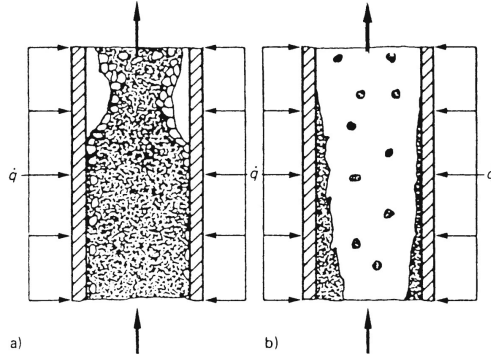


Figure 2.9: Boiling crisis mechanisms. (a) Film boiling (DNB). (b) Dryout (Mist flow). Reproduced from [34] with permission from Springer Nature.

system pressure, channel diameter, fluid properties such as the saturation temperature, and fluid subcooling.

A detailed prediction of the critical heat flux for flow boiling in tubes can be found in look-up tables. The 1995 Groeneveld look-up table for critical heat flux in tubes is designed to provide CHF values for 8 mm tubes at discrete values of pressure, mass flux, and dryout quality covering a wide range of pressures, mass fluxes, and vapor mass fractions [27]. It is based on an extensive data base of CHF values obtained in tubes with a vertical upward flow of boiling water.

Figures 2.10(a)–(b) show the critical heat flux versus the vapor mass fraction for flow boiling of water based on data from the 1995 look-up table for an inner tube diameter of 8 mm. Figure 2.10(a) compares three different mass fluxes at a system pressure of 1 bar. In Figure 2.10(b), three different system pressures are compared at a mass flux of $G = 50 \text{ kg}/(\text{m}^2\text{s})$. Negative values of the vapor mass fraction indicate subcooled fluid with its corresponding subcooled enthalpy relative to the enthalpy of vaporization. The graphs show the significant influence of the vapor quality on the critical heat flux. The CHF is highest for subcooled liquids, but decreases rapidly with increasing vapor fractions. For very small or negative vapor mass fractions, there is little effect of mass flux or pressure on CHF. The same is true for high vapor qualities close to 1. The CHF reaches values of up to $400 \text{ W}/\text{cm}^2$ at $\dot{x} = -0.1$, whereas it drops to zero at $\dot{x} = 1$. In between, higher flow rates and higher system pressure lead to an increase in CHF.

An empirical correction factor has been established to extent the CHF look-up tables to tube diameters other than 8 mm [34]:

$$\frac{\dot{q}_{crit,d}}{\dot{q}_{crit,d=8mm}} = \left(\frac{d}{8 \cdot 10^{-3}} \right)^{-0.5} \quad \text{for } 3 \cdot 10^{-3} \text{ m} \leq d \leq 25 \cdot 10^{-3} \text{ m} \quad (2.43)$$

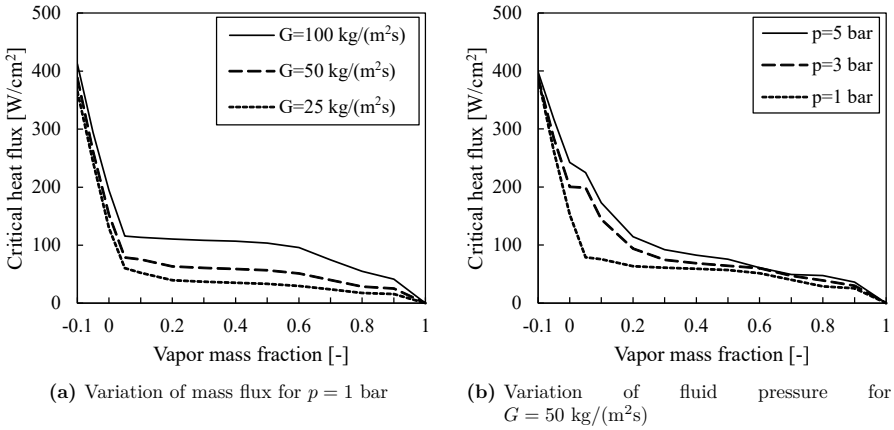


Figure 2.10: Critical heat flux versus vapor mass fraction for flow boiling of water. Tube inside diameter 8 mm. Own illustration based on data from the 1995 look-up table for critical heat flux in tubes [28].

Based on the correction factor, the impact of the channel diameter on the CHF can be estimated. For a channel diameter of 3 mm, the CHF increases by 63%, while for a channel diameter of 25 mm, the CHF decreases by 43%. Thus, the selected channel diameter has a large impact on the CHF value. Smaller channel diameters result in higher CHF values.

In horizontal tubes, boiling crisis typically occurs at lower vapor mass fractions than in vertical tubes due to phase separation caused by gravity. Under these conditions, the boiling crisis sets in at the top side of the channel, while the bottom wall remains wetted. However, for Froude numbers Fr higher than 10, the inclination of the tube has no effect on the occurrence of the boiling crisis [34]. Therefore, for estimation purposes and due to the lack of empirical data, the CHF values for vertical upward flow can be applied to horizontal flow channels.

2.5.4 Heat transfer in film boiling, dryout, and transition boiling regimes

Compared to the nucleate boiling regime, much less research data and publications are available on heat transfer in the film boiling, dryout, and transition boiling regimes. This is because, for most engineering applications, nucleate boiling is the desired state due to its high heat transfer rates, and evaporators are designed to operate well below CHF. Film boiling, dryout, and transition boiling are usually dismissed as undesirable operating states and therefore have less technical relevance than nucleate boiling and CHF.

Film boiling occurs at negative or low vapor weight fractions ($\dot{x} \leq 0.2$) [46] and at excess temperatures beyond the Leidenfrost point (see Table 2.4). In film boiling, a vapor film

completely separates the hot wall from the liquid fluid phase, so there is no direct contact between them. Thus, heat is transferred via conduction and radiation through the vapor film. For mass fluxes smaller than $500 \text{ kg}/(\text{m}^2\text{s})$, the literature recommends calculating the film boiling heat transfer coefficient using the same methods as for film boiling under pool boiling conditions [46].

For pool boiling, Bromley described the following relationship between the film boiling heat transfer coefficient α_{film} and the heat transfer by conduction α_{cond} and by radiation α_{rad} [10, 24]:

$$\alpha_{film} = \alpha_{cond} + \alpha_{rad}[0.75 + 0.25(1 + 2.62\alpha_{cond}/\alpha_{rad})^{-1}] \quad (2.44)$$

with $0 < \alpha_{rad} < 10 \alpha_{cond}$ as application range. The heat transfer coefficient by radiation can be derived using Equation 2.14.

The heat transfer coefficient by conduction through the vapor film in stable film boiling is expressed by [10, 24]:

$$\alpha_{cond} = K_f(L\Delta T)^{-1/4}(\lambda_v^3\rho_v\Delta h\Delta\rho g/\mu_v)^{1/4} \quad (2.45)$$

where λ_v , ρ_v , μ_v are the thermal conductivity, density, and dynamic viscosity of the vapor at the average temperature of the vapor film $T_m = 0.5(T_w + T_l)$. The wall superheat is defined as $\Delta T = T_w - T_l$. The differences in density and specific enthalpy between vapor and liquid are $\Delta\rho = \rho_l - \rho_v$ and $\Delta h = h_l - h_v$. For horizontal cylinders, the characteristic length L is the tube diameter d_h and the empirical parameter $K_f = 0.62$ [10].

A similar calculation method is recommended in Incropera [39], which results in slightly higher values for the film boiling heat transfer coefficient. In addition to the available empirical correlations, Groeneveld presented a look-up table for fully developed film-boiling heat transfer for steam-water inside vertical tubes [28].

For film boiling in subcooled flows, the following equation is recommended in literature for calculating the effect of liquid subcooling on the heat transfer coefficient [26, 46]:

$$\alpha_{sub} = \alpha_0(1 + 0.025\Delta T_{sub}) \quad (2.46)$$

α_0 is the heat transfer coefficient in saturated film boiling, and ΔT_{sub} is the temperature difference between the saturation temperature and the subcooled temperature. Consequently, subcooled flow results in higher heat transfer coefficients, even in stable film boiling.

Dryout occurs at high vapor fractions ($\dot{x} > 0.2$) in combination with excess temperatures beyond the Leidenfrost point [46]. This results in a mist flow containing liquid droplets

that do not contact the hot wall. The physical mechanisms of post-dryout droplet flow under thermodynamic nonequilibrium are very complex and less relevant to this work. Therefore, no further details on the subject are given here. A comprehensive review can be found in the literature, for example, in the work of Katsaounis and Kind [46].

For excess temperatures between the critical heat flux (ΔT_{crit}) and the Leidenfrost point (ΔT_{min}), transition boiling, named partial film boiling, occurs. This is characterized by alternating conditions between film and nucleate boiling. Transition boiling is less relevant to engineering than the other boiling regimes. Therefore, only few studies in the literature are related to transition boiling.

However, the heat flux and heat transfer coefficient can be estimated based on the fact that, on a double-logarithmic scale, the boiling curve is approximately linear in the transition boiling regime. Consequently, the following relation can be used to predict the heat flux $\dot{q}_{TB}$ in the transition boiling regime depending on the corresponding temperature difference ΔT_{TB} , if conditions at CHF ($\dot{q}_{crit}$, ΔT_{crit}) and the Leidenfrost point ($\dot{q}_{min}$, ΔT_{min}) are known [46]:

$$\ln \left(\frac{\dot{q}_{TB}}{\dot{q}_{min}} \right) = \ln \left(\frac{\dot{q}_{crit}}{\dot{q}_{min}} \right) \cdot \frac{\ln(\Delta T_{min}/\Delta T_{TB})}{\ln(\Delta T_{min}/\Delta T_{crit})} \quad (2.47)$$

Chapter 5 provides a further discussion and application of the reviewed context. It also presents a new heat transfer calculation model derived from the experimental results of this thesis, existing correlations, and methods from the literature.

In engineering applications, such as the design of heat exchangers, a practical method for improving poor heat transfer coefficients caused by boiling crises or preventing drywall conditions is using tube inserts such as hiTRAN wire matrix elements [20]. However, tube inserts are not always an effective solution and not always applicable due to the need for a custom fit. In addition, a boiling crisis still cannot be avoided at large excess temperatures.

2.5.5 Condensation

As in the case of boiling, condensation processes are widely used in many technical applications. Condensation heat transfer is characterized by large heat transfer rates with small temperature differences due to the involved phase change from vapor to liquid. Condensation occurs when the temperature of a wall surface is lower than the saturation temperature of the adjacent vapor. There are basically two modes of condensation heat transfer: Film condensation where a continuous liquid film is present, and dropwise condensation. For engineering applications, the most common type is film condensation. In this thesis, condensation is part of the boiling discharge concepts presented in Chapters 3 and 5. As the focus of this work is on the boiling heat discharge, and condensation is considered to be achieved with state-of-the-art heat exchangers, the physical background

of condensation is not further presented. Basic literature on the subject includes the books by Collier and Thome [14], Kandlikar [43], Incopera [39], and VDI Heat Atlas [42].

2.6 Thermal resistance and thermal contact resistance

The thermal resistance R in general is defined by the ratio of the temperature difference ΔT to the transferred heat $\dot{Q}$:

$$R = \frac{\Delta T}{\dot{Q}} \quad (2.48)$$

In a mPCM thermal energy storage, the hot storage material and the heat transfer fluid are separated by a wall of one or more layers. The process of steady state heat transport from the hot storage material to the cold heat transfer fluid through the separating walls is associated with the overall heat transfer coefficient U and characterized by the overall thermal resistance R_{th} :

$$R_{th} = \sum_{i=1}^n R_{th,i} = \frac{\Delta T}{\dot{Q}} = \frac{1}{UA} \quad (2.49)$$

The overall thermal resistance varies together with the heat transfer coefficient h of the heat transfer fluid, the thermal conductivities k of the storage material and solid layers, and the thermal contact resistances at the interfaces of adjacent layers. Therefore, the overall thermal resistance can be written as:

$$R_{th} = \sum R_h + \sum R_{cond} + \sum R_c \quad (2.50)$$

$$\text{with } R_h = \frac{1}{hA}, \quad R_{cond} = \frac{l}{kA}, \quad R_c = \frac{T_A - T_B}{\dot{Q}}$$

R_h is the thermal resistance of the heat transfer fluid, R_{cond} is the thermal resistance related to heat conduction, R_c is the thermal contact resistance, A the area, and l the length for conduction.

Thermal contact resistance R_c plays a significant role in the thermal discharge capability of the overall storage system. Figure 2.11 illustrates the temperature drop across two solid rough surfaces. The thermal contact resistance between adjacent layers is basically due to surface roughness effects. Even for two seemingly flat surfaces under high pressure, the actual contact area is only about 1–3% of the nominal contact area [8, 126]. The gaps are typically filled with air and the heat transfer is due to conduction across the actual

contact area and to conduction and radiation across the gaps. The small actual contact area leads to the thermal contact resistance.

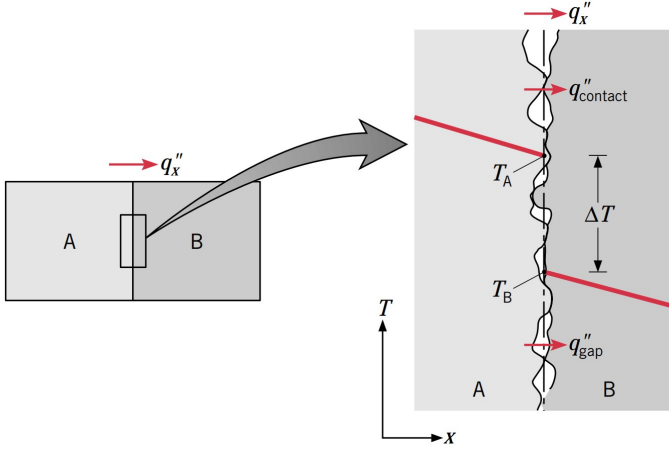


Figure 2.11: Temperature drop due to thermal contact resistance, reproduced from [39] with permission from John Wiley & Sons

The thermal contact resistance between solid parts is mainly dependent, beside others, on the materials of the contacting surfaces, the surface roughness of the manufacturing process, and the contact pressure. Typical R_c values for stainless steel are 0.7–4.0 cm²K/W for a contact pressure of 10,000 kN/m² and 6–25 cm²K/W for a contact pressure of 100 kN/m² [39]. Typical R_c values for aluminum are 0.2–0.4 cm²K/W for a contact pressure of 10,000 kN/m² and 1.5–5.0 cm²K/W for a contact pressure of 100 kN/m² [39].

The thermal contact resistance may be reduced by increasing the joint pressure or by reducing the roughness of the contacting surfaces. Furthermore, the contact resistance may be reduced by selecting an interfacial fluid of high thermal conductivity, such as thermally conductive paste, or by selecting an interfacial foil of high thermal conductivity in combination with high joint pressure. In the development of thermo electric generators (TEGs), the contact resistance is significantly reduced by using a very thin graphite foil under high joint pressure [32, 53]. In contrast to ordinary foils, graphite foils are capable for high temperature applications such as applied in this work.

Even though theories have been developed to predict R_c , the most reliable results have been obtained through experimentation [39]. Comprehensive reviews of thermal contact resistance are provided by Madhusudana et al. [68], Snaith et al. [99], and Incropera [39].

The thermal contact resistance between a plain solid of AlSi₁₂ and the ceramic matrix composite C/C–SiC was reported to decrease asymptotically with increasing temperature and increasing pressure based on an experimental study [102]. For temperatures of

150 °C and pressures over 3 MPa, the contact resistance was observed to be smaller than 1 cm²K/W, whereas it increased to about 2 cm²K/W for low contact pressures [102].

In literature, there is little or no information available about the temperature dependent thermal contact resistance of AlSi₁₂ with a contacting container housing in a real prototype environment considering the phase change of the storage material. Therefore, Chapter 4 deals with the qualitative and quantitative investigation of the thermal contact resistance in the full operating temperature range relevant to the thermal storage system.

2.7 Heat transfer fluids and current technical limitations

A heat transfer fluid (HTF) acts as a carrier of thermal energy and is needed for thermal discharge of the storage system and further distribution in the vehicle system in this work. There is a wide variety of different HTFs in literature, each having their advantages, drawbacks, and limitations. In this work, the operating temperature covers a wide range between 0–700 °C. Desired characteristics of an all-liquid HTF are therefore low melting point, high boiling point, low corrosion with components in contact with the HTF, low viscosity, high thermal conductivity, high heat capacity, and low cost.

HTFs for high-temperature applications can be classified into five main groups based on the fluid: (1) air and other gases, (2) water/steam, (3) thermal oils and organics, (4) molten-salts, and (5) liquid metals. Figure 2.12 gives an overview about the operating temperature range of different high-temperature HTFs.

Air and other gases

Air and other gases are commonly used in numerous engineering applications as HTF. Air is in particular easily available, simple to handle, and a low cost HTF. Air and other gases have the advantage of a wide operating temperature range, as the boiling point is very low. Therefore, air and other gases, such as CO₂ and nitrogen, are thoroughly capable of operating in a temperature range of 0–700 °C. However, the low density and heat transfer coefficient of single-phase gases are a drawback, especially when it comes to small temperature differences between HTF and heat source.

Water/Steam

Water is the most widely used and best-known HTF. It is used in numerous applications as liquid coolant HTF due to its excellent heat transfer characteristics. Water is an ideal heat transfer fluid for the temperature range from 0 °C to 100 °C. When it comes to high-temperature operation, the use of water is limited due to its boiling point of 100 °C at atmospheric pressure. The boiling point can be increased up to 373.9 °C by increasing the pressure up to the critical pressure p_{crit} of 220.64 bar [42]. Exceeding the boiling point results in evaporation and creation of steam. A two-phase system including evaporation

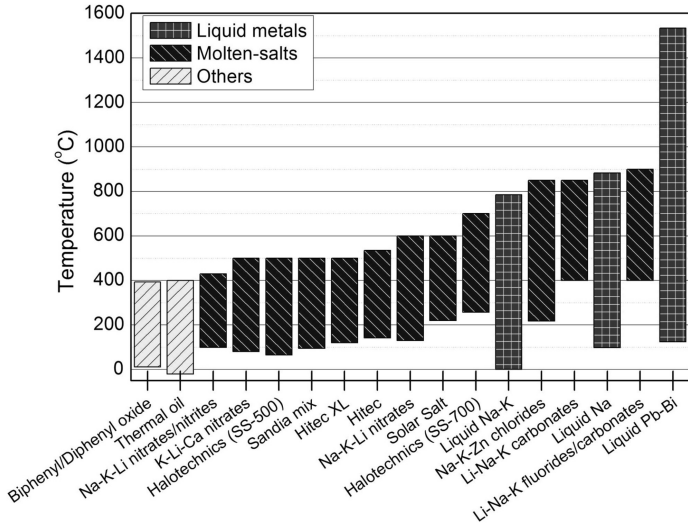


Figure 2.12: Operating temperature range of different high-temperature HTFs, adapted from [118]

and condensation poses additional engineering challenges, as described in the previous sections. Corrosion of water and high temperature steam in several different alloys has been reported in the literature and requires sufficient handling of corrosion inhibitors and material selection.

Thermal oils and organics

Thermal oils and organics are widely used in heat transfer applications, but have an upper temperature limit of approximately 400 °C, as seen in Figure 2.12. At low temperatures, these HTFs have the drawback of high viscosities, which are disadvantageous for good heat transfer.

Molten-salts

Molten-salts have been widely studied as HTF due to their high working temperature (more than 500 °C) and heat capacity, low vapor pressure, and good thermal and physical properties at elevated temperatures [118]. Molten-salt based HTFs are commonly used in modern concentrated solar power systems [118]. However, the use of molten salt is associated with the corrosion and erosion of piping and heat exchanger material [95]. Moreover, molten-salts are not suitable at lower operating temperatures, as it can be seen in Figure 2.12, due to their relatively high melting point. Many molten-salt HTFs are freezing at temperatures below 100–200 °C.

Liquid metals

Liquid metal HTFs are promising candidates for high temperature applications due to their high boiling points, high thermal conductivities, and great heat capacities. Using liquid metals, high heat transfer rates can be achieved. In the temperature range from 500 °C to 800 °C, the alkali metals sodium Na and potassium K and alloys of both metals are a preferred liquid heat transfer media [121]. Sodium is commonly used as coolant in nuclear reactors [71]. As seen in Figure 2.12, the liquid alloy NaK is the only HTF capable of being liquid in the operating temperature range between 0–700 °C. The melting point of NaK78 is -11 °C [121]. However, NaK is highly reactive with oxygen and water. Contact of Na with water can lead to explosion and flame formation [121]. These hazardous safety concerns are a major drawback when it comes to a vehicle application.

Based on the current technical limitations of the heat transfer fluids discussed, an all-liquid HTF concept for thermal discharge of the storage system is not practical. Alternatives to an all-liquid HTF concept are on the one hand, using a single-phase gas, or on the other hand, the evaporation of a liquid working fluid. Chapter 3 covers the concept development and further HTF selection.

2.8 Conclusions from the state of the art

The following main conclusions were drawn after studying the literature and reviewing the state of the art as well as the available calculation methods:

1. The application of high-temperature mPCM thermal energy storage for heat supply and cabin heating in mobile applications is new territory.
2. Solutions for the thermal discharge of high-temperature mPCM storage in the literature mostly use only latent heat and not sensible heat. There is a lack of heat transfer concepts and proven technologies for discharge over a wide operating temperature range.
3. An all-liquid heat transfer concept is not practical due to limitations of currently available heat transfer fluids.
4. The thermal contact resistance at the AlSi₁₂ container interface in a real storage environment is unknown, but it is crucial for the performance and simulation of the storage system.
5. The underlying physics of the thermal discharge process are complex, particularly in cases involving an evaporating working fluid due to flow boiling and the onset of boiling crises.

The current state of the art indicates a need for the development of new concepts and technical solutions. These should be supported by simulations and experiments. They should also improve the understanding of the physical characteristics of the thermal discharge.

3 Development of concepts for thermal discharge

This chapter focuses on developing appropriate heat extraction concepts for high-temperature thermal energy storage (TES) systems based on mPCMs in vehicle applications. The methodical concept development procedure is based on the VDI guideline 2221 [116]. Then, based on that and the state of the art described in the previous chapter, a conceptual approach and various heat extraction concepts are derived. These concepts are then evaluated, and two promising concepts are selected for further prototyping and investigation in the following chapters.

3.1 Storage system, boundary conditions, and requirements

In order to develop thermal discharge concepts, it is important to identify the storage system's main components and the relevant boundary conditions and requirements that affect concept development.

Geometric boundary conditions

One important boundary condition is the geometric arrangement of the discharge unit within the mPCM storage system. After reviewing the literature, two basic principles emerged as reasonable ways to arrange the discharge unit within the mPCM:

- (a) Solidification leading to a pressure boundary
- (b) Gravity assisted directional solidification

Figure 3.1 illustrates the two basic principles. Figure 3.1(a) illustrates the first principle, which takes advantage of the shrinkage effect that occurs when the mPCM solidifies and cools down. The discharge unit is centrally located within the storage material, and the discharge fluid ideally flows from the bottom to the top as it extracts heat. The shrinkage effect creates a pressure boundary between the storage material and the discharge unit when the mPCM is in its solid state.

Figure 3.1(b) illustrates the second principle, which takes advantage of gravity to avoid void defects in the mPCM close to the discharge unit. Because directional solidification

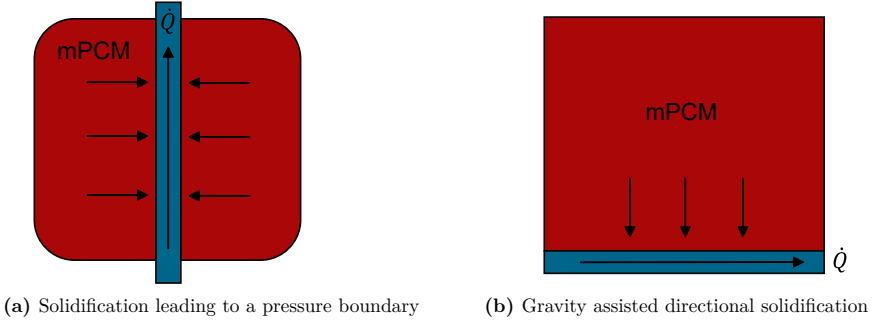


Figure 3.1: Principle geometric arrangements of discharge unit within the mPCM

starts at the colder bottom and moves from bottom to top, shrinkage cavities at the contact surface with the discharge unit are unlikely [16, 57, 60]. Otherwise, void defects may reduce the effective contact area and deteriorate the heat transfer capability.

Due to promising preliminary tests at DLR by Dicke and Kraft [16, 57] and a patent application on the topic [60], the principle of directional solidification was set as geometrical boundary condition for this thesis. This boundary condition was determined in the DLR research group before this thesis started. However, the main findings and conclusions from the upcoming chapters can be applied to different geometrical arrangements.

Figure 3.2 illustrates the schematic of the basic storage system arrangement, based on the principle of gravity assisted directional solidification. The storage system uses the mPCM (AlSi_{12}) storage material, which is accommodated in a box-shaped container. The basic storage system arrangement is presented in a patent application by Kraft et al. [60], which is considered as a boundary condition in this thesis.

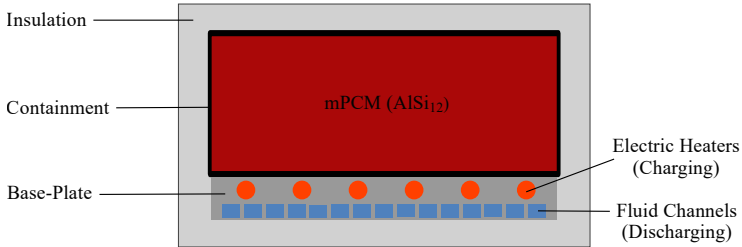


Figure 3.2: Schematic of the basic storage system arrangement considered as a boundary condition in this thesis. Own illustration derived from [57, 60].

Due to the liquid state of the mPCM, an enclosure is necessary to prevent effusion of the storage material. Consequently, direct contact heat transfer between the mPCM and the heat transfer fluid is not possible. A base plate is installed on the bottom. The base plate

contains electric heaters for charging heat and fluid channels for discharging heat. The selection of materials for the storage system components, such as the containment, base plate, and insulation, is detailed in Chapters 4 and 5.

As discussed in Subsection 2.2.2, liquid aluminum-silicon alloys are highly corrosive to metals such as steel substrates. Therefore, the containment is separated from the base plate to allow testing of different containment materials and configurations and to simplify the engineering and manufacturing process.

Storage material and temperature boundary conditions

In this thesis, the eutectic aluminum silicon alloy (AlSi_{12}) was selected as storage material for all further investigations. According to Rawson et al. [88], it is an ideal storage material candidate due to its latent heat of fusion, energy density, and cost, as discussed in Subsection 2.2.2. The physical properties of AlSi_{12} are detailed in Table 2.1 and additional relevant temperature-dependent material properties are provided in Appendix A.

Based on the selected storage material, the upper temperature level is 650 °C and the phase change temperature level is 577 °C. The lower temperature level is defined as ambient temperature at 15–20 °C. As shown in Figure 1.2, the discharge is highly transient with a wide operating temperature.

Heat output requirement

The energy demand for cabin heating in a vehicle depends on many parameters, as discussed in Section 2.1. For investigation and further prototyping, a heat output requirement of 1 kW is defined for the bench-scale prototype, and 10 kW for the application-scale prototype. Chapter 4 provides a more detailed description of the bench-scale and application-scale prototypes. The numbers do not correspond to a specific vehicle type, but they are in the order of magnitude for a bus application. A rough estimation of the heating demand is given in Section 2.1.

Functional requirements

Based on the mobile application, the following functional requirements are defined for further concept development:

- **Performance:** Achievement of the required heat output in a large operating temperature range
- **Efficiency:** Low electric power consumption of the fan or pump used for discharge in relation to extracted heat output; Low heat loss to the environment
- **Flexibility for heat distribution:** Potential to transfer heat to a vehicle distribution system, such as an air or a liquid distribution system

- **Controllability:** Functionality to control the heat flow in the operating temperature range; Heat extraction only on demand including an "On/Off" feature
- **High storage utilization:** Functionality to utilize a large amount of the stored latent and sensible heat (directly related to performance requirement)

System-related and other requirements

Based on the mobile application, the following system-related and other requirements are defined for further concept development:

- **Compact and lightweight design**
- **Simplicity:** Engineering and design as simple as possible
- **Practicability:** Practical feasibility for prototype development and future real-world vehicle application
- **Thermomechanical load:** Limitation of the induced temperature differences in the storage system caused by thermal discharge; Flexible movement of components if possible
- **Non-hazardous** heat transfer fluids
- **No major safety concerns**
- **Maintenance** and ease of **manufacturing** and **installation**
- **Competitive cost** of the discharge/storage system

3.2 Impact factors on discharge performance

The thermal discharge performance is one of the important functional requirements for the concept development. Therefore, a deeper analysis of the heat transfer from the storage material to the discharge fluid is necessary. For the analysis, the thermal resistances and the temperature profile are considered. Figure 3.3 shows an exemplary temperature profile and thermal resistances for heat transfer from mPCM to the discharge fluid, including the container and base-plate walls, as well as the thermal contact resistances at the mPCM-container interface and at the container-base-plate interface.

The discharge performance is strongly influenced by the overall thermal resistance R_{th} of the thermal network. According to Equation 2.49, it is defined that $R_{th} = \Delta T / \dot{Q}$, respectively $R_{th} = 1 / UA$, with ΔT the temperature difference between mPCM and fluid bulk temperature and U the overall heat transfer coefficient. The overall thermal resistance is the sum of the individual thermal resistances. These are caused by conduction (R_{cond}) in the mPCM, container wall, and base-plate wall, as well as by thermal contact resistances at the interfaces (R_c) and the resistance due to convection heat transfer in

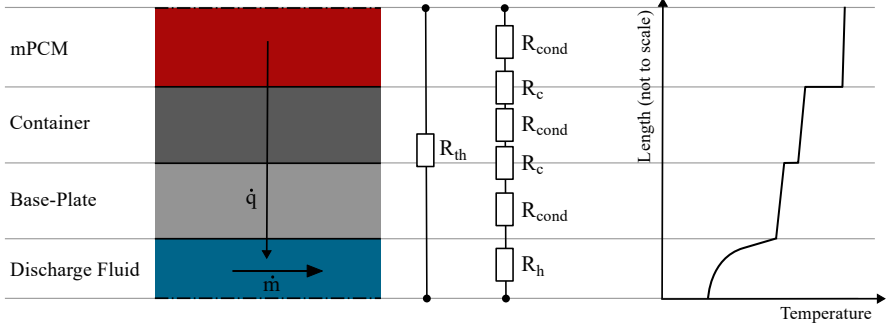


Figure 3.3: Exemplary temperature profile and thermal resistances for heat transfer from mPCM to the discharge fluid

the fluid channels (R_h). The smaller the thermal resistances, the better the heat transfer capability from mPCM to the discharge fluid.

The thermal resistance due to conduction inside the mPCM is neglectable compared to the total thermal resistance R_{th} because of the high thermal conductivity of the storage material. As discussed in Section 2.2.2, the thermal conductivity of AlSi_{12} is $170 \text{ W}/(\text{mK})$ in its solid state at 300°C . This accounts for a thermal resistance of $R_{cond} = 1/17000 \text{ m}^2\text{K}/\text{W}$ per 1 cm mPCM length in direction of heat transfer.

A large impact on R_{th} is expected from the thermal contact resistance R_c at the interfaces between mPCM-container and container-base-plate. The thermal contact resistance at the mPCM-container interface is so far unknown and target of investigations in this thesis in Chapter 4. For the thermal contact resistance at the container-base-plate interface, a common solid-solid flat surface contact resistance can be assumed. A thermal contact resistance of $R_c = 2.5 \text{ cm}^2\text{K}/\text{W}$ corresponds to $R_c = 1/4000 \text{ m}^2\text{K}/\text{W}$.

The thermal resistance due to conduction in the container and base-plate wall is strongly dependent on the selected material and the thickness of the wall. For stainless steel alloys with a conductivity of about $15 \text{ W}/(\text{mK})$, the thermal resistance is $R_{cond} = 1/1500 \text{ m}^2\text{K}/\text{W}$ per 1 cm length in direction of heat transfer, while for nickel with a conductivity of around $70 \text{ W}/(\text{mK})$, it is only $R_{cond} = 1/7000 \text{ m}^2\text{K}/\text{W}$ per 1 cm.

The effective heat transfer coefficient of the discharge fluid has a large impact on the discharge capability and the overall thermal resistance. For an effective heat transfer coefficient of $h = 50 \text{ W}/(\text{m}^2\text{K})$, it follows $R_h = 1/50 \text{ m}^2\text{K}/\text{W}$ (single phase air flow without fins). For $h = 1000 \text{ W}/(\text{m}^2\text{K})$ (compressed gas with CO_2 and high-performance fin structure), it follows $R_h = 1/1000 \text{ m}^2\text{K}/\text{W}$. For $h = 10000 \text{ W}/(\text{m}^2\text{K})$ (nucleate boiling of water), it follows $R_h = 1/10000 \text{ m}^2\text{K}/\text{W}$. Consequently, there is a factor of 200 between single phase air flow without fins and nucleate boiling heat transfer of water.

In addition to the heat transfer coefficient, the density and heat transport capacity of the fluid has a big impact on the discharge performance. The heat transport capacity

is defined by the specific heat capacity in single-phase flow and the specific enthalpy of vaporization in boiling two-phase flow. First, the density and heat transport capacity impact the temperature rise of the fluid in the direction of travel through the discharge section. Second, they impact the required volume of the fluid channels and therefore the size of the heat transfer section.

The discussion of impact factors on discharge performance assumed adiabatic boundary conditions and neglected heat loss to the environment. While low thermal resistance is beneficial for heat transfer from the mPCM to the discharge fluid, high thermal resistance is necessary to insulate the storage system and minimize heat loss.

3.3 Conceptual approach

According to the state of the art, a number of concepts for the thermal discharge of high-temperature heat storage systems have been published in the literature, as discussed in Subsection 2.2.4. However, most of these concepts utilize only the high-temperature latent heat of the storage material. Consequently, they operate nearly isothermally with respect to thermal discharge and are not designed for a vehicle application. This is why most existing concepts cannot be applied to high-temperature mPCM thermal energy storage for vehicle heat supply. As such a storage system is new territory in a vehicle application, a general conceptual approach is needed in the first step.

Figure 3.4 illustrates the basic conceptual approach for the developed thermal discharge. This approach follows three principal paths. The first path (I) works with direct air discharge. As air is the final transport fluid that transfers heat into the vehicle's cabin, using air as the heat transfer fluid (HTF) directly for heat extraction from the mPCM storage is the simplest approach. The second path (II) uses an intermediate working fluid to extract storage heat, which is then transferred to the air via a heat exchanger for further distribution. The third path (III) also uses an intermediate working fluid to extract heat from the storage and transfers it to the vehicle's liquid coolant circuit via a heat exchanger for further distribution. Finally, heat is transferred to the cabin air via the air/water glycol heat exchangers of the vehicle's coolant circuit. The author of this thesis has recently highlighted the impact of a wide operating temperature range on designing heat transfer systems for mPCM thermal storage in mobile applications [77].

Direct air discharge (I), which uses ambient air at atmospheric pressure, has the advantages of low complexity and high functionality, but shows low heat transfer coefficients. In contrast, discharge via an intermediate working fluid (II+III) offers higher heat transfer coefficients and good functionality, but is more complex.

Based on the literature review presented in Section 2.7, an all-liquid heat transfer concept is not practical in the operating temperature range of approximately 0 °C to 700 °C due to limitations of currently available heat transfer fluids. Although NaK is a liquid within the required operating temperature range, it is not considered further due to its hazardous

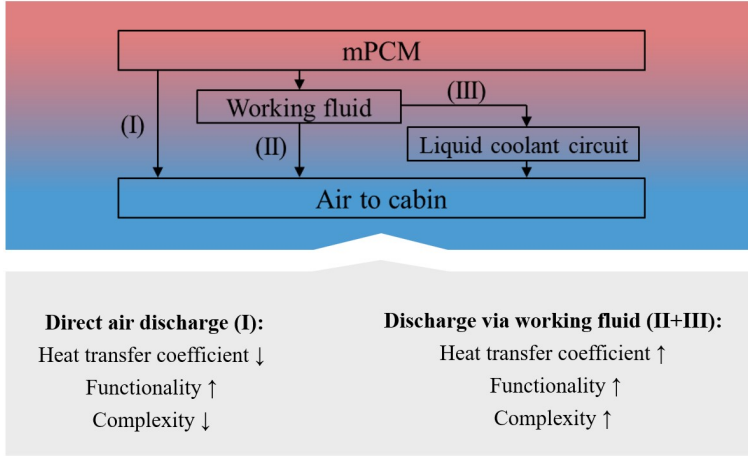


Figure 3.4: Basic conceptual approach for thermal discharge, adapted from Nees et al. [77]

safety concerns, which are critical for vehicle applications. Alternatives to an all-liquid HTF concept are using a single-phase gas as the HTF or evaporating a liquid working fluid.

The use of heat pipes is generally only applicable within a small operating temperature range, close to the saturation temperature of the working fluid. Therefore, conventional heat pipes are not suitable for the wide operating temperature range discussed in this thesis.

Using a gaseous fluid, such as ambient air or a compressed gas like CO_2 or nitrogen, allows for a much wider permissible temperature range. Thus, the large operating temperature range in this study is not a limiting factor. However, gases typically have a much lower heat transfer capacity compared to all-liquid HTFs or HTFs undergoing a phase change. This is especially true at low temperature differences due to the relatively low heat transfer coefficient of single-phase gas flow. Nevertheless, this can be compensated to some extent by installing a larger heat transfer surface area and volume.

In comparison to single-phase heat transfer systems, it is possible to reach heat transfer rates orders of magnitude higher with boiling heat transfer. Therefore, the high heat transfer coefficients associated with boiling make it attractive for application in a heat extraction concept. However, the complex two-phase nature of boiling poses a number of challenges, as discussed in Section 2.5. One of those challenges is that the nucleate boiling heat flux can only increase up to the critical heat flux. For higher temperature differences, a boiling crisis occurs due to film- and transition boiling, which deteriorates the heat transfer. As discussed in Section 2.5, the critical heat flux and the occurrence of the boiling crisis depend on various factors, including wall superheat, local vapor mass fraction, mass flux, pressure, and fluid properties.

The selection of an appropriate working fluid for evaporation and condensation is strongly influenced by the temperature level of the process and the saturation temperature of the working fluid. The required temperature level for transferring heat via a heat exchanger to the vehicle coolant circuit or to the air is approximately 50 °C to 80 °C. Ideally, the saturation temperature of the working fluid is slightly above. As water has a boiling point of 100 °C at atmospheric pressure, shows excellent heat transfer characteristics, is cheap, easily available, non-hazardous, and widely studied, it is considered as a suitable working fluid for the thermal discharge in this thesis. Another major benefit of using water is that pressurization or evacuation of the discharge system is not needed. Thus, the mechanical requirements for piping and components can be kept low.

3.4 Derivation of heat discharge concepts

Based on the conceptual approach presented, six different heat extraction concepts were developed and are presented in the following.

Direct air discharge (Concept 1)

Direct discharge of the TES using ambient air is the simplest and most obvious approach because only single-phase heat transfer is involved and air as the HTF is already the final heat transfer carrier to the cabin. In addition, the concept of using air for TES discharge has been proposed and studied by other researchers, as discussed in Subsection 2.2.4. The concept's purpose is to extract heat from the high-temperature mPCM storage across a broad operating temperature range and transfer it to the final transport medium, which is air distributed across the vehicle cabin. Figure 3.5 shows a schematic representation of this concept adapted to the boundary conditions of this thesis.

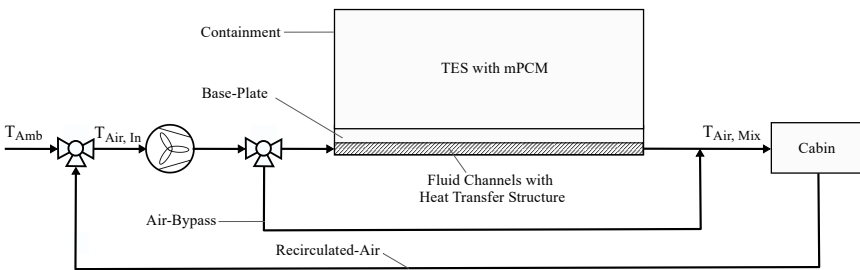


Figure 3.5: Schematic representation of the direct air discharge concept (Concept 1)

The thermal discharge is realized by forced convection of ambient air within the fluid channels of the heat transfer section, which is integrated into the TES. To compensate for the relatively low heat transfer coefficient of single-phase airflow, the heat transfer section has fins that increase the effective heat transfer area. A fan installed in the cool inlet line drives and controls the air flow. To regulate the temperature of the air entering

the cabin, the hot air exiting the TES is mixed with ambient air, which bypasses the TES via a line and control valve. This is necessary because the air entering the cabin should be limited to 40–70 °C for safety reasons. Cabin air can be recirculated via a connection to the discharge system inlet line using an additional control valve.

The advantages of this concept include high functionality and controllability of the heat extraction process across the full operating temperature range and low complexity. According to Equation 2.7, the temperature difference between the hot wall and the fluid bulk in an internal flow linearly impacts the convective heat transfer rate. Therefore, the thermal discharge performance at high temperature differences is promising for the proposed concept. However, at lower temperature differences, the discharge performance is expected to be insufficient.

Compressed gas cycle (Concept 2)

Figure 3.6 shows a schematic representation of the compressed gas cycle concept (Concept 2), illustrating its components and fluid channel arrangement. Thermal discharge is achieved through the forced convection of compressed gas within the fluid channels of the heat transfer section. To increase heat transfer, the heat transfer section has fins that increase the effective heat transfer area. The fluid channels and electric heaters are integrated into the base plate of the TES, as shown in Figure 3.6. The compressed gas flow is driven and controlled by a fan installed in the inlet line before the gas enters the hot TES. The extracted heat is then transferred via a heat exchanger to the vehicle coolant circuit for further distribution, as indicated in Figure 3.6. Alternatively, the extracted heat can be transferred directly to the cabin air via the heat exchanger.

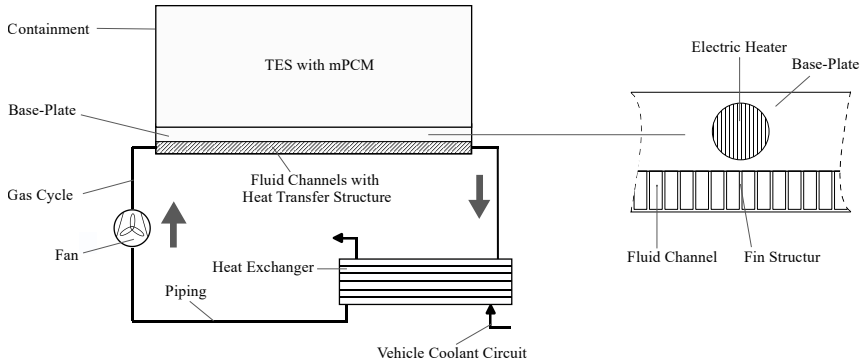


Figure 3.6: Schematic representation of the compressed gas cycle concept (Concept 2), showing the components and the arrangement of the fluid channels

The compressed gas cycle concept utilizes the characteristic that the heat transfer capacity of a single-phase gas flow increases with pressure. Based on the ideal gas law, the following relationship between pressure (p) and density (ρ) of a gas can be derived:

$$p = \rho R_s T \quad (3.1)$$

with R_s being the specific gas constant and T the gas temperature.

Consequently, under isothermal conditions, the pressure and density of a gas are directly proportional, meaning an increase in pressure leads to an equal increase in density. Increasing the gas pressure, and thus the density, has two main beneficial impacts on the heat transfer capacity:

- First, an increase in density leads to a proportional increase in mass flow ($\dot{m}$) for a given volume flow ($\dot{V}$), as $\dot{m} = \rho \dot{V}$. According to Equation 2.16, mass flow linearly impacts the heat transport capacity in single-phase flow.
- Second, an increase in density leads to an increase in the Reynolds number. According to Equation 2.10, fluid density linearly impacts Re. As the Nusselt number is a function of the Reynolds and Prandtl numbers ($Nu = f(Re, Pr)$), an increase in Re leads to an increase in Nu, and thus, the heat transfer coefficient increases.

Based on the above considerations, the higher the system pressure of the gas, the higher the resulting heat transfer capacity. However, increasing the system pressure goes hand in hand with higher mechanical requirements for all components in the compressed gas cycle, which is a major drawback of the concept.

As the compressed gas cycle operates in a closed system, gases other than air can be used as the working fluid. One promising working fluid is carbon dioxide (CO_2), as the product of density (ρ) and heat capacity (c_p) is higher than that of air. At a pressure of 5 bar and a fluid temperature of 100 °C, the product (ρc_p) is 41% higher for CO_2 than for air at the same pressure. At a fluid temperature of 600 °C, the product is 63% higher. This enables an improved heat transfer capacity. The fluid properties used for this comparison are based on the literature [42].

The advantages of this concept include high functionality and controllability of the heat extraction process across the full operating temperature range, as well as the ability to transfer extracted heat into the vehicle's coolant circuit. As the working fluid is single-phase, as in Concept 1, there is no need to consider complex two-phase flow characteristics. However, as the system is pressurized, engineering and component selection become more challenging.

Multi-stage boiling heat extraction (Concept 3)

According to Table 2.4, the temperature difference for the transition boiling regime is 30–120 K and for the film boiling regime is ≥ 120 K when using water as the heat transfer fluid. These temperature differences are easily exceeded during discharge of the TES. For water at atmospheric pressure, the boiling temperature is 100 °C, and the operating

temperature of the mPCM storage is up to 650 °C. Consequently, a significant impact due to the boiling crisis is expected over a wide operating temperature range if a direct contact boiling heat extraction concept is used, such as Concept 4. Besides performance losses due to the boiling crisis, controllability of the discharge, high thermomechanical loads due to fluctuating behavior during transition boiling, and large temperature differences are problematic.

In order to avoid the negative impact of the boiling crisis at high storage temperatures, Concept 3 is based on a multi-stage evaporator for thermal discharge. As the concept follows a novel approach, a patent application has been filed for this principle by the author [74]. The purpose of the concept is to extract heat of a high temperature storage source in a wide operating temperature range and transfer the heat to a coolant circuit which operates at a significant lower temperature level than the storage. Figure 3.7 illustrates the schematic of the concept with the relevant components. The concept idea is based on the evaporation of a working fluid for heat transfer in the thermal storage and its condensation for further heat transfer in a heat sink (heat exchanger/condenser). The working fluid is feed and controlled using a pump. Distilled water is considered to be an appropriate working fluid.

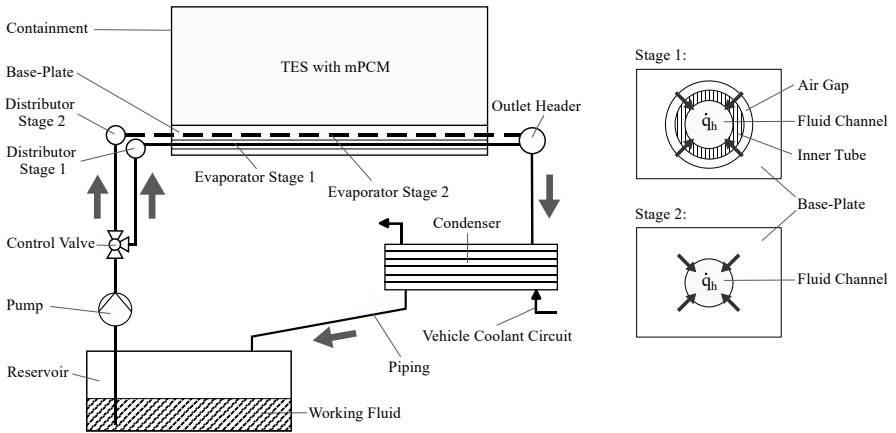


Figure 3.7: Schematic representation of the multi-stage concept (Concept 3) showing the components and the arrangement of the evaporator Stage 1 and Stage 2. The figure illustrates a coupled operation of Stages 1 and 2. Reproduced from Nees et al. [80].

The evaporator consists of two distinct boiling stages, which are integrated in the base plate of the thermal storage system. In Stage 1, the tubes are mechanically decoupled from the storage. The fluid channels are located in the inner tubes, which are inserted into holes in the base plate. A small gap between the base plate and the inner tube creates a thermal contact resistance that reduces the potential heat flow. The objective is to prevent a boiling crisis. The heat from the storage is transferred primarily via conduction

and radiation to the inner tube across the gap. The gap is filled with ambient air and is open to the environment, which allows for a pressure balance with the atmosphere. Another design option is to fill the gap with other gases or solids, if beneficial for the design, which is not discussed further here.

In contrast, Stage 2 involves direct contact between the working fluid and the base plate. In this case, the drill holes in the base plate serve directly as the fluid channels. Heat extraction is realized by forced convection of flow boiling in the channels using a fluid pump. Figure 3.7 shows the arrangement of the evaporator Stages 1 and 2.

The condenser is used to cool the superheated steam, condense the vapor, and sub-cool the condensate. The inlet flow can be considered as two-phase rather than full steam. The condenser is located on a lower level than the evaporator. The heat is transferred to the vehicle coolant circuit through the condenser. Alternatively, the heat can be transferred directly to the cabin air via a heat exchanger. The condensate is returned to a reservoir by gravity. This serves as a vessel for the working fluid and as a buffer for pressure changes in the system. The reservoir is intended to hold the entire liquid system volume of the working fluid.

Figure 3.8 shows the schematic representation for start-up and shut-down operation. The illustration demonstrates that during start-up at high-temperatures, only Stage 1 is operational, while Stage 2 is not in operation. The system is controlled by the control valve and the feed pump. The heat extraction can be shut down by draining the working fluid completely into the reservoir. This is the case for instance during charging the storage with electric heaters. For mid and low temperature levels, the coupled operation of both stages is considered, as shown in Figure 3.7.

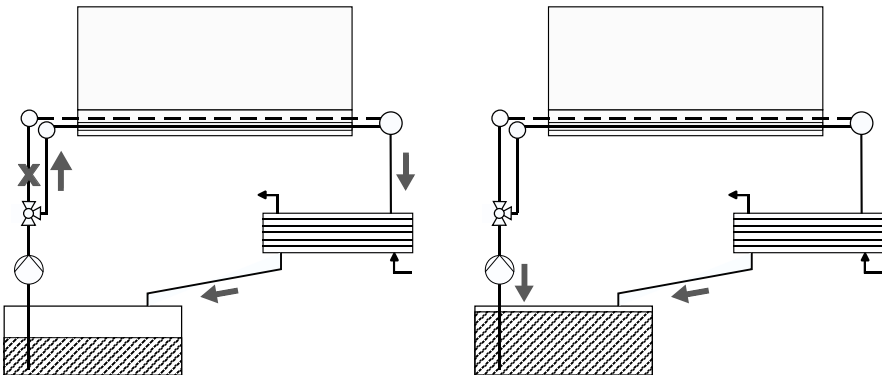


Figure 3.8: Schematic representation of start-up and shut-down process of Concept 3. Left: Start-up process at high TES temperature, with only Stage 1 being operational. Right: Shut-down process by drainage of the working fluid into the reservoir. Reproduced from Nees et al. [80].

With the novel multi-stage boiling concept, it is intended to avoid the boiling crises with the onset of film and transition boiling at large temperature differences between the heat source and the saturation temperature of the working fluid. As a consequence, an improved performance and controllability of heat transfer at large temperature differences is expected. In addition, the mechanical decoupling of Stage 1 and the limitation of the maximum heat flux is expected to reduce significantly the thermomechanical load during the start-up phase and during operation at high temperature differences. Nevertheless, the direct contact of Stage 2 allows high heat transfer rates to be achieved even at low temperature differences.

Operating and controlling both stages simultaneously is expected to achieve high specific heat output and controllability over a wide operating temperature range while significantly limiting the thermomechanical load. Additionally, the proposed concept enables the transfer of high-temperature storage heat to a state-of-the-art water-glycol vehicle cooling circuit.

Direct contact boiling heat extraction (Concept 4)

The direct contact boiling heat extraction concept (Concept 4) follows the same principle and arrangement as the multi-stage boiling concept (Concept 3), which is shown in Figure 3.7. However, Stage 1 is not installed. Thus, only the channels of Stage 2 are installed. Therefore, a control valve or additional fluid pump, as well as an additional distributor is not needed in the inlet line. The drill holes in the base plate serve as the fluid channels directly. Heat extraction is achieved through forced convection and flow boiling in the channels using a fluid pump. Consequently, the working fluid is in direct contact with the hot wall of the TES base plate. Distilled water is considered to be an appropriate working fluid. The extracted heat is condensed in a heat exchanger and transferred to the vehicle coolant circuit for distribution or, alternatively, transferred directly to cabin air via a heat exchanger.

Compared to the multi-stage concept (Concept 3), the direct contact boiling concept (Concept 4) has the advantage of simpler engineering and design with fewer components. However, disadvantages include lower expected performance at higher temperatures due to boiling crises, discharge controllability issues, and significantly higher thermomechanical loads. These relationships have already been explained alongside with Concept 3.

Resistance boiling heat extraction (Concept 5)

The resistance boiling heat extraction concept (Concept 5) follows the same principle and arrangement as the multi-stage concept (Concept 3), which is shown in Figure 3.7. However, Stage 2 is not installed. Thus, only the channels of Stage 1 are installed. Therefore, a control valve or additional fluid pump, as well as an additional distributor is not needed in the inlet line. The fluid channels are located in tubes inserted into drill holes in the base plate but are mechanically decoupled from it. The intention is to prevent a boiling crisis by placing a small gap between the base plate and the inner

tube. This gap creates thermal contact resistance, which reduces the potential heat flow. Consequently, there is no direct contact between the working fluid and the hot wall of the TES base plate. Heat extraction is achieved through forced convection and flow boiling in the channels using a fluid pump. As previously described, distilled water is considered to be an appropriate working fluid. The extracted heat is condensed in a heat exchanger and transferred to the vehicle coolant circuit for distribution or, alternatively, transferred directly to the cabin air via a heat exchanger.

Compared to the multi-stage concept (Concept 3), the resistance boiling concept (Concept 5) has the following advantages: simpler engineering and design, fewer components, and less effort required for controlling two coupled stages. However, the disadvantages are lower performance at lower storage temperatures. The underlying relationships have already been explained alongside with Concept 3.

Hybrid heat extraction (Concept 6)

Another approach to extract heat from a high-temperature storage source over a wide operating temperature range and avoiding the negative impact of boiling crises is the hybrid heat extraction concept (Concept 6). This concept couples the compressed gas cycle (Concept 2) with the direct boiling heat extraction (Concept 4). Figure 3.9 shows a schematic representation of the concept, illustrating the components and arrangement of the evaporator and gas channels. The hybrid concept follows a novel approach, and therefore, a patent application has been filed for this principle by the author [75].

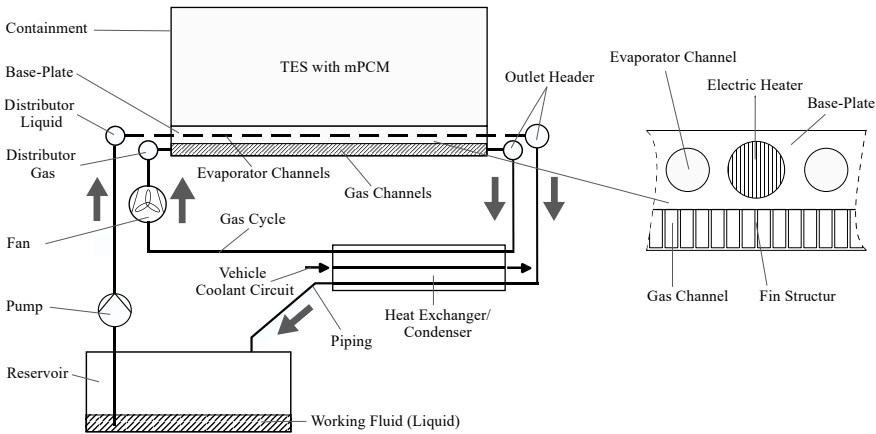


Figure 3.9: Schematic representation of the hybrid concept (Concept 6), showing the components and the arrangement of the evaporator and gas channels. The figure illustrates a coupled operation of evaporator and gas channels.

The heat extraction section consists of evaporator and gas channels integrated into the base plate of the TES, along with electric heaters. In the gas channels, heat extraction

takes place by forced convection of a gaseous working fluid, which is driven by a fan. An appropriate gaseous working fluid could be CO_2 , as described alongside with Concept 2. The evaporator channels are integrated into drill holes, enabling direct contact between the working fluid and the TES base plate. Heat extraction then occurs through forced convection of flow boiling in the channels using a fluid pump, as previously described for Concepts 3 and 4. Again, distilled water at pressures close to ambient is considered an appropriate working fluid for the evaporation-condensation cycle.

As shown in Figure 3.9, the extracted heat is transferred to the vehicle cooling circuit via a heat exchanger. Alternatively, the heat can be transferred directly to the cabin air. The heat exchanger acts as a three-stream heat exchanger, containing all process streams in one component. It is located at a lower level than the TES. The condensate from the evaporation-condensation cycle returns to a reservoir by gravity. The reservoir is intended to hold the entire liquid system volume of the working fluid and acts as a buffer for pressure changes.

Figure 3.10 shows the schematic representation of the start-up and shut-down process. The illustration shows that, during start-up at high temperatures, only the compressed gas cycle is operational while the evaporation-condensation cycle is not in operation. The system is controlled by the fan and liquid pump. Heat extraction can be shut down by stopping the fan of the compressed gas cycle and completely draining the working fluid of the evaporation-condensation cycle into the reservoir. This occurs when there is no heat demand, such as when charging the storage with electric heaters. For medium and low TES temperature levels, coupled operation of both stages is considered, as indicated in Figure 3.9.

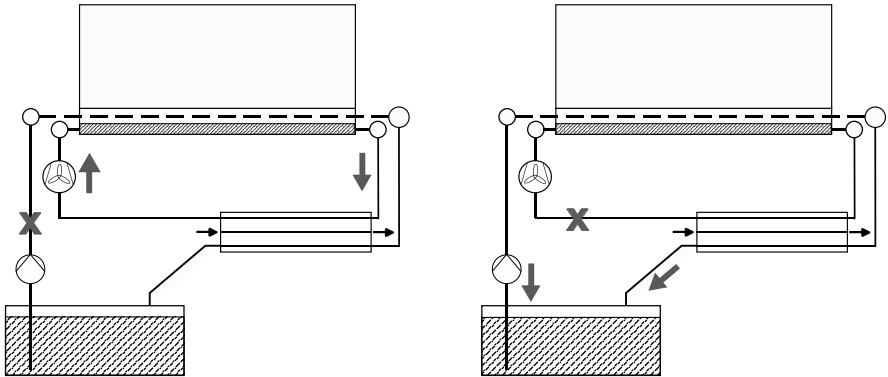


Figure 3.10: Schematic representation of start-up and shut-down process of Concept 6. Left: Start-up process at high TES temperature, with only the gas cycle being operational. Right: Shut-down process by drainage of the liquid working fluid into the reservoir.

This novel concept combines the advantages of single-phase gas discharge and direct contact boiling discharge, while avoiding the disadvantages that would result from individual operations. It is intended to prevent boiling crises involving film and transition boiling at large temperature differences. Consequently, improved performance and controllability of heat transfer at large temperature differences is expected. Additionally, a reduced thermomechanical load is expected compared to a discharge with direct contact boiling only. At lower storage temperatures, direct contact boiling allows for high heat transfer rates.

Operating and controlling both discharge cycles simultaneously achieves high specific heat output and controllability over a wide operating temperature range. At the same time, the negative impact of the boiling crisis is minimized, and the thermomechanical load is reduced. In addition, the proposed concept enables the thermal discharge of a high-temperature thermal energy storage into a state-of-the-art water-glycol vehicle coolant circuit.

Compared to the multi-stage boiling discharge (Concept 3), the hybrid concept is promising in terms of fast and accurate heat flow control. However, the pressurization of the gas system requires higher engineering efforts. Additionally, the system is less compact than Concept 3 due to the necessary piping and additional gas channels with a fin structure to be installed.

3.5 Evaluation of concepts and concept selection

In the following, the developed concepts described in the previous section are undergoing a comparison and evaluation based on different criteria. These criteria are part of the functional, system-related, and other requirements outlined in Section 3.1.

The six relevant criteria selected at this stage of development are: Performance in terms of achievable heat output rate at high, mid, and low temperatures; Efficiency in terms of the electric power consumption of the fan or pump used for discharge in relation to the achievable heat output rate; Simplicity in terms of the complexity of engineering, design, and calculations; Flexibility for heat distribution in terms of the potential to transfer heat to different distribution systems, such as an air or a liquid distribution system; Thermomechanical load in terms of the induced temperature differences in the storage system caused by the discharge and the flexible movement of components; Practicability in terms of the practical feasibility for prototype development and experimentation.

Other requirements outlined in Section 3.1, such as a good controllability of the heat flow, non-hazardous heat transfer fluids, and operation in the full temperature range, are not listed as relevant criteria because they are already fulfilled for all developed concepts. A cost comparison is not considered at this early stage of development.

Table 3.1 shows the evaluation matrix for the concept selection including the six criteria. The numbers 3, 2, 1, and 0 are used to indicate the score, with 3 being the best and 0 being the worst. The evaluation matrix follows a simple approach without weighting factors,

meaning that all criteria are equally weighted for the calculation of the total score. This is because, at this stage of development, all six criteria are considered equally important for the selection process. The numbers are summed up for each concept, resulting in a total score that allows comparison of the concepts. The rating of each criterion in the matrix is based on an engineering pre-design estimation.

Criteria	Air discharge (Concept 1)	Compressed gas (Concept 2)	Multi-Stage (Concept 3)	Direct boiling (Concept 4)	Resistance boiling (Concept 5)	Hybrid (Concept 6)
Performance	1	2	3	2	1	3
Efficiency	2	1	3	2	2	2
Simplicity	3	2	1	2	2	1
Flexibility for heat distribution	1	2	3	3	3	2
Thermomechanical load	2	1	3	0	3	1
Practicability	3	1	2	2	2	1
Σ Total score	12	9	15	11	13	10

Table 3.1: Evaluation matrix for concept selection. The numbers 3, 2, 1, and 0 are used to indicate the score, with 3 being the best and 0 being the worst.

Based on the evaluation matrix, Concept 3, the multi-stage boiling discharge concept, scores the highest due to its high performance, efficiency, flexibility in heat distribution, and low mechanical load. Additionally, it is promising for lab-scale testing. The resistance boiling concept (Concept 5) is second, and the air discharge concept (Concept 3) is third, scoring high for simplicity and practicability. The direct boiling concept (Concept 4) follows behind.

The lowest score goes to the hybrid concept (Concept 6) and the compressed gas concept (Concept 2). This is due to the low scores in practicability. The compressed gas circuit is more difficult to handle, and selecting an appropriate pressure-tight fan with a low compression ratio seems difficult based on a preliminary literature review. Nevertheless, Concepts 2 and 6 are still promising and should be taken in mind if these issues can be resolved.

The following two concepts were selected for further investigation, simulation, prototyping, and experimentation based on their scores in the evaluation matrix and other factors:

- Direct air discharge concept (Concept 1)
- Multi-stage boiling heat extraction concept (Concept 3)

Even though the air discharge concept does not receive the highest score in the evaluation matrix, it is highly relevant from a technical standpoint for vehicle applications. It is also the simplest and most practical approach for further prototyping.

The multi-stage boiling concept allows the extracted heat to be transferred to the vehicle's water-glycol circuit for further distribution throughout the vehicle, making it highly relevant from a technical standpoint. The multi-stage concept is essentially a combination of Concepts 4 and 5: Stage 1 equals Concept 5, and Stage 2 equals Concept 4. Thus, realizing the multi-stage concept automatically incorporates Concepts 3, 4, and 5, which is beneficial because three different concepts can be investigated at once.

Chapters 4 and 5 present a detailed investigation of the two selected concepts, including prototype development. Chapter 6 provides a comparison of the two concepts at the vehicle system level.

4 Direct air discharge – Development and investigation

In this chapter, the direct air discharge concept is further developed and investigated. First, an appropriate heat transfer channel geometry is selected based on the outcome of a developed calculation model. Then, bench-scale and application-scale prototypes were developed and investigated experimentally. Additionally, a conjugate numerical simulation model was developed, and the results are presented. Parts of Sections 4.2 and 4.3 of this chapter have been published in several publications by Nees et al. [76, 78, 79].

4.1 Design philosophy and development

4.1.1 Design philosophy

The general idea of the direct air discharge concept was presented in Section 3.4. Figure 3.5 shows a schematic representation of the concept. In this section, the focus is on the design philosophy and derivation of a calculation model to develop a detailed prototype geometry of the heat transfer section installed in the TES. The containment and general arrangement of the mPCM are not part of the prototype development and are considered boundary conditions, as discussed in Section 3.1.

Figure 4.1 illustrates the configuration of the air heat transfer section, including the geometric details of the fins and fluid channels. The heat transfer section is arranged on the bottom side of the TES according to the geometric boundary conditions discussed in Section 3.1. Therefore, there are no heat transfer sections installed on the side or top walls around the containment. This is due to the shrinkage effects of the mPCM, which would lead to the formation of gaps between the mPCM and the side and top walls during solidification and as the temperature is cooled down further. These gaps result in high thermal resistances, which are unfavorable for heat transfer. The fins and fluid channels are integrated into the base plate alongside the electric heaters. Due to the low heat capacity and relatively low heat transfer coefficient of forced convection airflow, the heat transfer section is arranged over the entire base area of the mPCM (A_{ref}). For manufacturing and installation reasons, the heat transfer section is elevated below the electric heaters.

In addition to what is shown in Figure 4.1, an insulation is arranged beneath and along the sides of the air heat transfer section, as well as around the outer walls of the base plate and containment.

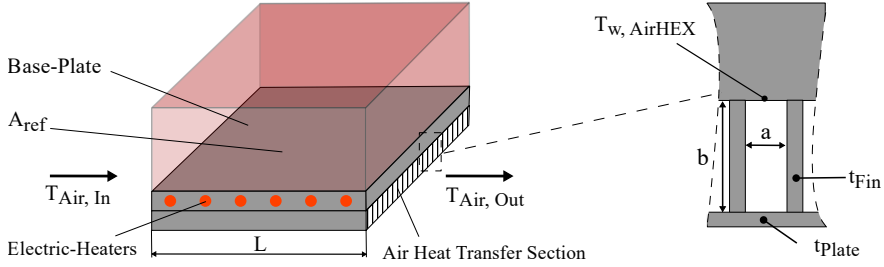


Figure 4.1: Arrangement of the air heat transfer section and geometric details, based on the geometric boundary conditions discussed in Section 3.1

For ease of manufacturing and installation, the electric heaters are rotated by 90 degrees to the flow direction of the fluid channels, as shown in Figure 4.1. This arrangement is also beneficial for numerical simulations, as symmetric boundary conditions can be applied more easily.

The extracted heat flow $\dot{Q}_{ex}$ can be calculated from the specific extracted heat flow $\dot{q}_{ex}$ and the reference area A_{ref} :

$$\dot{Q}_{ex} = \dot{q}_{ex} A_{ref} \quad (4.1)$$

Due to the maximum operating temperature of 650–700 °C, the selection of materials for the fins, base plate, and cover plate on the bottom is limited. Therefore, aluminum cannot be used. Suitable materials include high-temperature stainless steel alloys and nickel. Nickel materials, such as Ni201, have the advantage of high thermal conductivity, ranging around 57–79 W/(mK) [114] in the operating temperature range, compared to stainless steel alloys, which range around 13–26 W/(mK) (such as 1.4404, 1.4828, and 1.4980) [25, 113]. However, nickel is more expensive and not as widely available as stainless steel alloys (in terms of manufacturers). For the prototypes, stainless steel alloys were selected as fin material, as further discussed in Subsection 4.1.4.

To enhance heat transfer, fins are used to extend the surface area of the air heat transfer section. The underlying relationships were discussed in Subsection 2.4.2. A cover plate made from the same material group as the fins is used to close the fluid channels on the bottom side.

The following three aspects were considered for the design of the heat transfer section and the detailed fluid channel geometry, resulting in a conflict of objectives:

1. Performance: Achievement of the required heat output in a large operating temperature range; High specific heat flow; High amount of storage utilization.

2. Efficiency: Low electric power consumption of the fan, as the fan is driven by the energy from the vehicle's battery and the installed fan size influences volume, weight, and costs.
3. Compact design: Volume and weight of the heat transfer section are kept as low as possible.

Given the many degrees of freedom, such as fin type, channel height and width, fin thickness, mass flow, and Reynolds number, selecting an optimal geometry is challenging. Different performance evaluation criteria for enhanced surfaces are mentioned in the literature [122]. These criteria typically compare a reference (index 0) with a modified arrangement. One commonly used method is the thermal enhancement factor (η_{TEF}), which compares the Nusselt number (Nu) and the friction factor (f) [52, 69]:

$$\eta_{TEF} = \frac{Nu/Nu_0}{(f/f_0)^{1/3}} \quad (4.2)$$

This dimensionless parameter is used to evaluate the effectiveness of different geometrical arrangements. It considers both heat transfer and pressure drop under the assumption of an equal fan power requirement. However, since the parameter assumes equal Reynolds numbers, surface areas, and temperature increases for the compared arrangements, it can only be considered indicative. Using the equations discussed in Section 2.4 for Nu and f yields values of η_{TEF} smaller than 1 for various serrated and wavy fin geometries compared to plain fins. Based on this, plain fins and wavy fins with low amplitude (with similar thermohydraulic characteristics to plain fins) are promising candidates for the engineering application investigated in this thesis, which will be discussed further within Subsections 4.1.3 and 4.1.4.

4.1.2 Heat transfer calculation model

In this section, a heat transfer calculation model for the air discharge concept is derived. With the help of the model, the goal is to estimate the heat extraction performance of the concept and to select and design the appropriate heat transfer geometry. The model is based on an analytical approach and correlation-based equations for heat transfer coefficients. It is limited to heat transfer in the fluid channels and does not consider thermal resistances in the mPCM or layers in between. The model takes both heat transfer and fan power into account.

The model includes several assumptions and simplifications. Figure 4.1 shows the basic geometry. A constant wall temperature is assumed in the flow direction at the base plate interface ($T_{w,AirHEX} = const.$). The model is considered 0D/1D, rather than providing three-dimensional resolution of the real geometry. It considers a steady-state heat transfer process for heat conduction in the fins and forced convection in the fluid channels, while neglecting radiation effects. The temperature-dependent properties of the air and solid

materials are considered. For the air inlet flow, a temperature of 20 °C and ambient pressure are assumed.

Figure 4.2 shows a schematic representation of the modeling approach. The model uses an iterative approach until the energy balance is fulfilled ($\dot{Q}_h = \dot{Q}_m$). The model is initialized with input parameters, such as temperatures ($T_{air,in}$ and $T_{w,AirHEX}$), mass flow, electric fan power relative to heat flow, and the geometry, fluid, and material properties. The resulting air outlet temperature ($T_{air,out}$) is used as input for the iteration of the equations. The iteration continues until the deviation $\Delta\dot{Q}$ of the energy balances $\dot{Q}_h$ and $\dot{Q}_m$, which is determined by the method of least squares, is close to zero.

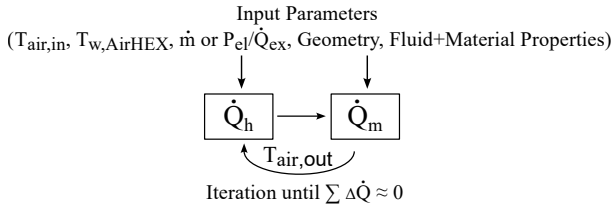


Figure 4.2: Schematic representation of the air heat transfer modeling approach

The energy balance due to convective heat transfer ($\dot{Q}_h$) in the air channels is calculated by:

$$\dot{Q}_h = h(\eta_{fin}A_{fin} + (A_{ref} - A_c)) \frac{dT_{in} - dT_{out}}{\ln(dT_{in}/dT_{out})} \quad (4.3)$$

$$\text{with } dT_{in} = T_{w,AirHEX} - T_{air,in} \quad \text{and} \quad dT_{out} = T_{w,AirHEX} - T_{air,out}$$

Here, h is the heat transfer coefficient; η_{fin} is the fin efficiency; A_{fin} is the finned surface area; A_{ref} is the reference area; and A_c is the cross-sectional area of the fin.

The surface area of the finned channel A_{fin} is calculated by assuming that the cover plate installed at the bottom acts as an extension of the fin:

$$A_{fin} = n_{ch}(2b + a)L \quad (4.4)$$

Here, n_{ch} is the total number of channels. As shown in Figure 4.1, b is the channel height, a is the channel width, and L is the channel length.

The cross-sectional area of all fins is calculated with the fin thickness t_{fin} as follows:

$$A_c = (n_{ch} + 1)t_{fin}L \quad (4.5)$$

The fin efficiency is calculated using Equations 2.22 and 2.24, under the assumption that the effective fin height is $b + a/2$:

$$\eta_{fin} = \frac{\tanh m(b + a/2)}{m(b + a/2)} \quad (4.6)$$

$$\text{with } m = \sqrt{\frac{hU}{k_{fin}A_c}}, \quad U = 2t_{fin} + 2(b + a/2)$$

Here, U is the fin perimeter, and k_{fin} is the thermal conductivity of the fin material.

The total width W of the air heat transfer section is calculated as follows:

$$W = n_{ch}a + (n_{ch} + 1)t_{fin} \quad (4.7)$$

The energy balance by mass transport ($\dot{Q}_m$) for a single-phase airflow is modeled as follows:

$$\dot{Q}_m = \dot{m}c_p(T_{air,out} - T_{air,in}) \quad (4.8)$$

with $\dot{m}$ being the total mass flow of the air and c_p being the specific heat capacity of the air considered for the average mean temperature between the inlet and the outlet.

The Nusselt number for plain fin geometries in the laminar regime is calculated using Equation 2.20, assuming a uniform wall temperature. In the turbulent regime, Equation 2.19, the Gnielinski correlation, is used. For serrated fins, Equations 2.25 and 2.26 are used. For high performance wavy fins, Equations 2.25 and 2.27 are used.

The heat transfer coefficient h is derived from Equation 2.8, which relates to the Nusselt number. The Reynolds number is calculated using Equation 2.10. Therefore, the hydraulic diameter d_h is calculated according to the geometric settings shown in Figure 4.1 as follows:

$$d_h = \frac{4ba}{2(b + a)} \quad (4.9)$$

The friction factor f for a plain fin geometry and laminar flow is derived from Equation 2.32,

and for turbulent flow, it is derived from Equation 2.30. For serrated fins, Equation 2.33 is used. For high performance wavy fins, Equation 2.34 is used.

The pressure drop from inlet to outlet is then calculated by:

$$\Delta p = f \frac{L}{d_h} \frac{\rho_{air,m} v_{air,m}^2}{2} + \zeta \frac{\rho_{air,in} v_{air,in}^2}{2} + \zeta \frac{\rho_{air,out} v_{air,out}^2}{2} \quad (4.10)$$

$$\text{with } v = \dot{m}/(\rho A_f) \quad \text{and} \quad A_f = n_{ch} b a$$

Here, A_f is the cross-sectional flow area of all fluid channels, and $\rho_{air,in}$, $\rho_{air,out}$, and $\rho_{air,m}$ are the air densities at the inlet, outlet, and the average mean temperature between the inlet and outlet, respectively. The air velocities ($v_{air,in}$, $v_{air,out}$, and $v_{air,m}$) are calculated using the corresponding densities at the inlet, outlet, and the average mean temperature between the inlet and outlet. The friction factor, ζ , is assumed to be 1.5, which corresponds to an additional pressure drop in the entrance and exit zones of the heat transfer section.

The electric power consumption (P_{el}) of the fan is then calculated using Equations 2.36 and 2.37. For fan efficiency, the value $\eta = 0.3$ is used, which is within the range of fans used in this study.

4.1.3 Parameter study and discussion

In order to select an optimal engineering design for prototyping, it is important to identify and estimate the parameters that primarily impact the performance, efficiency, and compactness of the air heat transfer section. The parameter study presented in this section is based on the required heat output of 10 kW, as discussed in Section 3.1, and takes into account the size of the application-scale prototype. However, the main findings and conclusions from this study are also valid for other heat output rates and storage sizes on a qualitative basis, such as the 1 kW bench-scale prototype.

The following parameters were identified to impact the design of the air heat transfer section:

- Minimum design temperature at which the required heat can still be extracted
- Channel height, respectively volume and weight of the heat exchanger
- Fin type and specific geometry applied, e.g. plain, serrated, or others (f- and j-factors)
- Base area of the mPCM/base plate for heat transfer (A_{ref})
- Fan power applied, respectively ratio $P_{el}/\dot{Q}_{ex}$

- Ratio of air heat transfer section length to total width (L/W)
- Solid material used for fins, cover-plate, and base plate
- Fin thickness and channel width
- Air inlet temperature

Figures 4.3 and 4.4 present the results of the parameter study based on the calculation model discussed in the previous section. The temperature shown in the graphs is the lower limit temperature, defined as the lowest $T_{w,AirHEX}$ temperature at which the required heat flow can be achieved. This temperature is relevant in real applications because it defines the operating temperature range of the storage system and the amount of heat that can be extracted.

The normalized volume used in the graphs is a dimensionless factor that simplifies the comparison of different designs. A channel height of 5 mm with a 1 mm cover plate installed over the full base plate bottom area is considered as the base volume of 1. The other volumes plotted in the graphs are based on channel heights of 12, 20, 30, and 45 mm with an identical cover plate.

According to Figures 4.3 and 4.4, an increase in normalized volume or channel height leads to a decrease in the lower limit temperature. An increase in normalized volume from 1 to 3.5 results in a significant decrease in the lower limit temperature. However, a further increase in volume results in only a slight decrease in the lower limit temperature. This is due to fin efficiency, which decreases as fin height increases. Therefore, the lower limit temperature cannot be reduced arbitrarily, even in the case of an infinite heat exchanger volume.

Figure 4.3(a) shows a comparison of different fin types, including a short serrated fin with a 5 mm serration length, a high-performance (HP) wavy fin, a long serrated wavy fin with a 20 mm serration length, and a plain and lazy ruffled (LR) wavy fin. In terms of lower limit temperature for the given setting, such as a ratio of electric fan power to heat flow of 1%, the plain and lazy ruffled wavy fin performs significantly better than the other fin types. This finding supports the assessment in Subsection 4.1.1 in the context of Equation 4.2. The plain and lazy ruffled wavy fin performs better than the other fin types because the lower pressure drop allows for higher mass flow rates and lower temperature differences between the fluid bulk and the hot wall.

Figure 4.3(b) indicates the impact of different base areas (A_{ref}). The larger the base area, the smaller the lower limit temperature for a given volume. Doubling the base area divides the heat flux by half for a given heat output. Halving the heat flux from 8 W/cm² to 4 W/cm² has a greater effect on the lower limit temperature than halving the heat flux from 4 W/cm² to 2 W/cm².

Figure 4.3(c) shows the comparison of various electric fan power-to-extracted heat output ratios ($P_{el}/\dot{Q}_{ex}$): 0.5%, 1%, 2%, and 5%. An increase in the $P_{el}/\dot{Q}_{ex}$ ratio leads to reduced

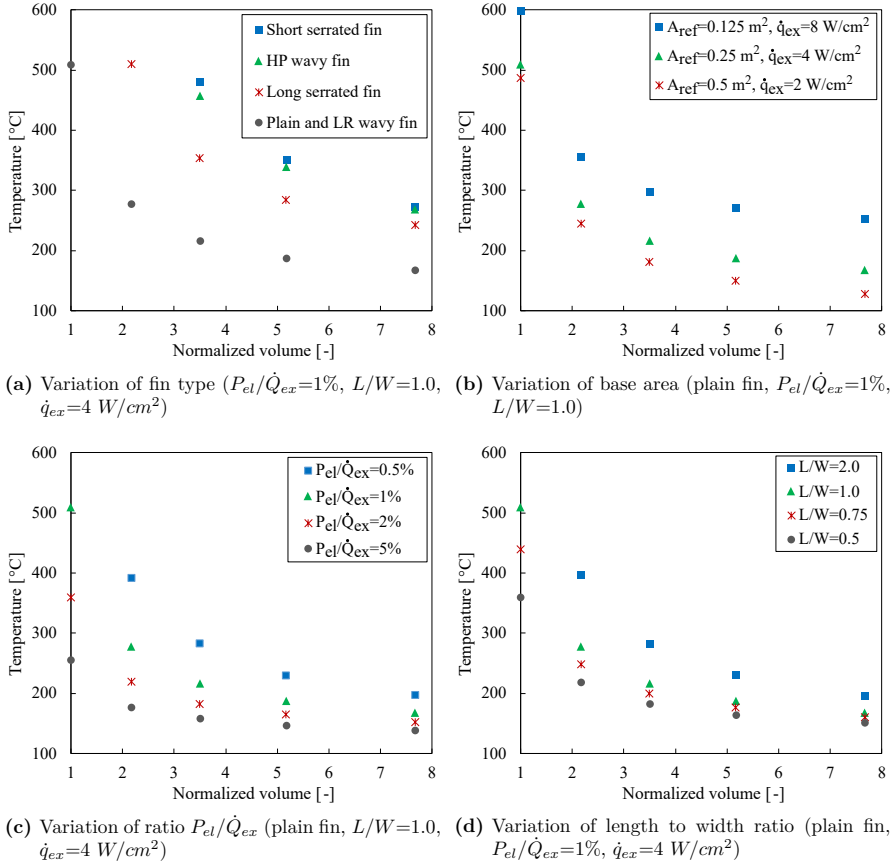


Figure 4.3: Required lower limit temperature for $T_{w,AirHEX}$ versus normalized volume of the air heat transfer section in dependence of different parameters (heat output 10 kW, fin material stainless steel alloy 1.4404, fin thickness 0.2 mm, channel width 1.5 mm)

lower limit temperatures for a given volume. Increasing the ratio from 0.5% to 1% for a normalized volume of 3.5 results in approximately 70 K smaller lower limit temperature, which corresponds to approximately 7% higher storage utilization of the AlSi_{12} storage material. Increasing the ratio from 1% to 5%, while maintaining the same volume, results in an increase in storage utilization of approximately 5% but requires 4% more electric fan power. Therefore, a $P_{el}/\dot{Q}_{ex}$ ratio of around 1–2% is reasonable from a design point of view. Larger ratios are less beneficial and require notably larger fan sizes and weights.

Figure 4.3(d) shows the impact of different length-to-width ratios of the air heat transfer section. Increasing the width and reducing the length, while maintaining an equal base

area, improves heat transfer by reducing the logarithmic mean temperature difference between the air and the wall in the flow direction. The lower limit temperature decreases by approximately 100 K for an L/W ratio of 0.5 compared to a ratio of 2.0 for a normalized volume of 3.5. However, a small L/W ratio results in larger air duct dimensions at the inlet and outlet, and makes an equal flow distribution across the channels more challenging.

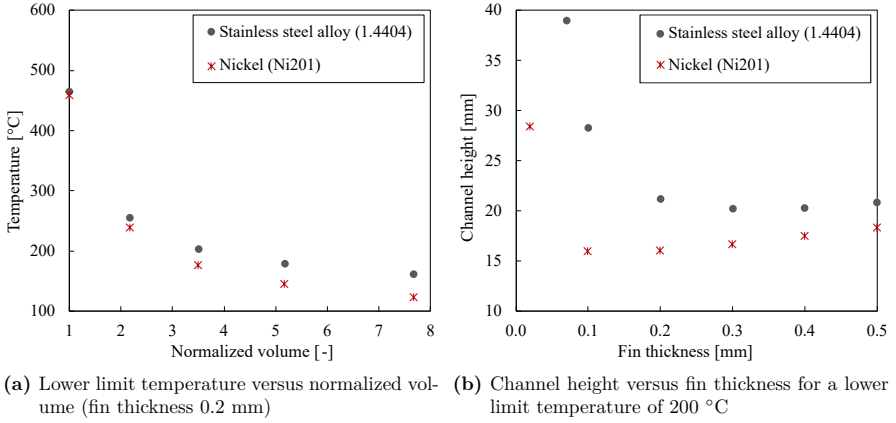


Figure 4.4: Impact of material selection and fin thickness on lower limit temperature and channel height (heat output 10 kW, $\dot{q}_{ex}=4\text{ W/cm}^2$, $P_{el}/\dot{Q}_{ex}=1\%$, $L/W=0.81$, plain fin design with 1.5 mm channel width)

Figures 4.4(a) and 4.4(b) illustrate the effects of material selection and fin thickness on the lower limit temperature and required channel height. As shown in Figure 4.4(a), the characteristic trend in terms of temperature versus volume is very similar for the two considered materials. However, using nickel (Ni201) instead of stainless steel alloys (1.4404) as the fin material reduces the lower limit temperature for a given volume. At a normalized volume of 3.5, the lower limit temperature decreases by 27 K with Ni201, allowing for approximately 3% higher storage utilization of the AlSi_{12} storage material. This is due to the higher thermal conductivity of Ni201 compared to 1.4404.

Figure 4.4(b) illustrates the relationship between channel height and fin thickness at a lower limit temperature of 200 °C for both Ni201 and 1.4404. Generally, using Ni201 instead of 1.4404 reduces the required channel height. A reasonable design selection in terms of channel height, volume, and weight is a fin thickness of 0.2 mm for stainless steel alloys and 0.1 mm for Ni201. For lower fin thicknesses, the required channel height and heat transfer section volume increase significantly. For higher fin thicknesses, the weight increases significantly without further reducing the volume.

Another parameter that impacts the design is the considered air inlet temperature. The smaller the air inlet temperature, the better the heat transfer, as the temperature difference between the fluid and the wall increases. The air inlet temperature depends on

whether the cabin air is recirculated and is in the range between the ambient and cabin temperature. In this study, a conservative approach is used to consider the air inlet temperature at 20 °C.

4.1.4 Prototype channel geometry

Based on the findings in the previous section and the limitations of the manufacturing process and geometrical boundaries, an appropriate design for the heat transfer section is selected for the bench-scale and application-scale prototypes. The manufacturing process for attaching the fins to the base plate differs for each prototype. In both cases, a cover plate is used to close the fluid channels on the bottom side.

Bench-scale prototype

Figure 4.5 illustrates the chosen channel geometry for the bench-scale prototype. The heat transfer section consists of parallel plain fin channels. These channels are milled directly from the base plate material, which is high-temperature stainless steel (type 1.4980). According to the manufacturer's information, this process is limited to fin thicknesses of larger than 1 mm. The selected dimensions are a channel height of 10 mm, a channel width of 4.85 mm, and a fin thickness of 1.65 mm, resulting in 28 channels, each 180 mm long. The geometric details of the heat transfer section are summarized in Table 4.2. More details about the prototype are provided in Section 4.2.

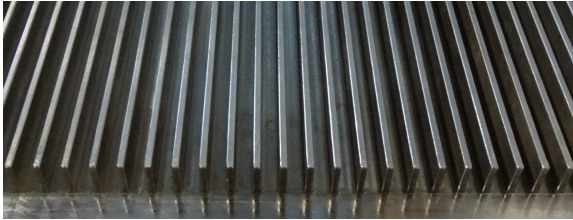


Figure 4.5: Channel geometry for the bench-scale prototype (plain fin)

Applying the calculation model developed in the previous section, the thermo-hydraulic performance of the heat transfer section was designed to achieve a heat output of 1 kW with a lower limit temperature of $T_{w,AirHEX}$ of 200 °C and a $P_{el}/\dot{Q}_{ex}$ ratio of approximately 1%.

Application-scale prototype

Figure 4.6 illustrates the selected channel geometry for the application-scale prototype. The heat transfer section consists of parallel lazy ruffled wavy fins. The fins are formed from thin sheets of metal and brazed to the base and cover plates in a vacuum oven. According to the manufacturer's information, this forming process is limited to fin thicknesses of less than 0.5 mm. Conventionally manufactured plain fins from the metal forming

process are typically limited to heights smaller than 10 mm. Lazy ruffled wavy fins allow for significantly larger fin heights and provide higher mechanical stability. According to the manufacturer's data, their thermohydraulic characteristics are similar to those of plain fins due to their low wave amplitude. Therefore, they are selected for prototyping. For larger prototypes and potential mass production, this manufacturing process is more appropriate than drilling each channel.

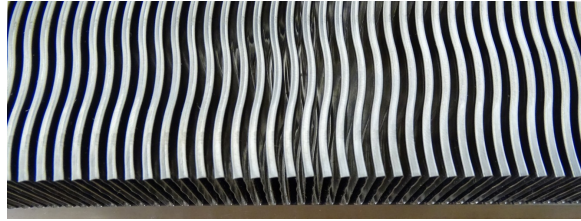


Figure 4.6: Channel geometry for the application-scale prototype (lazy ruffled wavy fin)

The selected dimensions are a channel height of 20 mm, a channel width of 1.5 mm, and a fin thickness of 0.2 mm, resulting in 326 parallel channels. The total width is 555 mm, and the channel length is 450 mm. This results in a length-to-total width ratio of 0.81. This ratio is a compromise between improved heat output performance and reasonable overall prototype dimensions. Due to limited availability, stainless steel type 1.4404 was used as fin material instead of nickel. Geometry details are summarized in Table 4.6. More details on the construction and prototype is given in Section 4.4.

Applying the calculation model developed in the previous section, the thermo-hydraulic performance of the heat transfer section was designed in order to achieve a heat output of 10 kW with a lower limit temperature of $T_{w,AirHEX}$ of 200 °C and a $P_{el}/\dot{Q}_{ex}$ ratio of approximately 1%.

4.2 Experimental investigation of bench-scale prototype

4.2.1 Prototype design and test bench

Figure 4.7 presents the experimental setup and schematic design of the bench-scale storage prototype, along with selected measurement points. The storage is based on a box-shaped graphite container design that accommodates the storage material $AlSi_{12}$. To prevent graphite oxidation at high temperatures, the container is pressurized with argon and enclosed by a steel container. The graphite based encapsulation concept is also described by Stahl et al. [103]. A steel base plate installed on the bottom contains electric heaters for charging and fluid channels for discharging heat. Heat extraction is achieved by forced convection of ambient air using a controllable fan. The test bench contains thermocouples to measure the temperature of the mPCM, base plate, and air section, as well as equipment

to measure air mass flow and fluid pressure drop. The steel container and base plate are surrounded by microporous insulation made of TH1000+ from Thermoline Manufaktur.

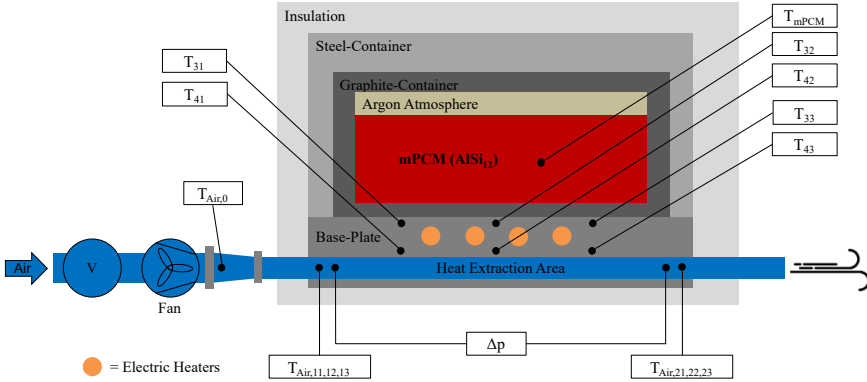


Figure 4.7: Schematic representation of the bench-scale storage prototype and experimental setup, adapted from Nees et al. [79]

The fan utilized in this investigation was an Ebm-Papst EC-radial blower (G1G108-AB17-02), with a variable fan speed achieved through the modulation of the supply voltage. A Bosch HFM5 mass sensor was installed in the air section for the mass flow measurement. It was calibrated beforehand using an Emerson Electronic Coriolis sensor. A KIMO Model C310 was used to measure the differential pressure of the air at the inlet and outlet of the heat extraction channels. Type-N thermocouples with diameters of 1 mm and 1.5 mm were installed for temperature measurement. The thermocouples were calibrated beforehand for a temperature range from ambient to 800 °C using a custom-made calibration setup with Pt100 and type S thermocouple reference sensors. The argon atmosphere was realized by connecting to an argon cylinder, which maintained constant pressure inside the prototype container during the experiments. Four electric heaters with a diameter of 6.5 mm and an individual installed power of 600 W from Türk+Hillinger were installed for the charging process. The experimental data were processed using a data-acquisition system from Delphin Technology AG. The hardware Expert Logger 400 and the software ProfiSignal were used for this purpose.

Figure 4.8 shows the installed setup and prototype in the lab. It shows the setup with and without insulation. Figure 4.9 presents the AlSi₁₂ storage material inside the graphite container and the graphite bottom plate. Figure 4.10 illustrates the base plate prototype for charging and discharging heat, as well as the structure of the plain fin channel geometry for heat extraction. The base plate and fins are made of high-temperature stainless steel (type 1.4980). The connecting parts and the channel cover plate were laser welded to the base plate. They are made out of stainless steel (type 1.4301).

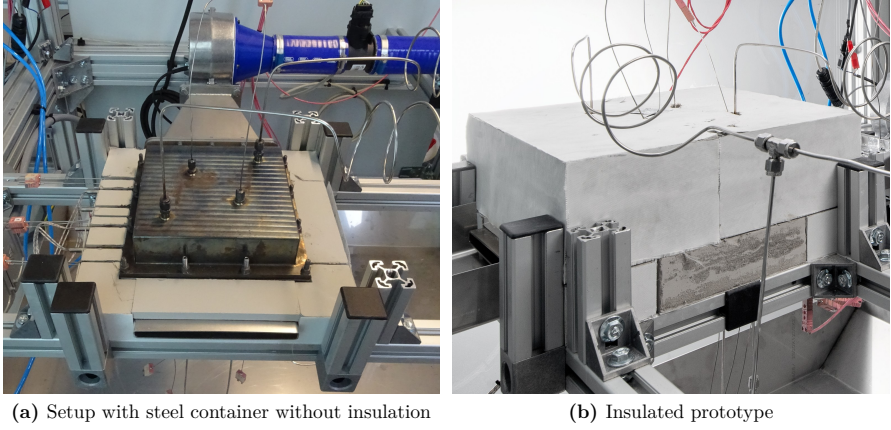


Figure 4.8: Test bench and prototype, adapted from Nees et al. [79]

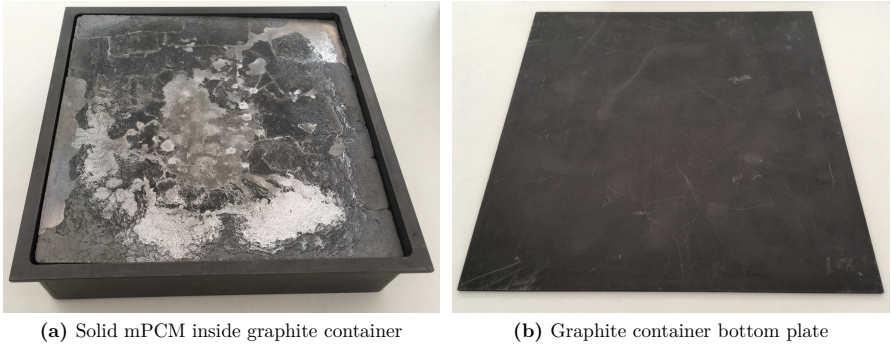


Figure 4.9: Graphite container and storage material, adapted from Nees et al. [79]

The basic data of the investigated bench-scale storage prototype are summarized in Table 4.1. The physical properties of the AlSi_{12} storage material are provided in Table 2.1 and Appendix A.

As shown in Figure 4.10(b), the air heat transfer section consists of parallel, rectangular, plain-fin channels that are drilled directly into the base plate. Table 4.2 summarizes the detailed channel geometry. This design was selected to achieve low pressure drop and high heat transfer for high-efficiency thermal discharge. The design philosophy and selection process are thoroughly discussed in Section 4.1.

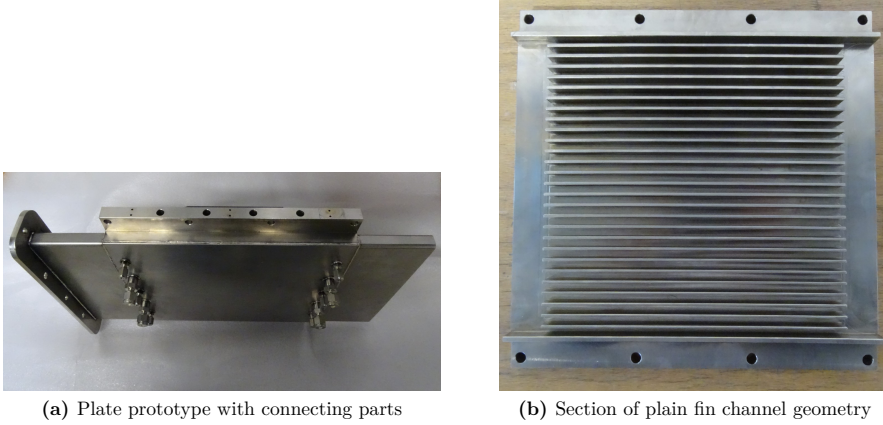


Figure 4.10: Plate prototype for heating and heat extraction with integrated plain fin geometry, adapted from Nees et al. [79]

Item	Specification
Total storage capacity	1.59 kWh (100 – 650 °C)
Total dimension (with insulation)	0.329 x 0.329 x 0.154 m
Total dimension (without insulation)	0.225 x 0.225 x 0.077 m
Weight total	15.49 kg (of which 3.73 kg is insulation)
Base area AlSi ₁₂ (A_{ref})	0.18 x 0.18 m
Weight AlSi ₁₂	2.45 kg
Electric heaters installed	4 x 600 W
Gravimetric energy density	102.6 Wh/kg (135.2 Wh/kg without insulation)
Volumetric energy density	95.4 Wh/L (407.9 Wh/L without insulation)

Table 4.1: Basic data of the bench-scale storage prototype, numbers based on data from Nees et al. [79] and additional information

Item	Specification
Type	Rectangular channel with plain fin design
Base area dimension with channels	0.18 x 0.18 m
Air channel geometry	10 x 4.85 mm (b x a), 28 parallel channels
Air channel hydraulic diameter	6.53 mm
Fin thickness	1.65 mm

Table 4.2: Detailed geometry of the air heat transfer section, based on data from Nees et al. [79]

The weight of the AlSi₁₂ storage material is relatively small compared to the total weight of the prototype. Consequently, only about 45% of the total storage capacity comes from

the AlSi_{12} material. Thus, a considerable amount of the total storage capacity originates from the sensible heat of other components, such as the base plate used for charging and discharging, as well as the graphite and steel containers.

4.2.2 Experimental procedure

Figure 4.11 illustrates the schematic procedure of the conducted experiments, as well as an exemplary charging and discharging curve including the temperature time profiles for the mPCM and base plate. The mPCM temperature corresponds to the reading of thermocouple T_{mPCM} in Figure 4.7, while the average base plate temperature was computed from the six thermocouple readings: T_{31} , T_{32} , T_{33} , T_{41} , T_{42} , and T_{43} .

In the first step, the storage was charged using the installed electric heaters. The charging process continued until the AlSi_{12} temperature reached 650°C , which is higher than the phase change temperature of 577°C , to ensure complete melting. In the second step, the discharge process was initiated. Therefore, the heat input by the electric heaters was stopped, and heat extraction was achieved by forced convection of ambient air using the installed fan.

Experiments were carried out by applying various constant fan power rates, constant mass flow rates, and constant heat output rates to study the heat extraction characteristics for different parameters. The operating temperature range of the storage material was between 650°C and 100°C . Experiments were conducted at least twice for each parameter set to verify the reproducibility of the measured data.

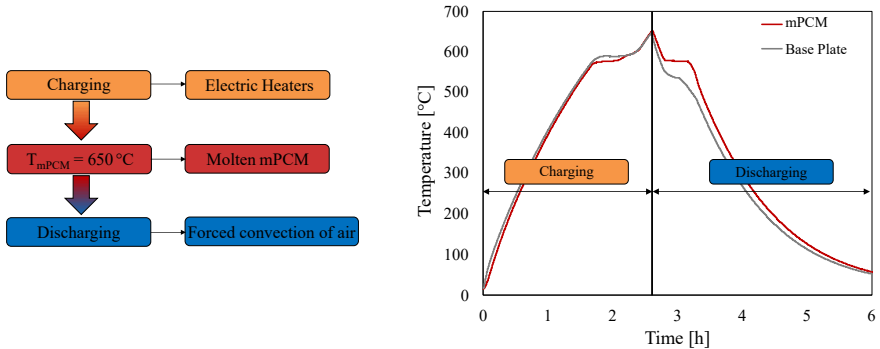


Figure 4.11: Experimental procedure and exemplary charging and discharging curve, adapted from Nees et al. [79]

The extracted heat flow $\dot{Q}_{ex}$ was derived from the measured data as follows:

$$\dot{Q}_{ex} = \dot{m}c_p(\bar{T}_{Air,21,22,23} - T_{Air,0}) \quad (4.11)$$

Where $\dot{m}$ is the measured air mass flow, c_p is the heat capacity of the air considered for the average mean temperature between outlet and inlet, and $\bar{T}_{Air,21,22,23}$ and $T_{Air,0}$ are the measured average air temperature at outlet and inlet.

The heat flux $\dot{q}_{ex}$ is then calculated with the reference area A_{ref} of the AlSi₁₂ base area:

$$\dot{q}_{ex} = \frac{\dot{Q}_{ex}}{A_{ref}} \quad (4.12)$$

The amount of accumulated extracted energy Q_{ex} is then determined from the time of discharge t as follows:

$$Q_{ex} = \int_{t=0}^{t=end} \dot{Q}_{ex} dt \quad (4.13)$$

4.2.3 Measurement uncertainty

To verify the quality of the experimental results, it is essential to analyze the measurement uncertainty of the equipment used. The relevant measurement points are shown in Figure 4.7 and include temperature, mass flow, and differential pressure measurements. The combination and propagation of uncertainties were conducted following the JCGM guidelines [41]. Supplemental data to this subsection are given in Appendix B.

The Bosch HFM5 mass sensor has a specified accuracy of $\pm 3\%$ according to the manufacturer's data sheet. It was installed to directly measure the air mass flow ($\dot{m}$) in the inlet line. Prior to installation, it was calibrated using a Coriolis sensor of the company Emerson Electronic.

The KIMO Model C310 was installed to accurately measure the pressure drop (Δp) across the air heat transfer section. According to the manufacturer's data sheet, the accuracy is ± 2 Pa and $\pm 0.2\%$ of the measured value using the SPI2-500 measurement module. As the gauge was mounted at a sufficient distance from the hot storage and connected via non-insulated capillary tubes, the effects of temperature were negligible.

To improve the precision of temperature measurements, type N thermocouples were calibrated using a custom calibration setup with PT100 and type S thermocouple reference sensors. The calibration was carried out in the final test bench setup with a Carbolite Gero laboratory chamber furnace. The thermocouples were calibrated for a temperature range from ambient to 800 °C. The total measurement uncertainty is a result of the combination of the uncertainty of the PT100 and type S reference sensor, the calibration setup, and the regression error. It is less than ± 1.4 K for all thermocouples over the entire temperature range.

The calculation of the extracted heat flow is based on the measurements of $\dot{m}$, $T_{Air,0}$, $T_{Air,21}$, $T_{Air,22}$, and $T_{Air,23}$, and is therefore subject to the propagation of uncertainties. The absolute uncertainty in the heat flow is approximately 160 W with 100% fan power

and a mPCM temperature of 650 °C. This corresponds to a relative uncertainty of 3%. The absolute uncertainty in heat flow decreases steadily to approximately 20 W at a mPCM temperature of 100 °C, while the relative uncertainty increases to approximately 6%. For a constant mass flow rate of 15 kg/h, the absolute uncertainty in the heat flow is approximately 60 W at a mPCM temperature of 650 °C, decreasing to around 6 W at 100 °C. The corresponding relative uncertainty ranges from 3 to 3.6% across the full temperature range.

4.2.4 Results and discussion

4.2.4.1 Constant fan power rate

Thermal discharge experiments were conducted by applying three constant fan power rates: 100%, 35%, and 5%. The results for the heat extraction characteristics are shown in Figures 4.12(a)–(d). Figure 4.12(a) shows the heat flow versus the mPCM temperature. Evidently, the heat flow ($\dot{Q}_{ex}$) increases with an increased storage temperature and fan power rate. A maximum heat flow of up to 7 kW was observed due to a short time-transient start-up effect that occurs when the fan is first activated. The storage material undergoes phase change at a temperature of around 577 °C. Before solidification, a heat flow of approximately 3.6 kW or 11 W/cm², respectively, is achieved at 100% fan power. After solidification, performance drops significantly to 2.2 kW or 7 W/cm², respectively, at the same fan power rate. This indicates a significant increase in thermal contact resistance at the mPCM-graphite-container interface due to the phase change from liquid to solid. At an mPCM temperature of approximately 285 °C, a heat flow of 1 kW or 3.1 W/cm² is still achieved with 100% fan power.

As shown in Figure 4.12(b), the phase change is visible due to the plateau of the mPCM temperature against time. The air outlet temperature drops sharply during the first few seconds of discharge, falling from approximately 560 °C to around 300 °C. After that, it gradually falls over time. This occurs because, during charging of the storage system, the fan is off, allowing the air in the fluid channels and fins to heat up to nearly the base plate temperature. After the discharge is initiated, the fan delivers cool ambient air, which replaces the initial hot air and is subsequently heated from inlet to outlet via heat transfer in the channels.

Figures 4.12(c)–(d) show the mass flow and pressure drop characteristics during the discharge. Discontinuity of the curves around the phase change temperature is observed, which is caused by the change in thermal contact resistance at the mPCM-graphite-container interface. As a consequence, the occurrence of the phase change can be determined directly via a mass flow or pressure drop measurement without further measurements required.

An additional noticeable effect can be seen in Figures 4.12(c)–(d). Although the fan power rate was set as constant during the experiment, the mass flow and pressure drop increased or decreased significantly with respect to the mPCM temperature. This is due to the significant change in air density and viscosity in the test section during discharge.

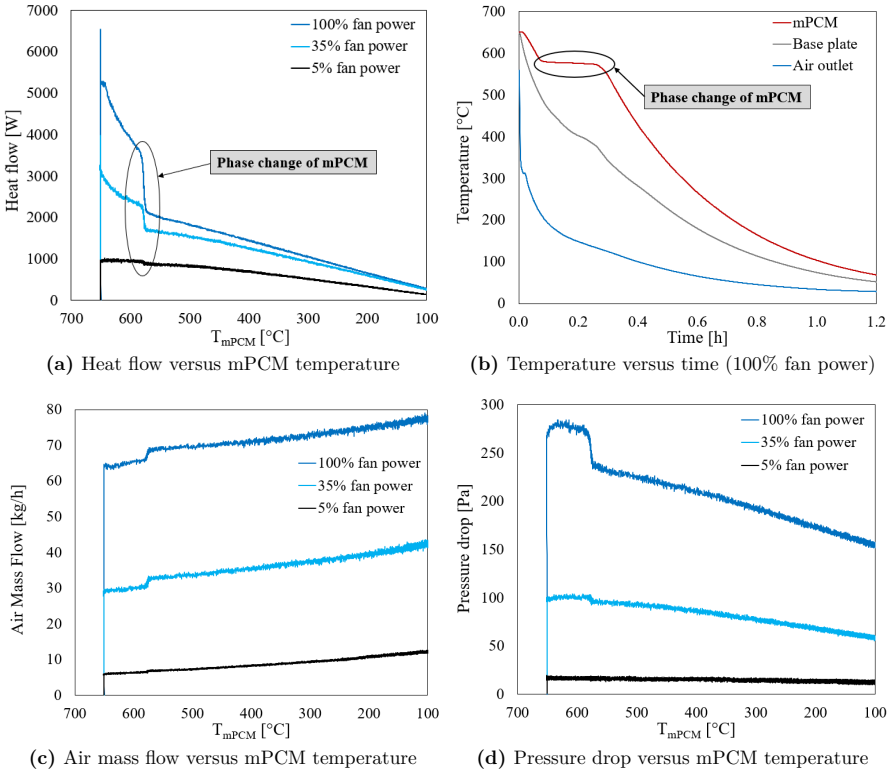


Figure 4.12: Heat extraction characteristics for constant fan power rate, adapted from Nees et al. [79]

The extracted energy (Q_{ex}) is derived from the experimental data in the temperature range of 650–100 °C as follows: 1.55 kWh (100% fan power), 1.54 kWh (35% fan power), and 1.50 kWh (5% fan power). The extracted energy differs slightly from the theoretical total storage capacity given in Table 4.1, which is most likely due to heat losses.

4.2.4.2 Constant mass flow rate

Thermal discharge experiments were conducted at three different mass flow rates: 60 kg/h, 30 kg/h, and 15 kg/h. The results for the heat extraction characteristics are shown in Figures 4.13(a)–(d). Figure 4.13(c) shows that a constant mass flow was successfully achieved over the full operating temperature range via the chosen fan speed control. The results in Figures 4.13(a)–(b) and 4.13(d) are qualitatively similar to those discussed in the previous section for a constant fan power rate, differing only in absolute numbers due to the difference in air mass flow between the experiments.

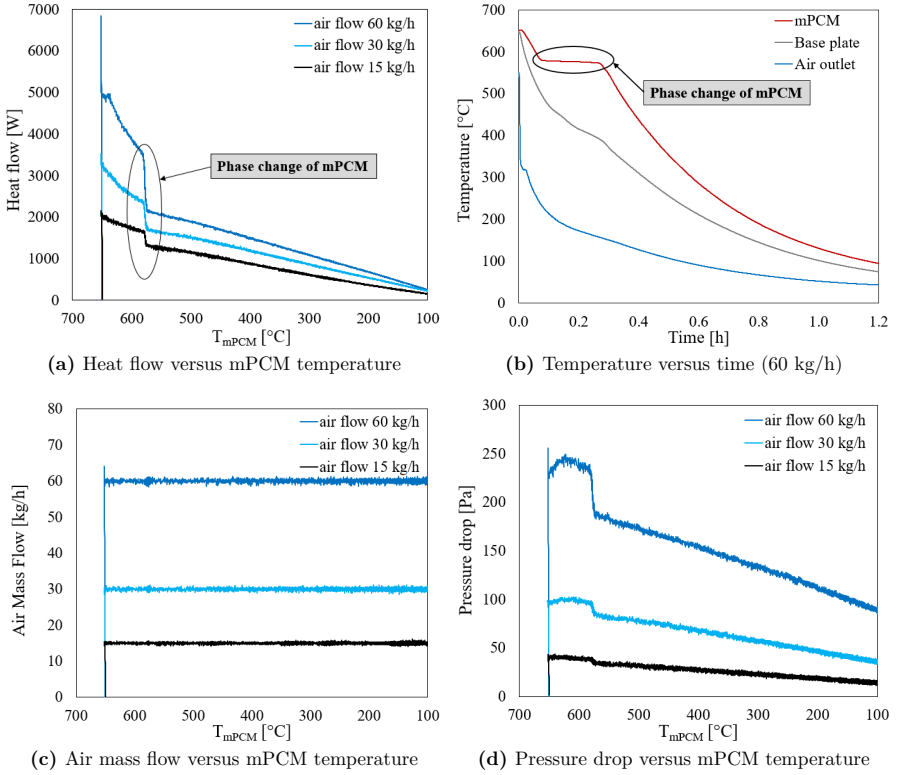


Figure 4.13: Heat extraction characteristics for constant mass flow rate, adapted from Nees et al. [79]

4.2.4.3 Constant heat flow rate

For real-world applications of thermal storage in vehicles, controlling constant heat output is important. Therefore, the heat extraction characteristics for a constant heat flow of 1 kW were studied and are shown in Figures 4.14(a)–(d). Based on the fan's control mechanism, accurate control of the heat flow was only possible to a certain extent, as seen in Figure 4.14(a) through slight fluctuations in the graph. A heat flow of approximately 1 kW was successfully controlled within the operating temperature range of 650 $^{\circ}\text{C}$ to 240 $^{\circ}\text{C}$. However, for temperatures lower than that, heat extraction of 1 kW could not be achieved.

As indicated in Figure 4.14(b), the temperature difference between the mPCM and the base plate in the solid state of the AlSi_{12} is relatively constant at about 60 K. This temperature difference is necessary to overcome the thermal resistances between the heating plate and the mPCM to achieve a total heat flow of approximately 3.1 W/cm^2 .

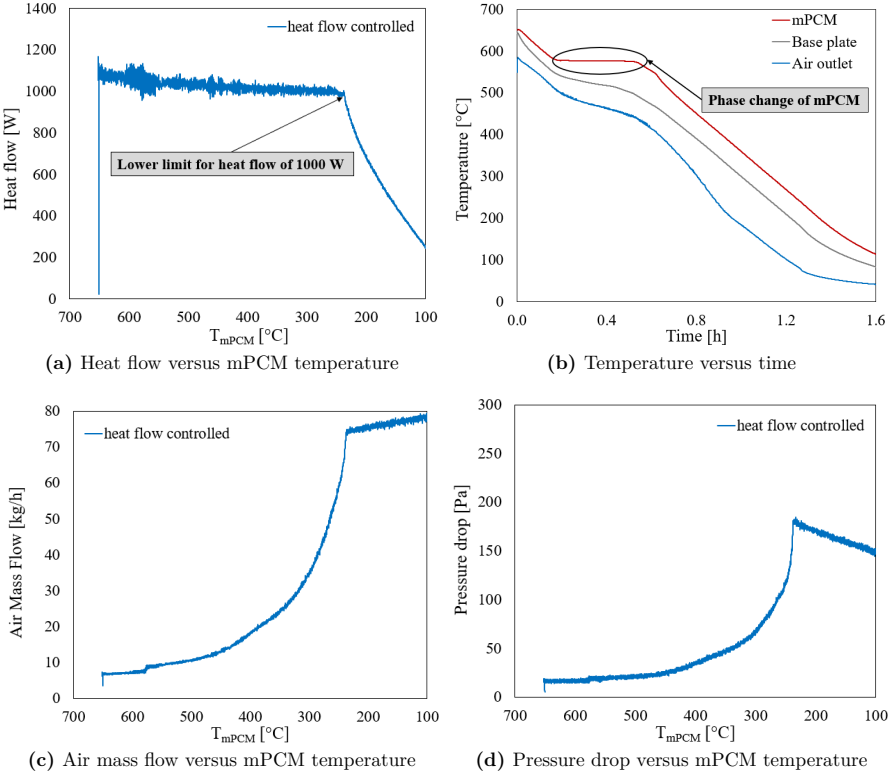


Figure 4.14: Heat extraction characteristics for constant heat output, adapted from Nees et al. [79]

As shown in Figure 4.14(c), the change in mass flow ranges from approximately 7–74 kg/h. This underlines the wide operating control range required for a constant thermal discharge. In other words, the mass flow required for the extraction of a heat flow of 1 kW at a storage material temperature of 650 °C is only about 15% of the mass flow needed for that of a temperature level of 240 °C.

By comparing the results with the investigations of the constant fan power rate, it is evident that the lowest $AlSi_{12}$ temperature, at which a heat flow of 1 kW can be achieved, differs by about 45 K. For a constant controlled heat flow, the lower limit for 1 kW is about 240 °C, whereas it is about 285 °C for a constant fan power rate of 100%. This difference can be explained by the sensible heat and temperature distribution in the components because the base plate cools down faster with higher discharge rates.

4.2.4.4 Efficiency of heat extraction

To evaluate the efficiency of the heat extraction system, the ratio of the fan's electric power consumption (P_{el}) to the total extracted heat flow ($\dot{Q}_{ex}$) is analyzed. The fan's electric power consumption was measured during the experiments and was about 25 W at 100% fan power. Figure 4.15 shows the ratio for three different fan power rates versus the mPCM temperature. In the liquid state of the mPCM, a ratio smaller than 0.7% was achieved for all fan power rates. Considering the high absolute value of the extracted heat, as shown in Figure 4.12(a), the heat extraction is highly efficient. In the solid state, at an mPCM temperature of 300 °C, the ratio is less than 2.3% at 100% fan power and less than 1% at 35% fan power. In both cases, the extracted heat is in the range of 1 kW.

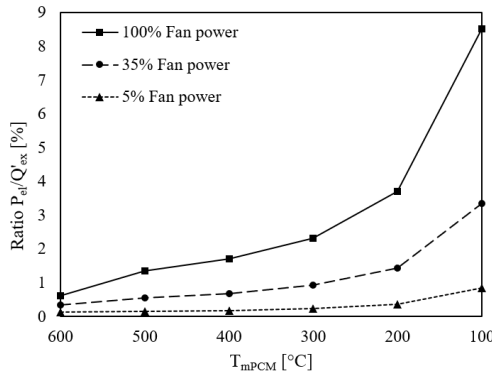


Figure 4.15: Ratio $P_{el} / \dot{Q}_{ex}$ versus mPCM temperature for different fan power rates, reproduced from Nees et al. [79]

At lower temperatures, the ratio clearly increases to up to more than 8%, as shown in Figure 4.15, and the extracted heat distinctly decreases. For temperatures below 200 °C, heat extraction becomes insufficient, and the amount of additional heat extracted by using a higher fan power is reduced, as shown in Figure 4.12(a). Thus, the heat extraction at lower temperatures is limited, and it cannot be overcome by increasing fan power or air mass flow.

For temperatures below 100 °C, the increase in heat extraction caused by a rise in fan power from 35% to 100% is smaller than the additional electric power consumed by the fan. Therefore, increasing fan power or mass flow is not beneficial when the storage temperature falls below a certain threshold.

Compared to the prototype design in Subsection 4.1.4, the $P_{el}/\dot{Q}_{ex}$ ratio in the experiment appears to be higher than the model's calculated outcome. However, there are two major differences between the experimental and design phases. First, the electric power consumption of the fan in the experiment includes the entire air section of the test bench, from the atmospheric inlet to the outlet. In the design phase, only the pressure

drop across the air heat exchanger was considered. Second, the experimental evaluation considers the mPCM temperature instead of the temperature at the air heat exchanger interface ($T_{w,AirHEX}$). This explains why the temperature in the experimental analysis is shifted and the electric power consumption is higher.

4.3 Conjugate heat transfer analysis of bench-scale prototype

This section presents a three-dimensional computational fluid dynamics (CFD) model of the bench-scale prototype that was investigated experimentally. Conjugate heat transfer (CHT) analysis allows for simultaneous heat transfer between solid and fluid domains. To validate the numerical model, the simulation results are compared with available experimental data. In order to elucidate the transitional character of the discharge, the conjugate model considers the phase change of the storage material, air flow in the fluid channels, and intermediate solid layers, including an analysis of thermal contact resistances. The resulting temperature distribution and heat flow are analyzed in the temperature range from 650 °C to 100 °C.

4.3.1 Model description

Fluid flow simulations are based on solving three fundamental governing equations: the continuity equation, the momentum equation, and the energy equation. These equations are mathematical formulations of the physical principles of conservation of mass, momentum, and energy. Together, they form a system of equations known as the Navier-Stokes equations. In CFD simulations, the Navier-Stokes equations are iteratively solved over a discretized fluid domain to calculate the physical quantities of the flow. Basic literature on the subject includes the books by Anderson [1] and Laurien and Oertel [64].

The conjugate heat transfer (CHT) analysis is modeled and simulated using the commercial software ANSYS Fluent, Version 2021 R2. The model's geometry and boundary conditions are carefully chosen to match the conditions of the experimental investigation in order to enable a validation of the model. Nevertheless, useful simplifications and assumptions are defined to simplify the model and to save calculation time and numerical resources.

The simulation model is based on the geometry of the bench-scale prototype presented in the previous Section 4.2. The schematic presentation of the prototype and experimental setup is illustrated in Figure 4.7, and the relevant dimensions of the air heat transfer section are summarized in Table 4.2. The validation of the model is based on the experimental data gained in the experiments.

For the numerical model, the geometry is simplified by modeling only one fluid channel, respectively two half channels, instead of the 28 channels in the actual geometry. The symmetrical structure of the geometry is therefore exploited by applying symmetry

boundaries under the assumption of equal fluid mass flow through each channel in the prototype. To simplify the geometry, the electric heaters and other inconsequential components, such as screws, are not modeled. The insulation around the mPCM casings is not simulated, and an adiabatic boundary is assumed in the domain. Heat losses to the environment are also neglected. Figure 4.16 shows the three-dimensional numerical model with selected boundary conditions.

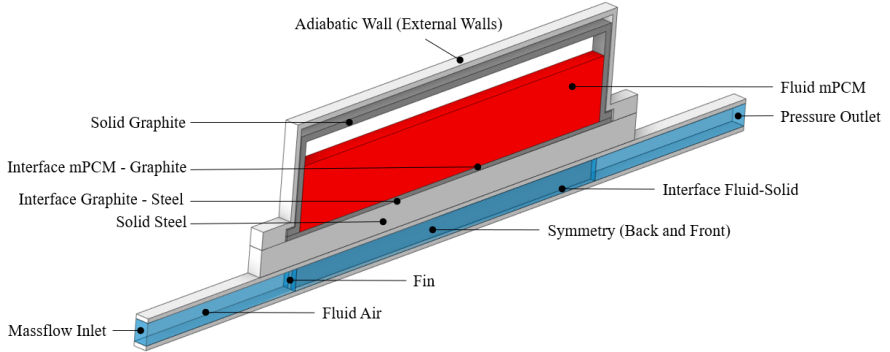


Figure 4.16: Three-dimensional numerical model with selected boundary conditions, reproduced from Nees et al. [76]

For the discretization of the simulation domain, a numerical mesh was created in Ansys Workbench Meshing, Version 2021 R2. Figure 4.17 shows visualizations of the mesh from a perspective view and a detailed view of the fluid channel. As hexahedral meshes offer high accuracy and necessitate relatively low computational resources, this mesh type was chosen. The near-wall boundary region was refined using inflation prism layers at the air fluid-solid interface.

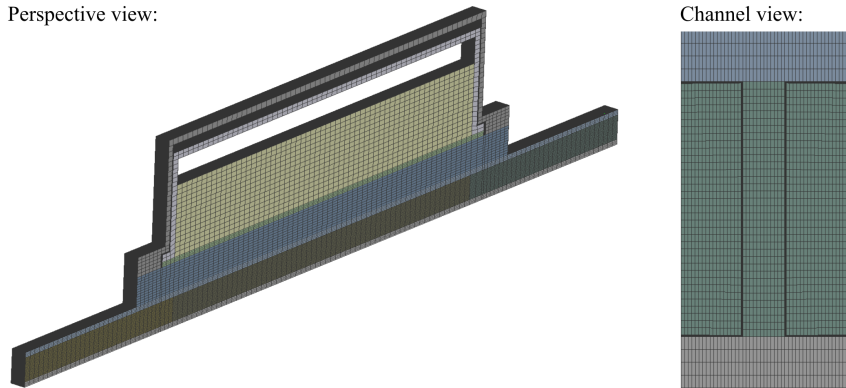


Figure 4.17: Visualizations of the mesh from a perspective view and a detailed view

Due to the mesh conformity constraint in Ansys Workbench Meshing, the solid volumes located above, below, and parallel to the fluid channel have the same mesh size as the fluid channel. This occurs even though a coarser mesh would be permissible for the solid volumes because no flow velocities are considered in solids and in the mPCM, and only the energy equation is solved.

To obtain an appropriate mesh size for an accurate numerical solution, a grid convergence study using a Richardson extrapolation was performed. Three different meshes were generated, a coarse, a medium, and a fine. The meshes have a refinement factor of 2 in cell size, and the simulation results of the relevant field variables were compared. The method of estimating the discretization error of the numerical mesh is presented in Appendix C. The study was carried out under steady state conditions with a mPCM temperature boundary. Based on that, the discretization error of the medium mesh is $\leq 0.31\%$ for the heat flow and the grid convergence index (GCI) is $\leq 0.39\%$. Thus, the medium mesh was selected for the simulations. Further grid size refinement does not yield significant improvement but requires tremendous additional computational effort. A variation in number of prism layers from 6 to 20 showed only marginal differences in heat flow. The wall distance of the first prism layer was chosen to achieve a y^+ value smaller than 1. Therefore, to save computation time, only six prism layers were considered for refining the fluid-wall boundary.

Table 4.3 summarizes the characteristics of the selected mesh for the transient CHT simulation. The mesh has a total of 469752 elements including six prism layers at the near-wall boundary of the air-fluid-solid interface. The selected mesh has an orthogonal quality of 0.969.

Type of mesh elements	Total number of elements	Elements in fluid air domain	Elements in mPCM domain	Elements in other domains (solids)
Hexahedral	469752	284426	41400	143926

Table 4.3: Characteristics of the selected mesh for the transient CHT simulation

The solid material properties, such as thermal conductivity, and specific heat capacity, are defined as a function of temperature. The fluid properties, such as density, thermal conductivity, specific heat capacity, dynamic viscosity, and kinematic viscosity, are temperature dependent. The solid material and fluid properties were selected according to Appendix A.

The convergence criteria for the solution were defined as follows: continuity ($1e-6$), momentum ($1e-6$), and energy ($1e-12$). For the solution to be considered converged at each time step, the given residuals must be achieved. This was the case for the numerical results shown.

In the transient simulation, only the temperature field is fully transient, whereas the transient phenomena in the flow field are not modelled. In this case, the fluid flow

problem is similar to a steady-state model in which the temperature boundary condition varies over time. Therefore, a sufficiently large time step can be selected. In this study, a time step of 10 seconds was chosen. The time step discretization error was studied empirically. The results showed that the solidification time and temperature plots of the mPCM, base plate, and air outlet remained consistent when the time step size was decreased.

At the inlet of the numerical domain, a constant air mass flow of 0.54 kg/h was set for the whole discharge time, which corresponds to a total mass flow of 15 kg/h in the experiments. The dedicated Reynolds number in the rectangular fluid channel varies between 530 and 1072, depending on the air bulk temperature and time of discharge. As the Reynolds number is much smaller than 2300, the flow is expected to be in the laminar regime. Comparing steady state simulations with the laminar solver and the $k - \omega$ SST turbulent solver showed no significant difference for the given flow rate. For this reason, and to save computational time, the laminar solver was selected for the transient CHT simulation, implying a laminar flow regime.

The initial temperature conditions for fluid and solid regions are selected based on experimental data. The initial mPCM temperature was set to 650 °C. The air temperature at the inlet boundary is also based on experimental data and decreases as discharge time progresses. Thermal radiation inside the fluid channel is modeled using the surface-to-surface radiation model. At the air fluid-solid interfaces, a thermally coupled wall boundary condition is defined.

For the numerical modeling of the phase change, the enthalpy-porosity model in ANSYS Fluent's solidification and melting model is selected. The enthalpy (H) of the mPCM is considered as the sum of its sensible heat ($h_{sensible}$) and latent heat (Δh_f):

$$H = h_{sensible} + \Delta h_f \quad (4.14)$$

The energy equation for the phase change is given as follows [4]:

$$\frac{\delta}{\delta t}(\rho H) + \nabla \cdot (\rho \vec{v} H) = \nabla \cdot (k \nabla T) + S \quad (4.15)$$

where H is the enthalpy, ρ the density, $\vec{v}$ the fluid velocity, k the thermal conductivity and S the source term. The liquidus and solidus temperatures are defined as 579 °C and 575 °C, respectively. These temperatures were selected to align with observations from the experiment and allow for good convergence in the simulation rather than defining a single phase change temperature of 577 °C. For the phase change of the mPCM, gravitational effects and natural convection flow within the mPCM volume are neglected, as the heat discharge is realized from the bottom. Therefore, heat conduction is the predominant mode of heat transfer within the mPCM volume. Additionally, volumetric changes due to phase changes are not modeled.

A temperature-dependent contact resistance is defined at the contact interface between the mPCM and the graphite using a shell conduction layer. The thermal contact resistance values are empirically determined under the constraint of fitting the discharge time and heat flow. The results of this study are presented in Table 4.4. At the graphite-base plate interface, a constant thermal contact resistance of $2.5 \text{ cm}^2\text{K/W}$ is defined using a thin-wall boundary condition. This assumption was made to align with the results from the simulation and the experiment and is within the range of common solid-to-solid thermal contact resistances in literature, as discussed in Section 2.6.

The resulting heat flow from the simulation is calculated using an energy balance. This considers the air mass flow, the heat capacity of the air calculated at the average mean temperature between outlet and inlet, and the temperature difference between the air temperatures at the aligned positions with the thermocouples in the experiment $T_{Air,0}$, $T_{Air,11,12,13}$ ($T_{Air,1}$), and $T_{Air,21,22,23}$ ($T_{Air,2}$).

4.3.2 Results and discussion

In order to validate the numerical model, the simulation results are compared to the experimental data using the same set of parameters, such as a total air mass flow of 15 kg/h . Figure 4.18(a) shows the temperature results for the mPCM (T_{mPCM}), base plate (T_{BP}), and outlet air ($T_{Air,2}$) against discharge time, obtained from the simulation and compared to the experimental data. The total time for the thermal discharge of the mPCM from a temperature of $650 \text{ }^\circ\text{C}$ to $100 \text{ }^\circ\text{C}$ differs by about 5% between the numerical model and the experiment. In general, the temperature curves show strong agreement.

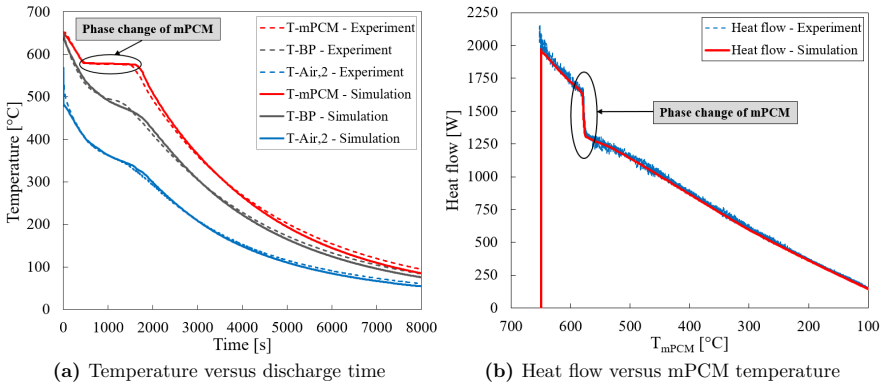


Figure 4.18: Comparison of simulation results for temperature and heat flow with experimental data, adapted from Nees et al. [76]

Figure 4.18(b) shows the heat flow versus the mPCM temperature, as obtained from the simulation and the experimental data. The heat flow curves are consistent within the temperature range of 650 to $100 \text{ }^\circ\text{C}$. In the liquid state, the simulated heat flow differs

by about 1–2% from the mean experimental values, except during the initial start-up phase at 650 °C, when the deviation is higher. In the solid state, the deviation from the simulation to the mean experimental values is about 2%. This deviation is within the 3 to 3.6% measurement accuracy of the experiment across the full temperature range, as detailed in Subsection 4.2.3. Therefore, the transient heat discharge simulation can be used to examine the heat extraction characteristics with sufficient accuracy, and the numerical model can be considered as validated.

The phase change occurs at around 577 °C, as seen in Figure 4.18(a) by the plateau of the mPCM temperature over time. As shown in Figure 4.18(b), the transition from liquid to solid is noticeable by the clear drop in heat flow caused by the increased thermal contact resistance at the mPCM-graphite interface.

Table 4.4 provides empirically determined values for thermal contact resistance at the mPCM-graphite interface. The empirical analysis showed that the time required for mPCM solidification and thermal discharge is highly sensitive to the definition of thermal contact resistances. It becomes apparent that the thermal contact resistance at the interface is temperature dependent. It is lowest in the liquid state of the mPCM, increases sharply during the phase change, and continues to rise gradually as the temperature decreases.

Interface	Temperature range [°C]	Thermal contact resistance [cm ² K/W]
mPCM-Graphite	650 to 575	0.7
	575 to 525	12.5 to 37
	525 to 300	37 to 60
	300 to 100	60 to 70
	<100	70

Table 4.4: Thermal contact resistances at mPCM-graphite interface based on empirical study, numbers reproduced from Nees et al. [76]

Considering the values reported in Table 4.4, the mPCM exhibits relatively low thermal contact resistance (0.7 cm²K/W) in its liquid state. This value is smaller than common solid-to-solid thermal contact resistances reported in the literature for contact pressures of 100 kN/m² or lower, as discussed in Section 2.6. These values indicate good heat transfer across the mPCM-container interface in the liquid state. In contrast, the thermal contact resistance values in the solid state (for temperatures below 575 °C) reach up to 70 cm²K/W at 100 °C. These values are relatively large compared to common solid-to-solid thermal contact resistances reported in the literature, as discussed in Section 2.6. Thus, the thermal contact resistance increases by a factor of 100 from liquid to solid mPCM across the investigated temperature range, thereby deteriorating heat transfer across the mPCM-container interface.

The probable reasons for the relatively high thermal contact resistance in the solid state, as compared to the values reported in the literature, are primarily macroscopic effects in the mPCM and the absence of a contact pressure boundary. These factors contribute to thermal distortion and higher surface roughness. As discussed in Section 2.6, literature values typically consider two flat surfaces with a contact pressure boundary. These are ideal conditions, which are not the case here. Therefore, the values reported in Table 4.4 do not apply to all AlSi₁₂-Graphit solid-to-solid interfaces, but are characteristic for the dimensions of the prototype investigated.

Figure 4.19 illustrates the liquid mass fraction from a cross-section of the mPCM volume during its transition from liquid to solid. The liquid mass fraction directly quantifies the amount of latent heat in the phase change material. A liquid mass fraction of 1 indicates a fully liquid state, while a liquid mass fraction of 0 indicates a fully solid state. Regions where the liquid fraction is between 0 and 1 are called mushy zones. Figure 4.19(a) illustrates the initial phase change at $t = 291$ s, and Figure 4.19(b) shows that, at $t = 651$ s, the liquid fraction first decreases in the mPCM volume near the air inlet. The upper right regions, which are farther from the air inlet, remain fully liquid. Figure 4.19(c) shows that at $t = 1011$ s, the volume close to the air inlet begins to solidify completely while large regions remain in the mushy zone. In Figure 4.19(d) at $t = 1771$ s, the entire storage material volume is in a solid state. Thus, according to the simulation, the storage material solidifies from complete liquid to complete solid in 1480 seconds. Despite the high thermal conductivity of the AlSi₁₂ storage material and the prototype's relatively small overall dimensions, the direction of solidification is noticeably impacted by the orientation of the air discharge flow.

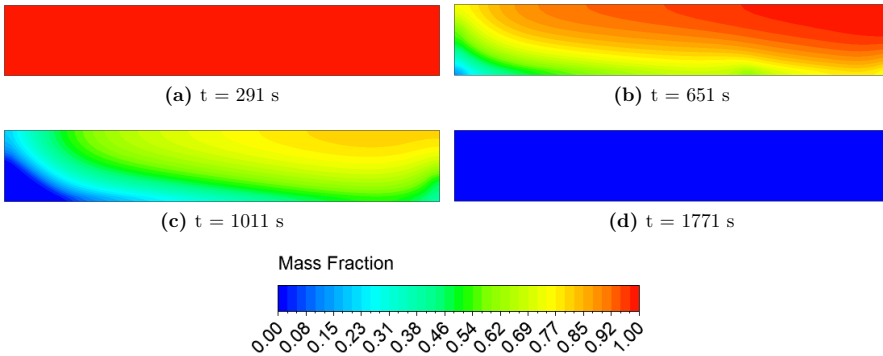


Figure 4.19: Liquid mass fraction of mPCM cross-section at various stages of solidification quantifying the discharge of latent heat, reproduced from Nees et al. [76]

Figure 4.20 shows the temperature distribution across the fluid channel, mPCM, base plate, graphite, and steel container during the phase change at $t = 1011$ s. Based on the temperature contours, there are strong temperature gradients in the air fluid and the

solid parts. The air temperature gradually increases from the inlet to the outlet in the x-direction, creating an uneven temperature distribution in the flow direction among the adjacent parts and the mPCM. Consequently, the heat flux is higher at the channel inlet than at the outlet. The resulting temperature gradient in the mPCM is approximately 6 K.

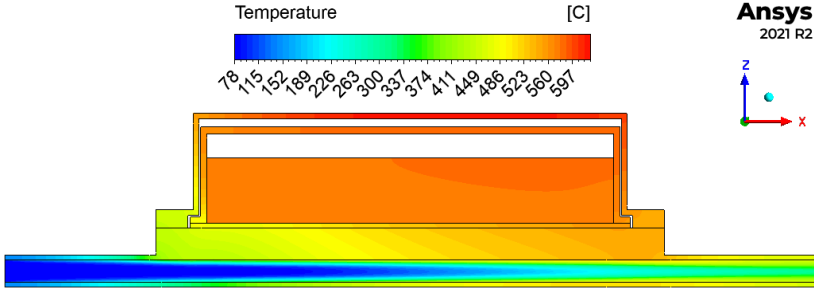


Figure 4.20: Temperature distribution at cross-section after $t = 1011$ s thermal discharge, reproduced from Nees et al. [76]

Figures 4.21(a)–(b) show the temperature and velocity contours viewed from the outlet of the fluid channel in the air heat transfer section at two different time steps.

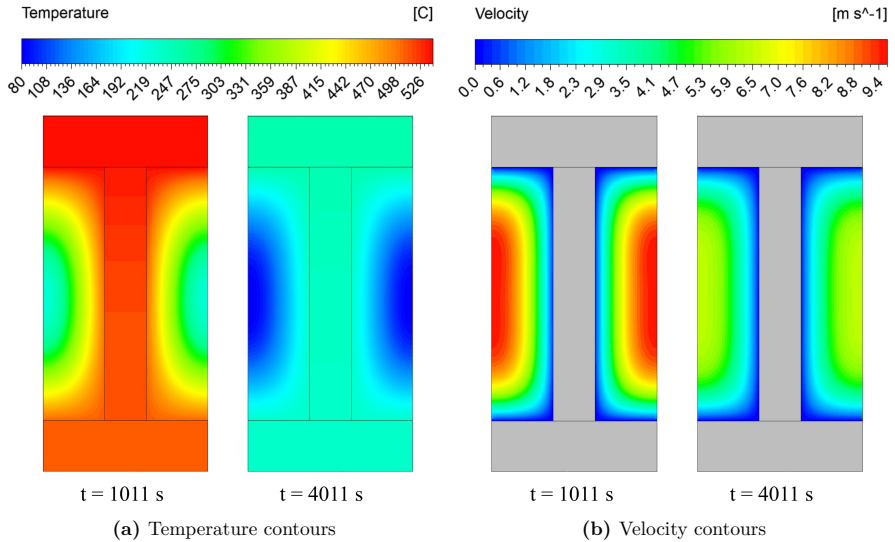


Figure 4.21: Temperature and velocity contours at channel outlet in the air heat transfer section at various time steps

As seen in Figure 4.21(a), temperatures in both the solid and fluid domains fall significantly from $t = 1011$ s to $t = 4011$ s. There are strong temperature gradients in the fluid from the center towards the wall for both time steps. As the flow is fully laminar, there are no fluctuations in the temperature field. Figure 4.21(b) shows that the fluid velocity in the channel decreases with an increase in discharge time. At $t = 1011$ s, the peak velocity is 9.6 m/s, whereas at $t = 4011$ s, it is only 6.4 m/s for the same mass flow rate. This is due to the strong temperature dependence of air density, which decreases as temperature increases. Furthermore, the fluid velocity is not distributed evenly across the channel. There is a bulk flow in the center with higher velocities, and the velocities decrease sharply near the walls.

4.4 Experimental investigation of application-scale prototype

4.4.1 Prototype design and test bench

Figure 4.22 illustrates a schematic representation of the application-scale prototype and experimental setup with selected measurement points. The storage is based on a box-shaped container design that accommodates the storage material AlSi_{12} . The container consists of side walls coated with aluminum oxide, an uncoated steel top plate, and a bottom plate made out of fiber ceramics, which acts as the interface layer between mPCM and base plate. The general prototype design and the encapsulation concept have also been reported in a DLR project report [87]. The base plate installed on the bottom contains electric heaters for charging. Fluid channels for discharging were brazed to the base plate bottom and heat extraction was realized by forced convection of ambient air using a controllable fan. The test bench contains thermocouples for temperature measurement of the mPCM, base plate, and air section, as well as air mass flow and pressure drop measurement devices. The storage system is surrounded by microporous insulation to minimize heat losses.

The fan utilized in this investigation was an Ebm-Papst EC-radial blower (model G3G133-RO15-04), with an individual power of 85 W and a variable fan speed. A thermal probe of type Höntzsch TA10 was installed in the air section for the mass flow measurement. The KIMO Model C310 was used to measure the differential pressure. Therefore, the pressure at the air inlet and outlet was averaged across four measuring points perpendicular to the flow direction, which were evenly distributed across the width.

For temperature measurement, Type-N thermocouples with diameters of 1 mm and 1.5 mm were installed. As for the bench-scale prototype, the thermocouples were calibrated beforehand for a temperature range from ambient to 800 °C using a custom-made calibration setup with Pt100 and type S thermocouple reference sensors. Eleven thermocouples were installed at different positions in terms of length, width, and height inside the mPCM. In the base plate, twelve thermocouples were installed at various positions. At inlet and

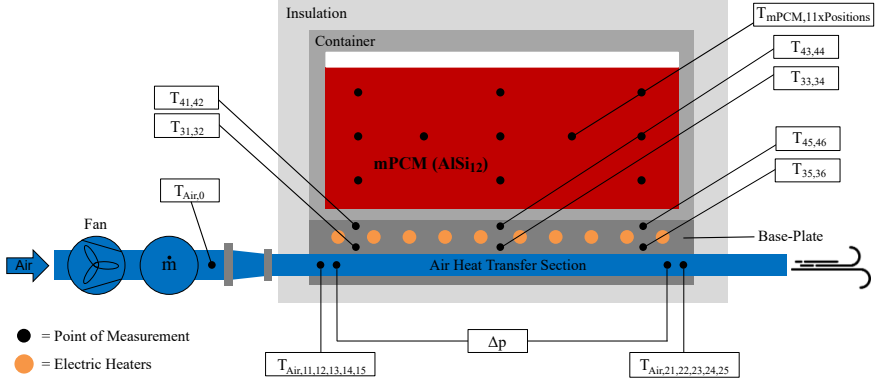
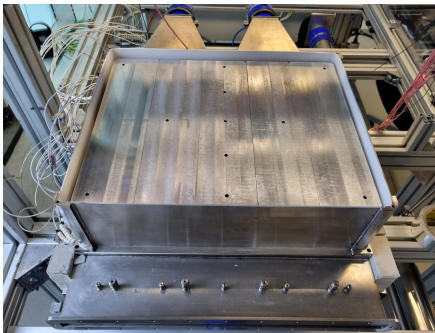


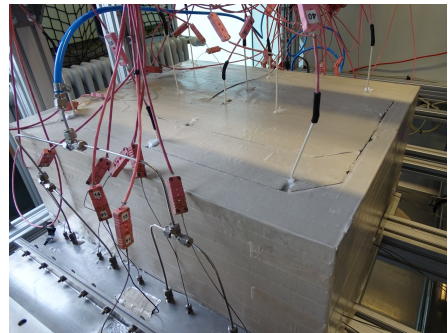
Figure 4.22: Schematic representation of the application-scale prototype and experimental setup

outlet of the air heat transfer section, five thermocouples were installed each, which were evenly distributed across the width. Ten electric heaters (10 mm diameter, 2000 W each, from Türk+Hillinger) were installed for the charging process. The experimental data were processed using a data-acquisition system from Delphin Technology AG, employing the Expert Logger 400 hardware and ProfiSignal software.

Figures 4.23(a)–(b) show the installed storage system prototype in the lab with and without insulation. Figure 4.23(a) presents the AlSi₁₂ storage material inside the container before commissioning. Figures 4.24(a)–(b) illustrate the installed base plate, made of 1.4828 high-temperature stainless steel, with air inlet and outlet channels and connection to the inlet distributor.



(a) Setup with container and mPCM



(b) Insulated prototype in test bench

Figure 4.23: Test bench with application-scale prototype, adapted from [87]

The fins of the air heat transfer section, made of 1.4404 stainless steel, were brazed to the base plate using nickel filler material during a vacuum brazing process, as shown in

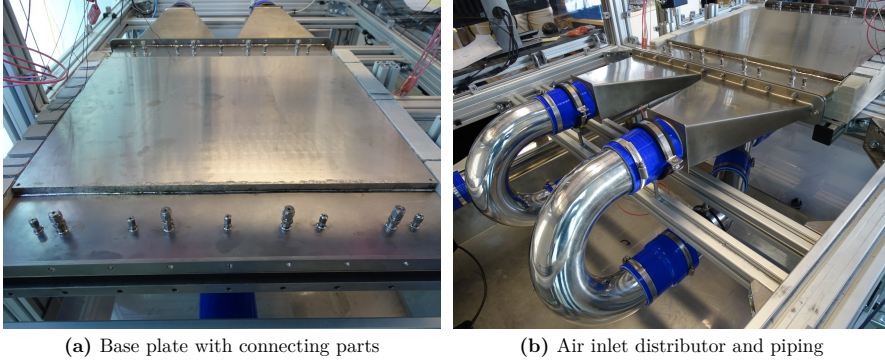


Figure 4.24: Base plate with connecting parts and air inlet line, adapted from [87]

Figure 4.25(b). The channels were closed using a cover steel plate on the bottom. The in- and outlet channels, compression fittings, and flanges were laser welded.

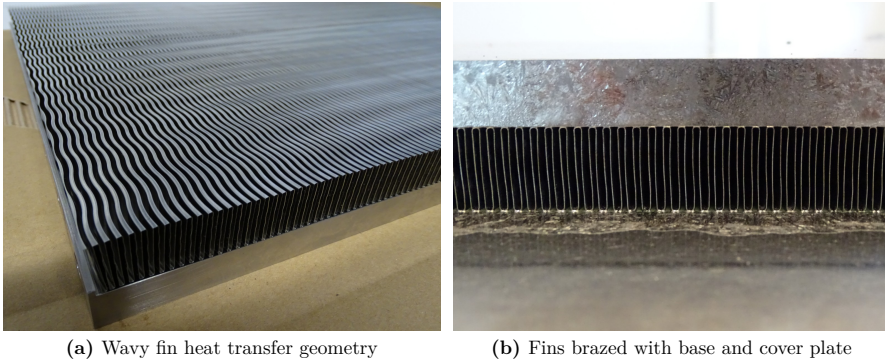


Figure 4.25: Air heat transfer geometry used in prototype before and after brazing

The basic data of the investigated application-scale storage prototype are summarized in Table 4.5. The physical properties of the AlSi_{12} storage material are provided in Table 2.1 and Appendix A.

The application-scale prototype differs from the bench-scale prototype mainly in its size and has a much greater thermal storage capacity. It also differs in the construction of the container and the fluid channel geometry. The thermal storage capacity is by a factor of 15.7 higher. Additionally, the gravimetric and volumetric energy densities are higher for the application-scale prototype compared to the bench-scale prototype because the weight of the AlSi_{12} accounts for a higher proportion of the total weight, while the proportion of insulation is lower.

Item	Specification
Total storage capacity	25 kWh (100 – 650 °C)
Total dimension (with insulation)	0.53 x 0.635 x 0.288
Total dimension (without insulation)	0.45 x 0.555 x 0.206
Weight total	134.9 kg (of which 12.3 kg is insulation)
Base area AlSi ₁₂ (A_{ref})	0.25 m ²
Weight AlSi ₁₂	68.5 kg
Electric heaters installed	10 x 2000 W
Gravimetric energy density	185.3 Wh/kg (203.9 Wh/kg without insulation)
Volumetric energy density	257.9 Wh/L (485.9 Wh/L without insulation)

Table 4.5: Basic data of the application-scale storage prototype, numbers based on data from [87] and additional information

Table 4.6 summarizes the detailed air channel geometry. As shown in Figure 4.25(a), the fins take the shape of lazy ruffled waves with a pitch of 25.4 mm and amplitude of 1.27 mm. Here, the amplitude refers to the distance between the peak and the valley of a wave. The specific design was selected to achieve low pressure drop and high heat transfer for high-efficiency thermal discharge. The design philosophy and selection process are thoroughly discussed in Section 4.1.

Item	Specification
Type	Lazy ruffled wavy fin design
Fin wave characteristics	1.27 mm amplitude, 25.4 mm wave pitch
Base area dimension with channels	450 x 555 mm (L x W, Ratio L/W=0.81)
Air channel geometry	20 x 1.5 mm (b x a), 326 parallel channels
Air channel hydraulic diameter	2.79 mm
Fin thickness	0.2 mm

Table 4.6: Detailed geometry of the air heat transfer section

4.4.2 Experimental procedure

The schematic procedure for charging and discharging is similar to the bench-scale prototype experiments illustrated in Figure 4.11. In the first step, the storage was charged using the installed electric heaters. In the second step, the storage was discharged through forced convection of ambient air with the installed fan. Therefore, the electric heaters were then turned off.

During the initial phase of the experiments, the base plate was heated up to temperatures higher than 577 °C in order to melt the storage material. However, due to leakage of the mPCM from the container housing during the initial melting, experiments involving phase changes had to be stopped. Consequently, only experiments without phase change

could be performed thereafter. Therefore, the base plate was heated to temperatures not exceeding 550 °C. The lowest operating temperature for the base plate was 100 °C. A limited number of experiments were carried out by applying different constant rates of fan power, mass flow, and heat output to study the air discharge characteristics.

The heat flow $\dot{Q}_{ex}$ was calculated from the measured data using the following energy balance:

$$\dot{Q}_{ex} = \dot{m}c_p(\bar{T}_{Air,21,22,23,24,25} - T_{Air,0}) \quad (4.16)$$

Where $\dot{m}$ is the measured air mass flow, c_p is the heat capacity of the air at the average mean temperature between outlet and inlet, and $\bar{T}_{Air,21,22,23,24,25}$ and $T_{Air,0}$ are the measured average air temperatures at outlet and inlet. The heat flux $\dot{q}_{ex}$ is calculated applying Equation 4.12.

The $T_{w,AirHEX}$ temperature at the bottom of the base plate, which is the interface to the air heat exchanger, is derived from the measured data as follows:

$$T_{w,AirHEX} = \bar{T}_{31,32,33,34,35,36} - 4/8(\bar{T}_{41,42,43,44,45,46} - \bar{T}_{31,32,33,34,35,36}) \quad (4.17)$$

The average base plate temperature at an elevation of 4 mm from the bottom of the base plate ($\bar{T}_{31,32,33,34,35,36}$) was calculated from the six thermocouple readings T_{31} , T_{32} , T_{33} , T_{34} , T_{35} , and T_{36} . The average base plate temperature at an elevation of 12 mm from the bottom of the base plate ($\bar{T}_{41,42,43,44,45,46}$) was calculated from the six thermocouple readings T_{41} , T_{42} , T_{43} , T_{44} , T_{45} , and T_{46} . The thickness of the base plate was 16 mm.

4.4.3 Measurement uncertainty

It is essential to verify the quality of the experimental results by analyzing the measurement uncertainty of the equipment used. Figure 4.22 shows the relevant measurement points, which include temperature, mass flow, and differential pressure. The combination and propagation of uncertainties were conducted according to the JCGM guidelines [41]. Supplemental data to this subsection are given in Appendix B.

The thermal probe of type Höntzsch TA10 has a specified accuracy of 2% from the measured mass flow plus 0.02 m/s in the measured velocity according to the manufacturer's data sheet. Considering the inside diameter of the measuring pipe and the lowest mass flow in the experiments, the total accuracy sums up to <2.6%. The sensor was already calibrated ex-factory.

The KIMO Model C310 was installed to accurately measure the pressure drop (Δp) across the air heat transfer section. According to the manufacturer's data sheet, the accuracy is ± 2 Pa and $\pm 0.2\%$ of the measured value. As the gauge was mounted at a sufficient

distance from the hot storage and connected via non-insulated capillary tubes, the effects of temperature were negligible.

To improve the precision of temperature measurements, type N thermocouples were calibrated using a custom calibration setup with PT100 and type S thermocouple reference sensors. Calibration was carried out in the final test bench setup with a Carbolite Gero laboratory chamber furnace. The thermocouples were calibrated for a temperature range from ambient to 800 °C. The total measurement uncertainty is a result of the combination of the uncertainty of the PT100 and type S reference sensor, the calibration setup, and the regression error. It is less than ± 1.7 K for all thermocouples over the entire temperature range.

The calculation of the extracted heat flow is based on the measurements of $\dot{m}$, $T_{Air,0}$, $T_{Air,21}$, $T_{Air,22}$, $T_{Air,23}$, $T_{Air,24}$, and $T_{Air,25}$ and is therefore subject to the propagation of uncertainties. For the 10 kW design case, the absolute uncertainty in the heat flow is between 280 W and 105 W within the investigated temperature range. This corresponds to a relative uncertainty of between 2.6% and 2.8%.

4.4.4 Results and discussion

During the first melt-up experiment, mPCM leaked between the bottom and side walls of the container, causing the container housing to fail. The charging process was immediately stopped, and the fan was activated to cool the mPCM and prevent further leakage. The development and engineering of the container housing were outside the scope of this thesis. At the time of the initial leakage, the mPCM had only partially melted. High temperature differences were observed between the mPCM and the base plate, indicating high thermal contact resistance. Consequently, only experiments without phase change of the mPCM could be performed in the following.

In contrast to the bench-scale prototype, no conjugate CFD heat transfer analysis of the transient thermal discharge was performed for the application-scale prototype. Due to mPCM leakage during the initial experiment, the available data are insufficient for analyzing the transient thermal discharge from the mPCM to the air discharge fluid, including the phase change of the mPCM and the evaluation of the thermal contact resistance. Consequently, neither validation of such a model nor an empirical study of the thermal contact resistance is possible.

Therefore, this subsection's results and discussion focus only on the characterization of the air heat exchanger instead of analyzing the complete storage discharge performance. The experimental results are compared instead with the outcome of the heat transfer calculation model presented in Subsection 4.1.2 to validate the model.

4.4.4.1 Constant mass flow rate

Figures 4.26(a)–(b) show the experimental heat flow characteristics for a constant air mass flow rate of 125 kg/h, as well as the results of the developed heat transfer calculation

model. The model is presented in Subsection 4.1.2. The heat flow reaches a maximum of approximately 19 kW at 550 °C and decreases as the temperature falls. At 100 °C, the heat flow is about 2.5 kW. The measured air pressure drop peaks at around 800 Pa. Meanwhile, the air mass flow remains constant across the plotted temperature range, as illustrated in Figure 4.26(b).

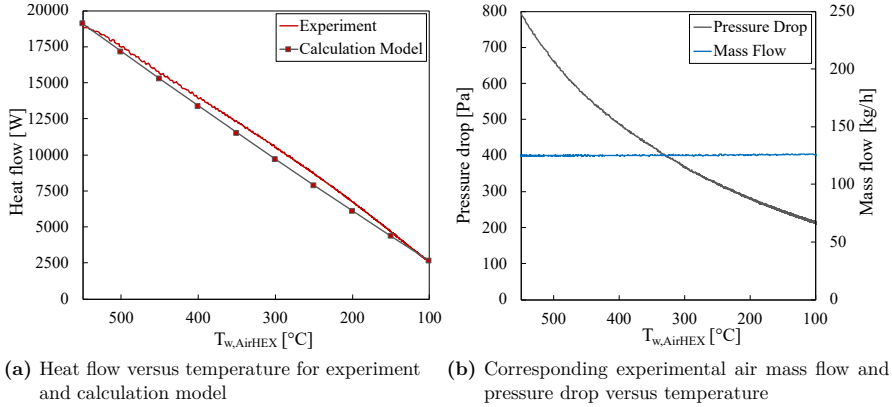


Figure 4.26: Experimental heat flow characteristics for a constant air mass flow rate of 125 kg/h and comparison with calculation model

As shown in Figure 4.26(a), the heat flow curves of the experiment and the calculation model are consistent, with only slight deviations. The maximum deviation between the calculated and experimental data is less than 10% within the plotted temperature range. The average deviation is approximately 5%. These deviations are likely due to the model's assumption of a constant wall temperature ($T_{w,AirHEX}$) in the flow direction.

4.4.4.2 Design heat flow rate

In real-world applications, it is important to achieve and control the design heat flow rate in a large operating temperature range. This is one of the main functional requirements for the thermal discharge system. Figures 4.27(a)–(b) show the experimental heat flow characteristics for a controlled design heat flow of 10 kW, as well as the results of the developed heat transfer calculation model. The model is presented in Subsection 4.1.2. Based on the experimental data, a heat flow of 10 kW, respectively 4 W/cm², is achieved until a lower temperature limit of $T_{w,AirHEX}$ of approximately 205 °C is reached. However, for temperatures lower than that, the heat flow decreases despite an increase in air mass flow and a fan power rate of 100%.

As shown in Figure 4.27(b), the measured air pressure drop peaks at approximately 500 Pa, and the air mass flow reaches a maximum of 240 kg/h. Despite an increase in air mass flow, the pressure drop decreases between 400–275 °C and 205–100 °C. This

occurs because the decreasing air temperature has a greater effect on the pressure drop than the increase in mass flow for the given case. The decrease in temperature leads to an increase in air density and a decrease in velocity, resulting in a reduction in pressure drop, according to Equation 2.28.

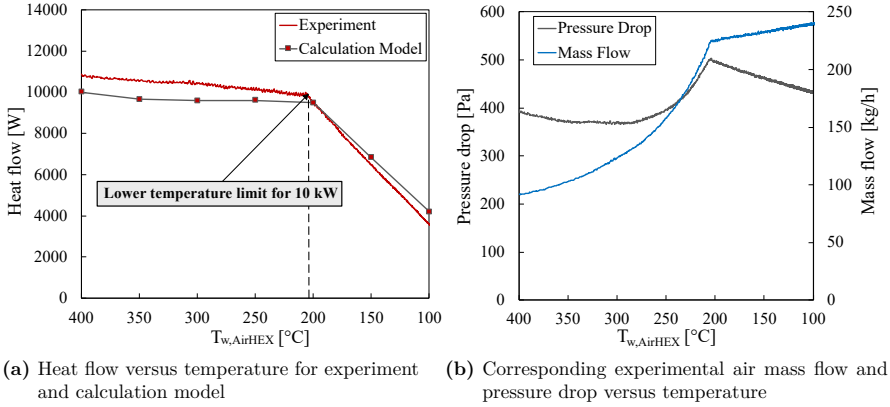


Figure 4.27: Experimental heat flow characteristics for design case of 10 kW and comparison with calculation model

As shown in Figure 4.27(a), the heat flow curves of the experiment and the calculation model are consistent, with only slight deviations. Except for temperatures close to 100 °C, the maximum deviation between the calculated and experimental data is less than 9% within the plotted temperature range. The average deviation is approximately 7%.

The accuracy of the calculation model results, as well as the uncertainty of the experimental measurements, implies that the model can be considered validated and used to estimate the heat flow. The model provides a relatively simple and fast way to estimate and design future prototypes. A more accurate estimation can be achieved by applying a conjugate CFD analysis, as presented in Section 4.3. However, this method is much more extensive. The model's validation implies that the parameter study in Section 4.1.3 is meaningful, and that the prototype geometry selection process is based on a verified conceptual approach.

4.4.4.3 Efficiency of heat extraction

The efficiency of the heat extraction system was evaluated by analyzing the ratio of the fan's electric power consumption (P_{el}) to the total extracted heat flow ($\dot{Q}_{ex}$). The fan's electric power consumption was measured during the experiments and was about 85 W at 100% fan power. Figure 4.28 shows the $P_{el} / \dot{Q}_{ex}$ ratio versus temperature for the 10 kW design heat flow rate discussed in the previous Subsection 4.4.4.2. At the lower temperature limit to maintain a heat flow of 10 kW, the ratio is approximately 0.9%. For

temperatures higher than that limit, the ratio continuously falls; for temperatures lower than that limit, the ratio gradually rises. At 400 °C, the ratio is approximately 0.3%, whereas it is about 2.4% at 100 °C. These results demonstrate the high efficiency of the developed heat extraction concept.

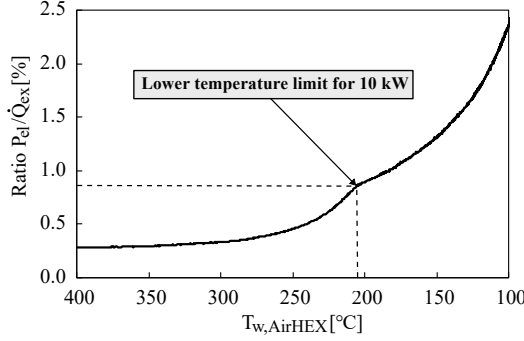


Figure 4.28: Ratio $P_{el} / \dot{Q}_{ex}$ against temperature for design heat flow rate

5 Multi-stage boiling discharge – Development and investigation

In this chapter, the novel multi-stage boiling heat discharge concept, introduced in Chapter 3, is further developed and investigated. Therefore, a prototype is constructed, experimentally investigated, and the results are analyzed and discussed. Based on these results, a heat transfer calculation model is derived further on. Parts of Sections 5.2 and 5.3 of this chapter have been published in Nees et al. [80], and a patent for the concept has been filed [74].

5.1 Design philosophy and simulation

5.1.1 Multi-stage design philosophy

The general idea of the multi-stage heat discharge concept has already been presented in Chapter 3. A schematic representation of the concept showing the components and the arrangement of the stages is shown in Figure 3.7, and the operating procedure for start-up and shut-down is illustrated in Figure 3.8.

The multi-stage boiling concept consists of two distinct boiling stages integrated into the base plate of the heat storage system. A schematic representation of the Stage 1 and 2 channel concept is shown in Figure 5.1. Stage 1 uses tubes that are mechanically decoupled from the base plate. The fluid channels are located in inner tubes that are inserted into holes in the base plate. A small gap between the base plate and the inner tube creates a thermal contact resistance that reduces the potential heat flow to prevent a boiling crisis. Heat transfer across the gap is by conduction and radiation, while natural convection can be neglected due to the small gap size. The gap is filled with ambient air. Heat transfer across the inner tube wall is by solid convection only. Stage 2 uses direct contact of the working fluid with the base plate. In this case, the holes in the base plate serve directly as fluid channels. In both stages, heat extraction is accomplished by forced convection and flow boiling in the channels.

The purpose of the Stage 1 design is to avoid the boiling crises with the onset of film and transition boiling at large temperature differentials, which results in improved performance and controllability of heat transfer. The thermomechanical decoupling and the limitation of the maximum heat flux can significantly reduce the thermomechanical loads during the start-up phase and operation at high temperature differentials. At the same

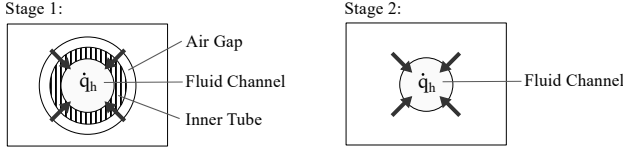


Figure 5.1: Schematic representation of Stage 1 and Stage 2 channel concepts

time, the Stage 2 design allows for high specific heat transfer capacity to be achieved at low temperature differentials.

Figure 5.2 shows the critical heat flux of water at 1 bar versus the vapor mass fraction for various flow rates and an inner channel diameter of 3 mm. The values are based on the 1995 look-up table for critical heat flux in tubes [27] and are corrected for the tube diameter using Equation 2.43. Although the values in the tables are given for vertical upstream flow in tubes, they are also valid for horizontal tubes at higher mass flow rates [34]. At lower mass flow rates, gravity-driven flow separation occurs, and therefore the critical heat flux tends to be lower than for vertical tubes. The negative vapor mass fraction on the horizontal-axis represents subcooled liquid with the corresponding enthalpy fraction. The dashed line in the figure indicates a constant heat flux of 25 W/cm^2 . Staying below this value means that no boiling crisis will occur for vapor mass fractions up to $\dot{x}$ greater than 0.9. In other words, the nucleate boiling regime would be predominant for a large range of vapor mass fractions. According to Table 2.4, the critical heat flux is reached at relatively small temperature differences of about 30 K between the wall and the saturation temperature of the fluid. Therefore, the boiling crisis occurs over a wide temperature range when the discharge is started at an elevated temperature level of more than 600°C .

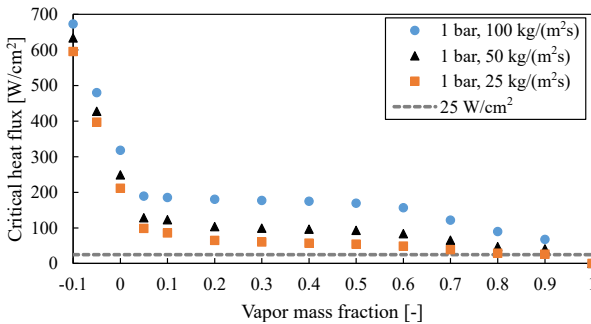


Figure 5.2: Critical heat flux of water at 1 bar versus vapor mass fraction for various flow rates and an inner channel diameter of 3 mm. The dashed line indicates a heat flux of 25 W/cm^2 . Values are based on the 1995 look-up table for critical heat flux in tubes [27] and Equation 2.43 for correction of the tube diameter.

Figure 5.3 shows the resulting heat flux across the air gap and the inner tube wall for various gap widths versus the temperature $T_{w,S1}$ of the surrounding base plate. An air gap of 0.15 mm shows heat fluxes of less than 25 W/cm² over the entire temperature range. Smaller gaps show higher heat fluxes, while larger gaps show lower heat fluxes. An air gap of 1 mm provides less than 7 W/cm² over the entire temperature range, or only about 17% of the heat flux of an air gap of 0.15 mm. Therefore, a gap width of 0.15 mm seems to be a good fit in terms of sufficiently high performance, but high enough thermal resistance to be below the critical heat flux of the boiling crisis.

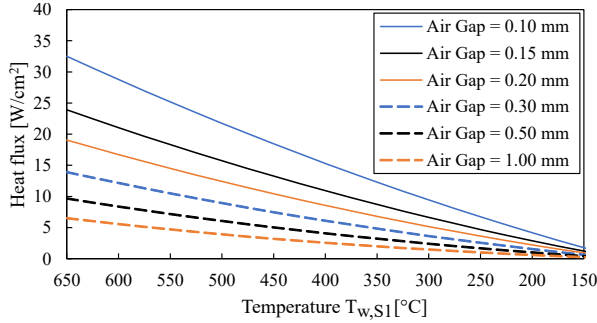


Figure 5.3: Resulting heat flux across air gap and inner tube wall for various gap widths versus temperature (inside fluid channel wall temperature 110 °C, stainless steel tube, inside channel diameter 3 mm, outside tube diameter 4 mm)

Moreover, limiting the maximum heat flux also protects against unintentionally getting very high local heat fluxes. As Figure 5.2 illustrates, heat fluxes of up to several hundred watts per square centimeter are possible with subcooled liquid. These high heat fluxes are prone to causing damage due to induced high thermomechanical stresses and fatigue. Therefore, an air gap width of 0.15 mm is selected for the prototype development, as further shown in the next sections.

Owing to manufacturing tolerances, accuracy during mechanical installation, and movement due to thermal expansion while heating up and cooling down the storage system, an equal air gap width of Stage 1 over the entire surface area cannot be guaranteed. Moreover, slight contact between the inner tube and the base plate is possible. However, as the drill hole in the base plate and the inner tube have a circular geometry with different dimensions, the theoretical overlapping interface is minimal. Even a 10% overlap in the circumferential direction combined with a 10% overlap in the flow direction, e.g. due to tube bending, would result in a total overlap of only 1% of the total available surface area. In addition, a common solid-to-solid thermal contact resistance at the overlapping surface would further limit heat transfer at the contact interface. In that case, a slight increase in heat flux is expected, but the general design philosophy of the multi-stage concept remains valid.

Nevertheless, a high accuracy of the drill hole and the inner tube is important to achieve

the desired heat flux as given in Figure 5.3. For the accuracy of the concept, a maximum manufacturing tolerance of ± 0.05 mm for both the drill hole and the outside diameter of the inner tube is considered sufficient.

5.1.2 Simulation of 2D-temperature profile

In order to analyze the resulting wall temperature in Stage 1 and Stage 2 channels under various boundary conditions, a simulation model is set up in ANSYS Fluent, Version 2023 R2. For the numerical model, the geometry is simplified by simulating only one fluid channel of Stage 1 and Stage 2 and only a small segment of the real prototype configuration. The prototype geometry is described in more detail in the next section. Based on this, the modeled fluid inside channel diameter for Stages 1 and 2 is 3 mm, the height of the base plate is 12 mm, the wall thickness of the inner tube is 0.5 mm, and the air gap width is 0.15 mm. Figure 5.4 shows the quasi two-dimensional numerical model with selected boundary conditions.

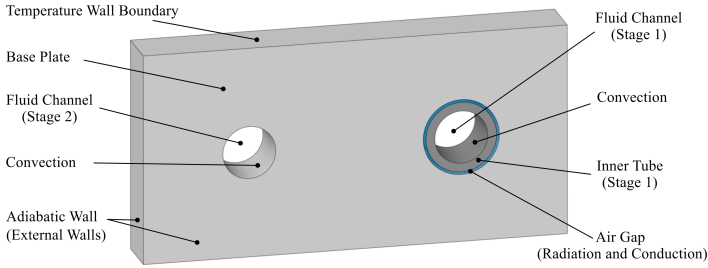


Figure 5.4: Numerical model with selected boundary conditions

The stationary model has a temperature wall boundary on the top side. Heat transfer in the channels is modeled by a convection boundary with a given heat transfer coefficient and fluid temperature. The air gap is considered by conduction and radiation heat transfer. Therefore, the surface-to-surface radiation model is applied. An adiabatic boundary is assumed in the domain for all exterior walls except the top wall. The thermal conductivity of the air and solid components is defined as a function of temperature. For meshing, polyhedral meshes were used with sufficiently small cell sizes.

A high temperature case and a low temperature case were chosen for the simulation to show the effect on the temperature profile and the wall temperatures. In both cases, a coupled operation of Stages 1 and 2 is defined. The working fluid is water at atmospheric pressure with a saturation temperature of 100 °C assumed in this model. For the high temperature operation case, a temperature of 550 °C was selected at the top wall of the base plate. The fluid temperature in Stages 1 and 2 was set to 100 °C. As film boiling is expected in Stage 2, the heat transfer coefficient was set to 300 W/(m²K). This is an approximate value determined using the equations from Subsection 2.5.4. In Stage 1,

no film boiling is expected and the heat transfer coefficient was set to $10,000 \text{ W}/(\text{m}^2\text{K})$, which is an approximate value determined using the equations from Subsection 2.5.2.

For the low temperature operation case, the temperature wall boundary at the top wall of the base plate was 180°C . In Stage 2, a heat transfer coefficient of $10,000 \text{ W}/(\text{m}^2\text{K})$ and a fluid temperature of 100°C were assumed. As the inner tube temperature of Stage 1 is expected to be too low to maintain boiling, a single-phase liquid heat transfer coefficient of $800 \text{ W}/(\text{m}^2\text{K})$ and a fluid bulk temperature of 70°C were assumed.

Figures 5.5(a)–(b) show the resulting temperature profiles for coupled operation of Stage 1 and Stage 2. For visibility reasons, the temperature scale in Figures 5.5(a) and 5.5(b) is different.

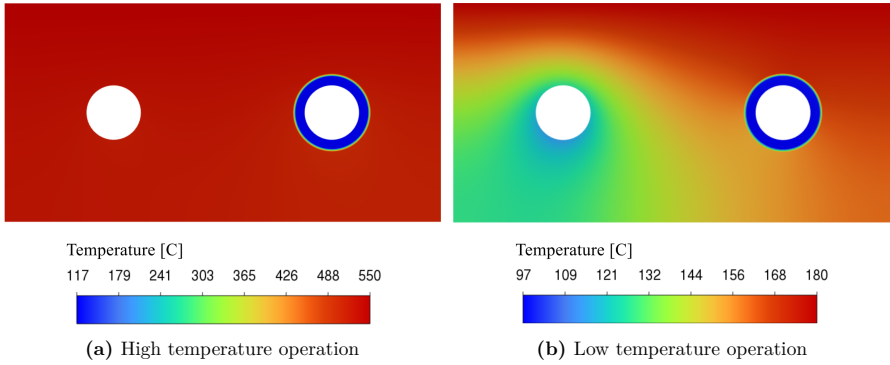


Figure 5.5: Temperature profile for coupled operation of Stage 1 and Stage 2 for high and low temperature regime

Figure 5.5(a) shows the results for the high temperature operation case. A steep temperature drop is visible across the air gap from the base plate to the inner tube for Stage 1. This is caused by the high thermal contact resistance of the air gap. The average surface wall temperature of the interface between the inner tube and the fluid is 117°C . Thus, the wall superheat is well below the superheat for film and transition boiling. Consequently, despite the high temperature regime, no film boiling occurs in Stage 1 and the operation is in the nucleate boiling regime with high heat transfer performance. In contrast to that, the average surface wall temperature in Stage 2 is 517°C and consequently, film boiling occurs resulting in lower heat transfer performance as in Stage 1.

Figure 5.5(b) shows the results for the low temperature operation case. Again, there is a steep temperature drop across the air gap of Stage 1. As the surface wall temperature of the inner tube is below 100°C , only single-phase heat transfer occurs with lower heat transfer coefficients than for boiling. In contrast to that, high heat transfer rates are achieved in Stage 2, as the surface wall temperature is above 100°C , but below the

critical wall superheat for film and transition boiling. The simulation results confirm the working principle of the proposed multi-stage concept.

Analyzing the temperature profiles, the fluid channel wall temperature in Stage 1 is quite uniform in the circumferential direction for both high and low temperature operation. For Stage 2, the fluid channel wall temperature is less uniform in the circumferential direction and noticeably higher on the top side than on the bottom side. In general, despite the small distances in the base plate, there is a noticeable effect of the thermal conductivity of the steel base material on the temperature pattern. Thus, the assumption of a constant fluid wall temperature in the circumferential direction is quite applicable for Stage 1 and less applicable for Stage 2.

5.2 Functional prototype and experimental methods

5.2.1 Experimental setup

Figure 5.6 shows the schematic of the prototype test bench and experimental setup. The storage material is AlSi_{12} , which is housed in a box-shaped graphite container. The container was pressurized with argon and enclosed by a steel container to prevent graphite oxidation at high temperatures. A steel base plate is installed on the bottom. The base plate contains electric heaters for charging heat and fluid channels for discharging heat. The basic TES concept is identical to that presented in Section 4.2 for the bench-scale prototype, with the exception of the heat extraction concept. Instead of a direct air discharge system, a multi-stage boiling discharge system has been installed.

As described in the previous sections, the fluid channels are separated into Stage 1 and Stage 2 channels. The extraction of heat was achieved through a closed-loop boiling and condensing cycle using water as the working fluid, where the extracted heat was controllable and could be shut off. The fluid channels in Stage 1 were located in the inner tubes, which were inserted into the holes in the base plate. The inner tubes were mechanically decoupled from the storage system. A small gap between the base plate and the inner tube creates thermal contact resistance, reducing the potential heat flow to prevent a boiling crisis. Stage 2 involved direct contact between the working fluid and the base plate. In this case, the drilled holes in the base plate served as the fluid channels directly. Heat extraction was realized through forced convection flow boiling in the channels via a fluid pump.

The extracted storage heat was transferred to a separate cooling water circuit via a state-of-the-art heat exchanger. The heat exchanger condensed and sub-cooled the two-phase working fluid. The separate cooling water circuit simulated the vehicle's coolant circuit, and the inlet temperature could be adjusted as needed to an appropriate temperature in the experiments. The fully condensed and sub-cooled water was piped to a reservoir. From the reservoir, the water was drawn using a fluid pump. The reservoir also acts as a pressure buffer and had a pressure relief valve mounted for safety concerns. Figure 5.7 illustrates a simplified fluid and heat flow diagram of the experimental setup.

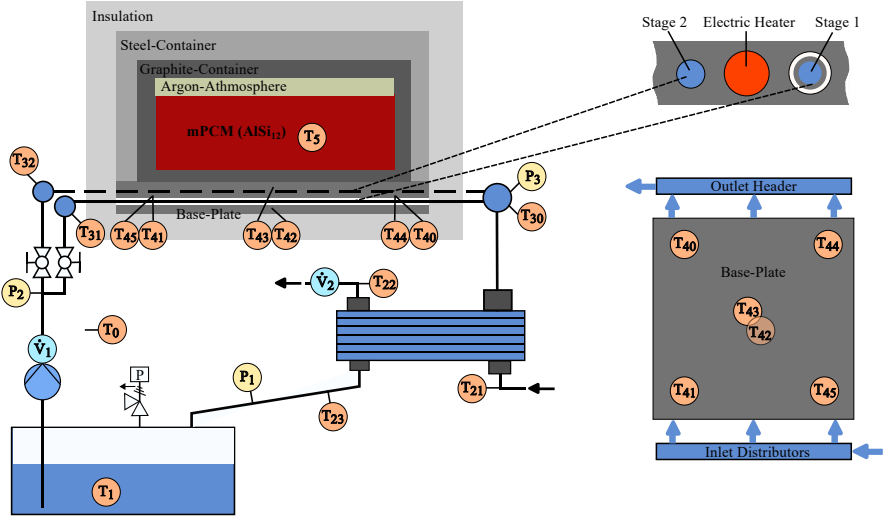


Figure 5.6: Schematic representation of the prototype test bench and experimental setup, reproduced from Nees et al. [80]

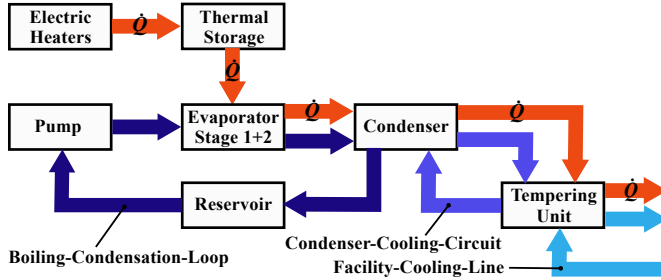


Figure 5.7: Simplified fluid and heat flow diagram of the experimental setup, reproduced from Nees et al. [80]

The test bench was equipped with multiple thermocouples at various locations to measure the temperature of the mPCM, base plate, and fluid circuits. In addition, devices were installed to measure flow and pressure. For temperature measurement, thermocouples of type N with diameters of 1 mm and 1.5 mm from Electronic Sensor GmbH were installed. These thermocouples were calibrated beforehand for a temperature range from ambient to 800 °C using a custom calibration setup with Pt100 and type S thermocouple reference sensors.

For pressure measurement in the boiling-condensing cycle, Wika S-20 absolute pressure gauges with a measuring range of 0–4 bar were installed at the outlet of the pump, in the outlet header, and between the condenser outlet and reservoir inlet. A Ritmo R033 17-7

diaphragm pump (Fink Chem+Tec) was used to supply water from the reservoir to the evaporator stages. Due to its highly accurate displacement, it is able to simultaneously measure the volume flow $\dot{V}_1$ with high accuracy. The pump was calibrated beforehand. A Sika induQ magnetic-inductive flow sensor was installed to measure the flow in the condenser's cooling water line. For condensation, a custom made and state-of-the-art condenser (designed by Terhoeven GmbH) with a minimum rated power of 5 kW was used. The cooling system for the condenser was a Regloplas 90S temperature control unit (Regloplas AG). The Regloplas unit was connected to the domestic cooling water system at the DLR facility.

The argon atmosphere was realized by connection to an argon cylinder and a constant pressure inside the prototype container was maintained during the experiments. Four electric heaters with a diameter of 6.5 mm and an installed individual power of 600 W (Türk + Hillinger) were installed for the charging process. The experimental data were processed using a data acquisition system from Delphin Technology AG. For this purpose, the Expert Logger 400 hardware and ProfiSignal software were employed. The storage system was insulated using the microporous MB1000 insulation from Becker Insulation.

5.2.2 Prototype design and construction

Figures 5.8(a)–(b) show the test bench and the installed prototype setup from different perspectives. Figure 5.8(a) displays the test bench with the installed prototype in the laboratory, including the storage system, condenser, reservoir, inlet distributors, and outlet piping. The components are arranged on different levels. The reservoir is on the lowest level, and the steel container containing the mPCM and graphite encapsulation is on the top level. The condenser and fluid pump are arranged in between. Figure 5.8(b) shows another view of the test bench and prototype setup, displaying the feed pump, valves, electric heaters, and outlet piping. These components are arranged in parallel to each other. In the experiments, the valves were used to manually open or close the feed lines of boiling Stages 1 and/or 2. The basic data of the storage prototype used in the experiments are listed in Table 5.1.

Figures 5.9(a)–(b) show the arrangement of the base plate used for charging and discharging heat. The base plate is made of 1.4980 high-temperature stainless steel alloy. Figure 5.9(a) illustrates the arrangement of the Stage 1 and Stage 2 fluid channels in the base plate and the drill holes for installation of the electric heaters and thermocouples.

The tubes of Stage 1 are mechanically decoupled from the storage system. The Stage 1 fluid channels are located in the inner tubes, which are inserted into the holes in the base plate. The inner tube has an outer diameter of 4 mm and a wall thickness of 0.5 mm. A small gap, averaging 0.15 mm, between the base plate and the inner tube creates thermal contact resistance. This thermal resistance reduces the potential heat flow to prevent a boiling crisis. The gap width was designed to limit the heat flow from the base plate to the Stage 1 fluid channels to approximately 25 W/cm². According to the 1995 look-up

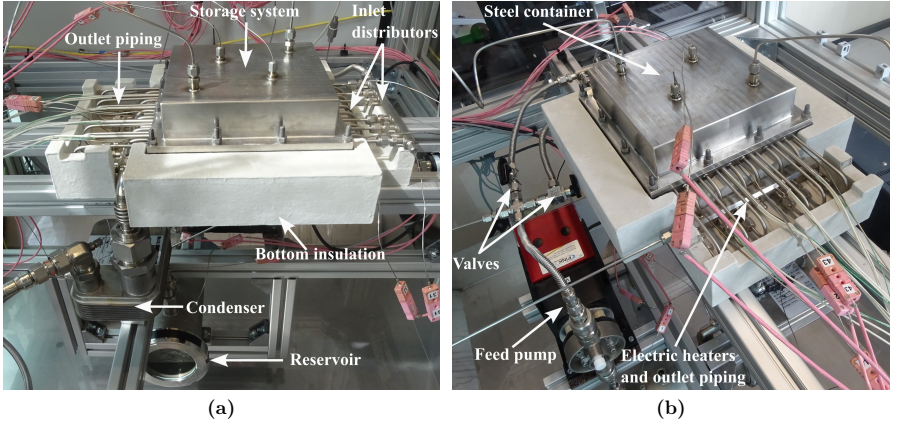


Figure 5.8: Test bench and prototype setup shown from different perspectives, illustrating the storage system and components within the test bench, reproduced from Nees et al. [80]

Item	Specification
Total storage capacity	1.4 kWh (100 – 650 °C)
Total dimension (with insulation)	0.329 x 0.329 x 0.141 m
Total dimension (without insulation)	0.225 x 0.225 x 0.063 m
Weight total	12.78 kg (of which 3.29 kg is insulation)
Base area AlSi ₁₂	0.18 x 0.18 m
Weight AlSi ₁₂	2.45 kg
Electric heaters installed	4 x 600 W
Gravimetric energy density	109.5 Wh/kg (147.5 Wh/kg without insulation)
Volumetric energy density	91.7 Wh/L (439.0 Wh/L without insulation)

Table 5.1: Basic data of the storage prototype used in the experiments (without piping)

table for critical heat flux in tubes [27], this is well below the critical heat flux even for high vapor mass fractions, given the geometry and mass flux in the experiment.

The Stage 2 fluid channels have a hydraulic diameter of 3 mm. This is the exact same diameter as Stage 1. Thus, both stages have six identical fluid channels. The Stage 2 connecting tubes were bent and laser welded to the base plate. Both the inner tubes of Stage 1 and the connecting tubes of Stage 2 are made of stainless-steel type 1.4571. The prototype channel geometries of Stages 1 and 2 are listed in Table 5.2.

Figure 5.9(b) illustrates the arrangement of the base plate, including the inlet distributor stages and the collecting outlet header. The inlet distributors of Stages 1 and 2 are separated. Thus, each boiling stage has its own inlet distributor to allow for operation of a single stage only. The outlet header is a collecting pipe which combines the flows from

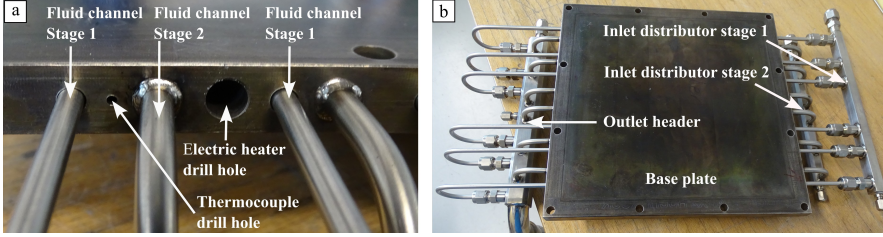


Figure 5.9: Base plate arrangement. (a) Detail view: Fluid channels of Stage 1 and Stage 2 and drill holes for installation of electric heaters and thermocouples. (b) Top view: Base plate with inlet distributor stages and collecting outlet header. Reproduced from Nees et al. [80].

Item	Specification
Type	Circular channel
Channel hydraulic diameter	3 mm (Stages 1 and 2, each)
Number of channels	6 x (Stages 1 and 2, each)
Length (in base plate)	225 mm
Tube Stage 1	4 mm x 0.5 mm
Tube gap (average) Stage 1	0.15 mm

Table 5.2: Prototype channel geometry of Stages 1 and 2

Stages 1 and 2. Swagelok components were laser-welded to the inlet distributors and outlet header to facilitate the assembly of connecting parts. To allow for thermal expansion and ease of installation, a U-bend was provided for the inner tube of Stage 1. In addition, flexible expansion bellows were installed in the inlet and outlet piping connected to the distributors and outlet header.

In Figure 5.10(a), the AlSi_{12} storage material inside the graphite container and the graphite bottom plate is illustrated. In Figure 5.10(b), the graphite bottom plate and the graphite container with the storage material are mounted on the base plate prior to integration into the test bench. The graphite and steel encapsulation concept presented here is identical to the one discussed in Section 4.2 and published by Nees et al. [78, 79] and Stahl et al. [103].

Figures 5.11(a)–(c) show the insulated prototype from different perspectives. The storage system was insulated with a custom-made microporous insulation. High-temperature insulating fiber wool was used to insulate the inlet distributors, outlet header, and piping to the condenser. In addition, the condenser was insulated to prevent heat loss.

Compared to the total weight of the prototype, the weight of the AlSi_{12} storage material, at 2.45 kg, was relatively small. As a consequence, only about half of the total storage capacity relied on the AlSi_{12} material. Thus, a significant portion of the total storage capacity relied on the sensible heat of components such as the base plate, graphite, and

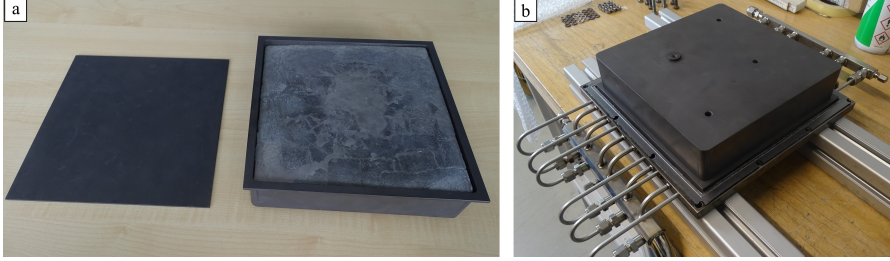


Figure 5.10: Storage material and graphite encapsulation, identical to the one discussed in Section 4.2 and published by Nees et al. [78, 79] and Stahl et al. [103]. (a) mPCM in graphite container with graphite bottom plate. (b) Graphite bottom plate and container with mPCM mounted on base plate. Reproduced from Nees et al. [80].

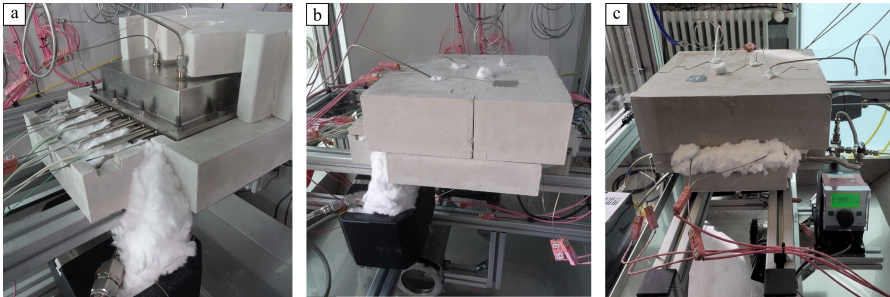


Figure 5.11: Insulated prototype from different perspectives. (a) Partially insulated. (b)–(c) Fully insulated. Reproduced from Nees et al. [80].

steel container. It is important to note that the investigated prototype is small-scale and differs from what would be used in full-scale, real-world applications in the future. This investigation aims to demonstrate the working principle and physical characteristics of the novel heat extraction system in combination with the mPCM thermal storage. Consequently, the dimensions of the condenser, reservoir, pump, and other components were oversized for the purpose of an initial experimental investigation. In full-scale application, the ratio of storage material to other components would be more favorable, enabling higher storage energy densities as the system is scaled-up.

The physical properties of the AlSi_{12} storage material are provided in Table 2.1 and Appendix A. In the applied temperature range of the experiments, more than 50% of the operating thermal energy of the mPCM is above 577 °C, corresponding to the state of charge (SoC). The capability to increase the extraction of heat across the entire operating temperature range for a given heat flow correlates to higher specific storage density. Therefore, an effective thermal discharge is essential at high, medium, and low storage temperature levels.

5.2.3 Experimental procedure

First, the storage system was evacuated and then filled with argon up to an absolute pressure of 1.2 bar to avoid oxidation of the graphite container during the experiments. Thereafter, the storage was charged using the installed electric heaters. The charging process continued until the AlSi_{12} temperature reached $650\text{ }^{\circ}\text{C}$, which is above the phase change temperature of $577\text{ }^{\circ}\text{C}$. This procedure ensured complete melting of the mPCM. To realize this, the base plate was heated to a core temperature of $660\text{ }^{\circ}\text{C}$. The base plate core temperature was calculated as the average of the T_{42} and T_{43} thermocouple readings at the center. The charging process was controlled in order to keep a temperature difference between mPCM and the core base plate of less than 200 K. In contrast, the average base plate temperature was calculated as the arithmetic average from the six thermocouple readings in the base plate T_{40} , T_{41} , T_{42} , T_{43} , T_{44} , and T_{45} . A complete charging process required approximately 1.7 kWh of electrical energy.

The Regloplas condenser cooling system was set to a flow rate ($\dot{V}_2$) of 7.9 l/min water. This flow rate remained constant throughout all conducted experiments. The condenser inlet temperature was maintained at a temperature of $60\text{ }^{\circ}\text{C}$. This temperature was chosen to simulate a vehicle coolant circuit, and it was kept constant throughout the experiments.

After complete charging, the heat input by the electric heaters was stopped. Following this, the discharge process was started in the next step. Before starting the feed pump, the valves in the feed lines of Stage 1 and Stage 2 were manually opened or closed, depending on which stage was being operated. In Stage 1 discharge experiments, the Stage 1 supply line valve was fully open, and the Stage 2 supply line valve was fully closed. In Stage 2 discharge experiments, it was vice versa. Experiments were conducted only for a separate operation of Stages 1 and 2. A coupled operation of Stages 1 and 2 was not tested due to limitations of the experimental setup and because a coupled control mechanism has not yet been implemented. The thermal discharge was achieved by forced convection flow boiling after starting the pump. The pumped volume flow maintained constant throughout an experiment. In real-world vehicle applications, a coupled operation of both stages is intended. Thus, the water flow rate in each stage is controlled to optimize the heat flow as required for cabin heating.

Experiments were carried out by applying three constant flow rates (5, 10, and 15 l/h) for each stage in order to study the heat extraction characteristics under different parameters. The operating temperature range of the storage material was between $650\text{ }^{\circ}\text{C}$ and $100\text{ }^{\circ}\text{C}$ for all discharge experiments. Experiments were performed at least twice for the same set of parameters to verify and demonstrate the reproducibility of the measured data. A schematic procedure of the experiments carried out, as well as an exemplary charging and discharging curve, are shown in Figure 5.12. The mPCM temperature corresponds to the reading of thermocouple T_5 in Figure 5.6. A full charge and discharge cycle took several hours to complete.

In the initial phase, experiments were carried out under atmospheric conditions, with the reservoir being open to the environment. Later on, a safety valve was mounted on the

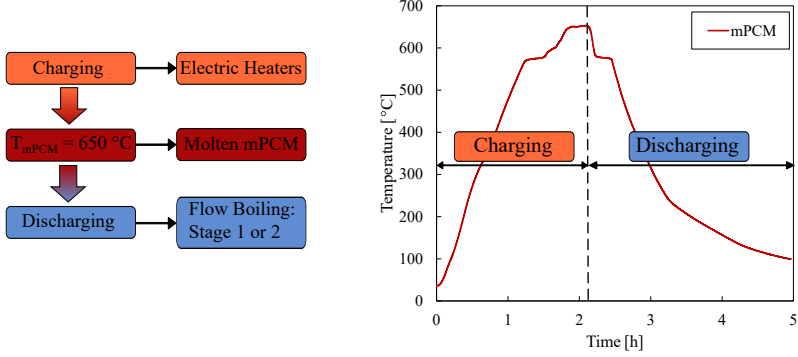


Figure 5.12: Experimental procedure and exemplary charging and discharging curve, reproduced from Nees et al. [80]

reservoir, and experiments were carried out under pressure-tight conditions.

The extracted heat flow $\dot{Q}_{ex}$ was calculated from the measurement data via the energy balance as follows:

$$\dot{Q}_{ex} = \rho_2 \dot{V}_2 c_{p2} (T_{22} - T_{21}) + \rho_1 \dot{V}_1 c_{p1} (T_{23} - T_1) \quad (5.1)$$

The dominant heat flow is transferred to the Regloplas condenser cooling circuit ($\dot{V}_2$) and is determined by the temperature difference between the condenser outlet and inlet ($T_{22} - T_{21}$). The heat transferred to the reservoir via the volume flow $\dot{V}_1$ is a small correction term which considers the temperature difference between the condenser outlet and the water temperature in the reservoir ($T_{23} - T_1$). The water densities ρ_2 and ρ_1 are based on the respective temperatures T_{22} and T_1 . The specific heat capacities c_{p2} and c_{p1} are based on the respective mean temperatures. The heat flow was averaged over 120 s for individual measurements to improve the analysis and visualization of the data.

The heat flux $\dot{q}_{ex}$ in the evaporator stages depends on the channel surface area A_s and is derived as follows:

$$\dot{q}_{ex} = \dot{Q}_{ex} / A_s \quad (5.2)$$

The amount of accumulated extracted energy Q_{ex} is calculated via integration from the time of discharge t using the following equation:

$$Q_{ex} = \int_{t=0}^{t=end} \dot{Q}_{ex} dt \quad (5.3)$$

The mass flux G in the evaporator channels is calculated from the mass flow $\dot{m}_1$ divided by the flow cross section of the channels A_f :

$$G = \rho_1 \dot{V}_1 / A_f = \dot{m}_1 / A_f \quad (5.4)$$

The density ρ_1 is based on the temperature T_1 . An equal distribution of the mass flow across the six parallel fluid channels is assumed. The flow cross section is the same for Stage 1 and Stage 2.

The vapor mass fraction $\dot{x}$ is defined as the ratio of the vapor mass flow $\dot{m}_v$ to the total mass flow $\dot{m}_1$:

$$\dot{x} = \dot{m}_v / \dot{m}_1 \quad (5.5)$$

The vapor mass fraction $\dot{x}$ at the vaporizer outlet is estimated under the assumption of thermodynamic equilibrium and uniform flow distribution across the fluid channels as follows:

$$\dot{x} = \frac{\dot{Q}_{ex} - \dot{Q}_{sub}}{\dot{m}_1 h_v} = \frac{\dot{Q}_{ex} - \dot{m}_1 c_{p1,sub}(T_s - T_1)}{\dot{m}_1 h_v} \quad (5.6)$$

In the equation, $\dot{Q}_{sub}$ is the heat flow required to heat up the subcooled water from T_1 to the saturation temperature T_s and $c_{p1,sub}$ is the corresponding heat capacity. According to interpolation with data from [42], at the DLR laboratory site with an atmospheric pressure of 960 mbar, T_s is 98.4 °C and the specific enthalpy h_v is 2260.7 kJ/kg.

5.2.4 Measurement uncertainty

In order to verify the quality of the measurement results, it is essential to analyze the measurement uncertainty of the equipment used. The relevant measurement points installed are illustrated in Figure 5.6. These include temperature, volume flow, and pressure measurements. The combination and propagation of uncertainties were done according to the JCGM guidelines [41]. The methods applied as well as supplemental measurement uncertainty data are given in Appendix B.

The installed Wika S-20 pressure gauges have a specified accuracy of $\pm 0.5\%$ of the full-scale range. The measured values at ambient pressure were adjusted to the metrological data at the DLR site. Temperature effects were neglectable because the gauges were mounted at a sufficient distance from the hot storage and were connected via non-insulated capillary tubes.

The Sika induQ magnetic-inductive flow sensor was installed to accurately measure the condenser cooling flow of the Regloplas 90S unit. The sensor was calibrated prior to

installation. Considering the accuracy of the calibration and the accuracy given in the manufacturer's data sheet, the combined accuracy of the $\dot{V}_2$ flow measurement is $\pm 2.3\%$. The volume flow of the evaporator feed water was measured directly using the installed Ritmo R033 17-7 diaphragm pump. This pump is capable to measure the flow with high accuracy due to its precise displacement. The pump was calibrated in the laboratory according to the recommendations given in the manufacturer's data sheet. Considering both the accuracy of the calibration and the accuracy of the flow measurement, the combined accuracy of the volume flow $\dot{V}_1$ is $\pm 1.1\%$.

To increase the accuracy of the temperature measurement, the type N thermocouples used were calibrated using a custom calibration setup with PT100 and type S thermocouple reference sensors. Calibration was performed in the final setup on the test bench using a Carbolite Gero laboratory chamber furnace. Thermocouples exposed to the high storage temperatures were calibrated for a temperature range from ambient to 800 °C. This includes all thermocouples in the mPCM, base plate, inlet distributors, and outlet header. The total measurement uncertainty is a result of the combination of the uncertainty of the PT100 and type S reference sensor, the calibration setup, and the regression error. It is less than ± 1.3 K for all thermocouples over the entire temperature range. Thermocouples not exposed to high temperatures were calibrated only for a temperature range from ambient to 200 °C. This includes all thermocouples installed in the condenser Regloplas cooling circuit and in the reservoir. For these, the total measurement uncertainty is less than ± 0.13 K for the given temperature range.

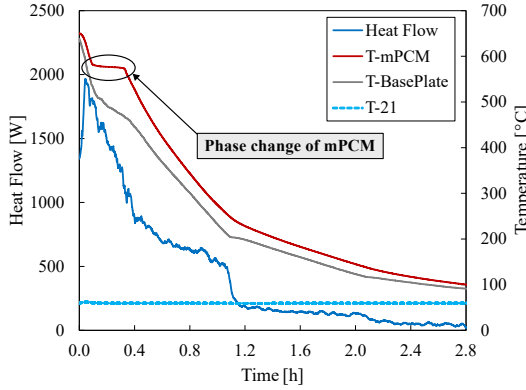
The calculation of the extracted heat flow is based on the measurement of $\dot{V}_2$, T_{22} , T_{21} , $\dot{V}_1$, T_{23} , T_1 , and is therefore exposed to the propagation of uncertainties. The uncertainty in the heat flow results, as discussed in the next section, is in the order of about 100 W, with a maximum uncertainty in the range of 99–131 W across all experiments. This uncertainty is dependent on the specific mass flows and the temperature levels in the experiments. Thus, the uncertainty in the experimental results for the accumulated extracted energy is approximately 0.1 kWh per hour of discharge time.

5.3 Experimental results and discussion

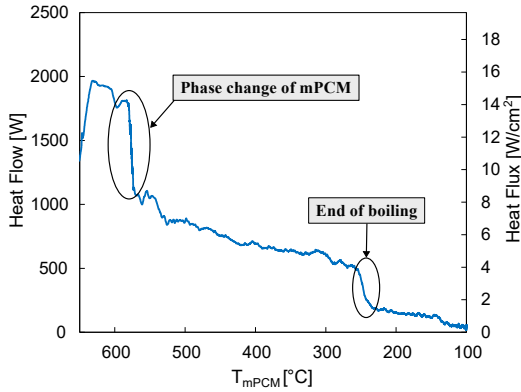
5.3.1 Thermal discharge characteristics of Stage 1

Figures 5.13 and 5.14 show the results for the Stage 1 discharge experiments. To elucidate the physical behavior of the discharge, the experiments were conducted with three different constant flow rates throughout the experiment. Figures 5.13(a)–(b) show the heat flow for Stage 1 at a constant flow rate of 10 l/h and 64 kg/(m²s), respectively, plotted against discharge time and mPCM temperature. The heat flow peaks at approximately 2 kW and decreases over time as both the mPCM and base plate temperatures decrease. The corresponding peak heat flux is 15.5 W/cm².

In Figure 5.13(a), the phase change of the mPCM is indicated by the plateau in the mPCM temperature. In Figure 5.13(b), the phase change of the mPCM is indicated by the



(a) Heat flow and temperatures versus discharge time



(b) Heat flow and heat flux versus mPCM temperature

Figure 5.13: Heat discharge characteristics of Stage 1 (for 10 l/h, 64 kg/(m²s)), reproduced from Nees et al. [80]

significant decrease in heat flow around the phase change temperature of approximately 577 °C. This was due to a large increase in thermal contact resistance at the mPCM-container interface during and after solidification. This phenomenon has already been observed and discussed in detail in Sections 4.2 and 4.3, in the context of investigations of the direct air discharge prototype.

At a mPCM temperature of approximately 250 °C, or a discharge time of about 1.1 h, the heat flow decreases significantly. This effect is related to the end of boiling in the fluid channels. Thus, it is expected that the inner tube wall temperature is not high enough to maintain boiling in the channels. As the temperature drops, only single-phase heat transfer occurs, with lower heat transfer rates than those seen during flow boiling. The condenser inlet temperature showed only small fluctuations throughout the experiments and remained almost constant at a temperature level of 60 °C.

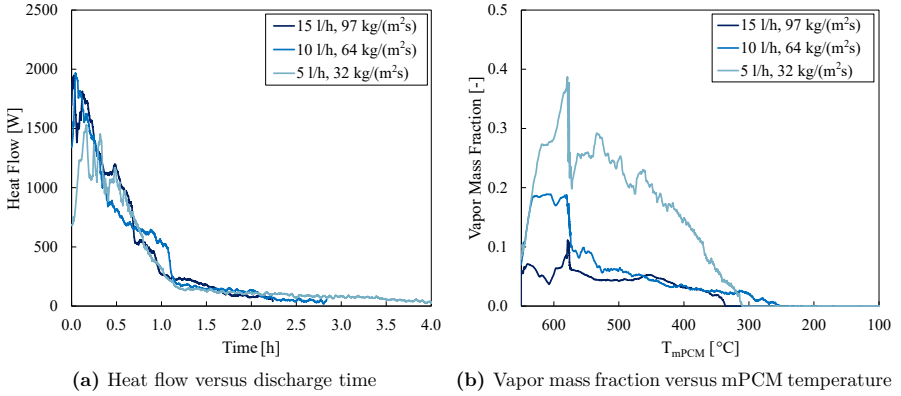


Figure 5.14: Heat discharge characteristics of Stage 1 for various flow rates, reproduced from Nees et al. [80]

Figure 5.14(a) shows the heat flow for various flow rates versus the time of discharge. Experiments with flow rates of 15 l/h (97 kg/(m²s)) and 5 l/h (32 kg/(m²s)) showed similar qualitative behavior to the 10 l/h (64 kg/(m²s)) discharge. However, the discharge time and the peak heat flow differs. Generally, higher flow rates resulted in shorter discharge times.

Figure 5.14(b) shows the corresponding vapor mass fraction for various flow rates at the outlet of Stage 1 versus the mPCM temperature. The vapor mass fraction is more of an estimate, based on the assumptions of thermodynamic equilibrium and uniform flow distribution across the fluid channels. It was calculated using Equation 5.6. The numbers provided give an indication of the order of vaporization, while actual values may be different. A vapor mass fraction of 1 indicates complete vapor, while a vapor mass fraction of 0 indicates complete liquid. The vapor mass fraction was between 0 and 0.39 for all three flow rates in the temperature range studied. Consequently, the outlet flow was two-phase rather than observing complete vaporization.

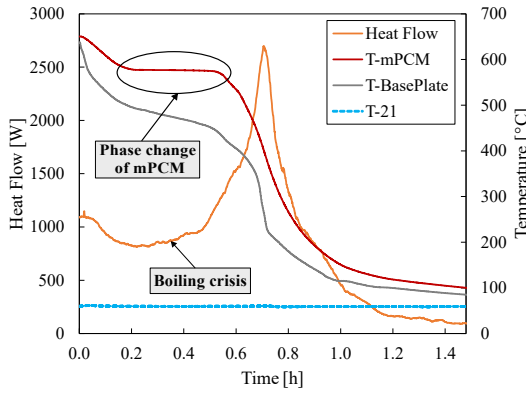
The total amount of extracted heat was 1.23 kWh for 15 l/h and 10 l/h, and 1.20 kWh for the 5 l/h discharge, determined in the 650–100 °C temperature range of the experiments. The difference from the total storage capacity of 1.4 kWh was most likely due to heat losses during the experiment, in addition to measurement uncertainties for the heat flow.

5.3.2 Thermal discharge characteristics of Stage 2

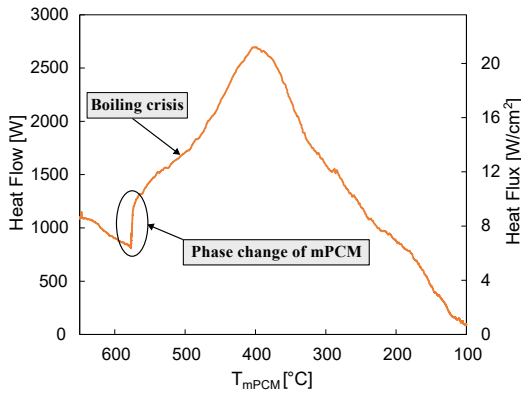
Figures 5.15 and 5.16 show the results for the Stage 2 discharge experiments. As in Stage 1, experiments were conducted with the same three different constant flow rates to allow for a comparison between the stages. Figures 5.15(a)–(b) show the heat flow for Stage 2 at constant flow rate of 10 l/h and 64 kg/(m²s), respectively, plotted against discharge time and mPCM temperature. The heat flow forms the shape of a peak, rising

to approximately 2.7 kW at a mPCM temperature of around 400 °C. The corresponding peak heat flow is approximately 21 W/cm².

At the peak, the heat flow decreases in both higher- and lower-temperature directions, as well as with increasing discharge time. The heat flow reaches a minimum of approximately 0.8 kW around the phase change temperature of 577 °C. The heat flux behavior indicates the occurrence of the boiling crisis. This finding is supported by the relatively low performance at high temperatures and the decrease in heat flow above 400 °C, as visible in Figure 5.15(b). The same is true for discharge times of less than approximately 0.7 hours, as indicated in Figure 5.15(a). The resulting vapor forms an insulating layer on the wall surface because of the significant temperature difference between the hot channel wall and the water's saturation temperature, which deteriorates the heat transfer coefficient.



(a) Heat flow and temperatures versus discharge time



(b) Heat flow and heat flux versus mPCM temperature

Figure 5.15: Heat discharge characteristics of Stage 2 (for 10 l/h, 64 kg/(m²s)), reproduced from Nees et al. [80]

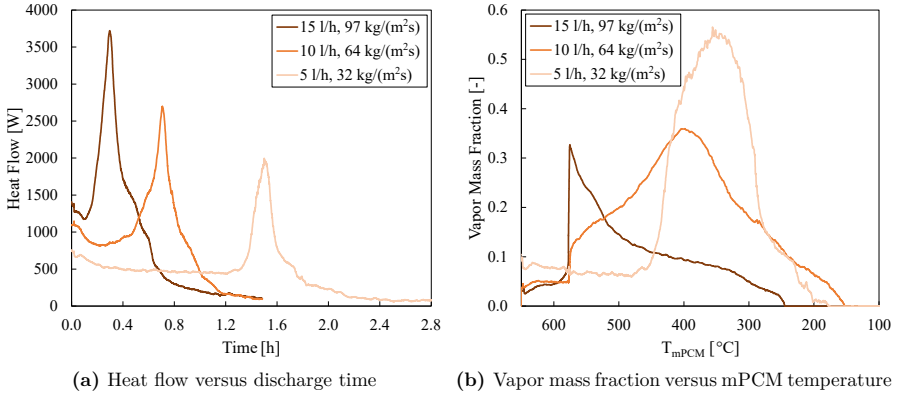


Figure 5.16: Heat discharge characteristics of Stage 2 for various flow rates, reproduced from Nees et al. [80]

In Figure 5.15(a), the phase change of the mPCM is indicated by the plateau of the mPCM temperature. In Figure 5.15(b), the phase change of the mPCM is indicated by the increase in heat flow around the phase change temperature of about 577 °C. This behavior is the opposite of what is shown in Figure 5.13(b). It is caused by increased thermal contact resistance, which reduces possible heat flow to the fluid channels. This mitigates the deteriorating effects of the boiling crisis. As in Stage 1, the condenser inlet temperature showed only small fluctuations throughout the experiments and remained almost constant at a temperature level of 60 °C.

Figure 5.16(a) shows the heat flow for various flow rates versus the time of discharge. Experiments with flow rates of 15 l/h (97 kg/(m²s)) and 5 l/h (32 kg/(m²s)) showed similar qualitative behavior to the 10 l/h (64 kg/(m²s)) discharge. However, the discharge time and the peak heat flow differs. Again, higher flow rates resulted in shorter discharge times.

Figure 5.16(b) shows the corresponding vapor mass fraction for various flow rates at the outlet of Stage 2 versus the mPCM temperature. As in the previous sections, the vapor mass fraction is more of an estimate, based on the assumptions of thermodynamic equilibrium and uniform flow distribution across the fluid channels, and was again calculated by Equation 5.6. The vapor mass fraction was between 0 and 0.57 for all three flow rates. Thus, as observed for Stage 1, the flow was two-phase rather than experiencing complete vaporization.

The total amount of extracted heat was 1.28 kWh for 15 l/h and 5 l/h, and 1.24 kWh for the 10 l/h discharge. These values are slightly higher than those for Stage 1 because the total discharge time is shorter. Shorter discharge times go hand in hand with reduced heat losses. Taking into account heat losses and measurement uncertainties, the extracted

heat for Stages 1 and 2 is reasonable in terms of the overall energy balance, considering the total storage capacity of 1.4 kWh.

5.3.3 Evaluation of thermal discharge performance

For further evaluation of the thermal discharge performance, the resulting heat flow of Stages 1 and 2 were compared for the same flow rate relative to the state of charge of the mPCM storage material (SoC). For the analysis, the measured mPCM temperature was assigned to its corresponding thermal energy and related to the accumulated mPCM energy within the 100–650 °C temperature range. The latent energy of the mPCM was assigned to a temperature range of ± 5 K around the phase change temperature. The results of the heat flow versus the state of charge for Stages 1 and 2 are shown in Figure 5.17. These are based on a constant flow rate of 10 l/h and 64 kg/(m²s), respectively.

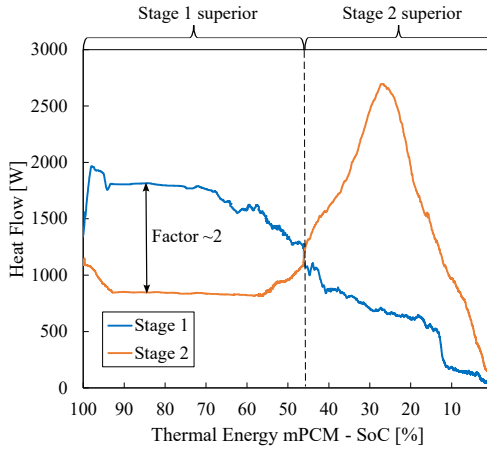


Figure 5.17: Heat flow versus state of charge (SoC) of mPCM storage material for Stages 1 and 2 (for 10 l/h, 64 kg/(m²s)), reproduced from Nees et al. [80]

The experimental results show that Stage 1 provided a higher heat output than Stage 2 for the upper 54% of the stored thermal energy. For the lower 46%, Stage 2 provided a higher heat output than Stage 1. The improvement in the Stage 1 design compared with the Stage 2 design is in the order of a factor of two in the high-temperature regime. In this regime, the latent energy of the mPCM is stored. The intersection of Stages 1 and 2 is at approximately 570 °C mPCM temperature. This corresponds to a base plate temperature of around 450 °C. In Stage 1, the boiling crisis can be largely avoided for the mechanically decoupled fluid channels, according to the qualitative and quantitative results.

In Stage 2, the boiling crisis significantly deteriorated the performance of the direct-contact fluid channels. This was as expected, which was therefore an underlying motivation for developing the novel concept. It can be concluded that Stage 1 was superior

in the high-temperature range, whereas Stage 2 was superior in the medium- and low-temperature ranges. Based on these results, the advantages of the novel two-stage design are demonstrated. Therefore, the general idea of the multi-stage concept introduced in Chapter 3 can be regarded as a proved and constructive novel discharge approach.

Coupling and controlling the two stages allows for high specific heat transfer rates over a wide range of operating temperatures. A coupled operation of Stages 1 and 2 was not tested so far, but is intended for future implementation. Improvements in the design could be achieved by carefully adjusting the width of the gap in Stage 1. By doing so, the peak performance could be increased to the same level as that of Stage 2.

5.3.4 Efficiency of heat extraction

The efficiency of the heat extraction system was evaluated by determining the ratio of the pump's electrical power consumption (P_{el}) to the total extracted heat flow ($\dot{Q}_{ex}$). The pump's electrical power consumption was measured during the experiments. This consumption includes the entire loop from the reservoir to the boiling stages, to the condenser, and back. For a volume flow rate of 10 l/h, the electric pumping power remained relatively constant at approximately 11 W. The time, temperature, or boiling stage used had no significant effect on the electric power required. Figure 5.18 shows the ratio $P_{el}/\dot{Q}_{ex}$ against the mPCM temperature for Stages 1 and 2 at a flow rate of 10 l/h.

As the heat extraction system is sufficiently insulated, the impact of heat losses on the $P_{el}/\dot{Q}_{ex}$ ratio is negligible. In general, high insulation is beneficial for efficiency because it minimizes heat loss.

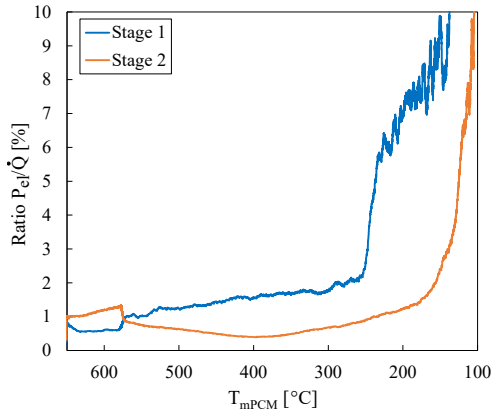


Figure 5.18: Ratio $P_{el}/\dot{Q}_{ex}$ against temperature (for 10 l/h, 64 kg/(m²s)), reproduced from Nees et al. [80]

In Stage 1, the ratio was lowest in the liquid state of the mPCM, reaching a value of 0.5%. This ratio increases with decreasing temperature. At 400 °C, the ratio was 1.5%.

At temperatures lower than 250 °C, the ratio increased significantly. Thus, the heat extraction with Stage 1 becomes more and more inefficient for lower temperatures than that.

In Stage 2, the ratio was slightly higher than 1% in the liquid state of the mPCM. This ratio reaches a minimum of 0.4% at a mPCM temperature of 400 °C. At lower temperatures than 150 °C, the ratio increases sharply. At low temperatures, an efficient discharge is mainly limited due to the high thermal contact resistance at the mPCM-container interface. In summary, it can be concluded that the proposed concept is highly efficient for extracting storage heat and transferring it to a cooling circuit.

5.3.5 Time-dependent temperature pattern

Exposure to high temperatures and rapid thermal transients can cause thermomechanical damage to mechanical components due to high stresses and thermostructural fatigue. Even moderate temperature differences can lead to thermomechanical failure mechanisms. Minimizing both static and transient temperature gradients is the most effective way to reduce these stresses and extend the lifetime of components.

Therefore, the novel concept presented aims to reduce temperature gradients in the storage system. During start-up, the storage system temperature is above 600 °C, while the inlet temperature of the subcooled water is below 100 °C. These are extreme thermal transient conditions. The direct contact between boiling water and the hot storage components is critical, especially at the joints between the Stage 2 fluid channel inlet tubes and the base plate. In contrast, the mechanically decoupled tubes in Stage 1 do not have joints with the base plate. Additionally, the discharge process in Stage 1 significantly reduces temperature gradients in the storage components.

Figures 5.19(a)–(b) show the time-dependent temperature pattern at the measurement points in the base plate and mPCM for Stages 1 and 2. In general, the temperature gradients at a given time are significantly smaller for Stage 1 than for Stage 2 for most of the discharge time.

For visualization purposes, the resulting temperature gradient and corresponding heat flow are highlighted at a discharge time of 0.1 h in Figures 5.19(a) and 5.19(b). This time step was selected because it represents the high-temperature regime within a few minutes after start-up. For this period of time, the maximum temperature difference between the measurement points is less than 120 K for Stage 1's mechanically decoupled tubes. In contrast, it exceeds 330 K for the direct contact fluid channels in Stage 2. Meanwhile, the heat flow in Stage 1 is two times greater than in Stage 2. This indicates that Stage 1 operation is capable of reducing effectively temperature gradients in the storage system compared to direct contact heat extraction in Stage 2. Consequently, operating Stage 1 at high temperatures is beneficial not only in terms of performance but also in reducing both static and transient temperature gradients. This feature can extend the lifespan of the storage system.

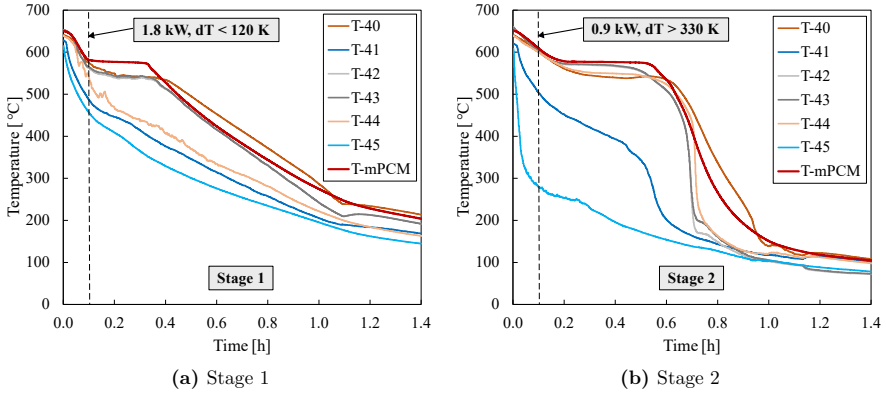


Figure 5.19: Time-dependent temperature pattern in base plate and mPCM for Stages 1 and 2. For comparison, the resulting temperature gradient is highlighted at 0.1 h discharge time with the corresponding heat flow. Reproduced from Nees et al. [80].

5.3.6 Closed-loop system pressure characteristics

Aside from performance, the pressure to be reached in a closed-loop, self-contained system, is another critical aspect for real applications. To address this issue, additional experiments were conducted under pressure-tight conditions. Before the experiments started, a pressure leakage test was performed. This leakage test revealed a pressure loss of 40 mbar per hour at an absolute pressure of 1.8 bar, which was acceptable for the experiments in terms of the pressure tightness of the boiling-condensation-discharge loop.

Figures 5.20(a)–(b) show the resulting pressures and heat flow for Stages 1 and 2 against the discharge time at a flow rate of 10 l/h. The maximum absolute pressure in the outlet header (P3) reached approximately 1340 mbar for Stage 2 and 1320 mbar for Stage 1. The absolute pressure remained below 1.4 bar at all measuring points in the system. This absolute pressure is low enough to be considered uncritical for a future technical application. The pressure (P2) measured at the pump outlet showed fluctuating behavior. This was not the case for the pressures at P1 and P3.

The amount of heat flow is directly related to the pressure level, which peaks during periods of intense vaporization. The peak heat flow values for both stages in the self-contained system were equivalent to those at atmospheric pressure. Nevertheless, compared to experiments conducted at atmospheric pressure, the water saturation temperature shifted slightly towards higher values. At an absolute pressure of 1330 mbar, the saturation temperature (T_s) is 107.5 °C, and the specific enthalpy (h_v) is 2236.3 kJ/kg, based on interpolation with data from [42]. Thus, there is an increase by 9.1 K in (T_s) compared to atmospheric conditions at 960 mbar.

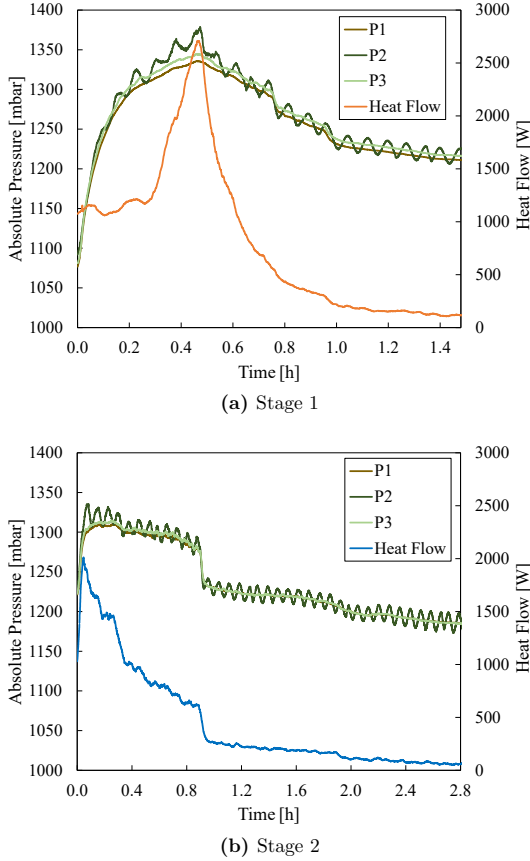


Figure 5.20: Pressure and heat flow versus discharge time in pressure-tight condition (10 l/h, 64 kg/(m²s)), reproduced from Nees et al. [80]

5.4 Derivation of heat transfer calculation model

Based on the experimental results, a heat transfer calculation model for the multi-stage concept is derived in this section. With the help of the model, the goal is to roughly estimate the heat flow of the discharge system under various boundary conditions and to further investigate the boiling heat transfer characteristics. Due to the complex physical effects during the thermal discharge (various boiling regimes of the working fluid, phase change of the storage material, complex and transient temperature field, etc.) it is a challenge to model the resulting heat flow in an accurate way. The heat transfer processes in the working fluid are more complex than for the developed direct air discharge concept presented.

In the first step, the development of a physical heat transfer model is presented based on

an analytical approach and correlation-based equations for heat transfer coefficients. In the second step, the physical model is adapted by adjusting certain model parameters in order to better fit the experimental results obtained in the previous section. The model is focused on the multi-stage boiling discharge in the evaporator stages. The condensation of the working fluid in the condenser is therefore not part of the model.

5.4.1 Development of a physical heat transfer model

In this section, the developed physical model is presented. The heat flow is estimated for Stage 1 and Stage 2 in dependence on the fluid channel wall temperature and the fluid mass flow. The model includes several assumptions and simplifications. A constant wall temperature at the fluid channel interface (T_w , $T_{w,S1}$, $T_{w,S2}$, $T_{w,tube}$) is assumed in both circumferential and flow directions. The model is considered to be of 0D/1D in nature, rather than providing three-dimensional resolution of the real geometry. The model considers a steady-state heat transfer process from the base plate channel wall to the working fluid in Stages 1 and 2.

These assumptions and simplifications are expected to allow for an accurate prediction of the quantitative and qualitative heat flow. At the same time, they reduce the computational requirements tremendously. A schematic representation of the heat transfer from the base plate to the bulk fluid with corresponding temperatures is illustrated in Figure 5.21.

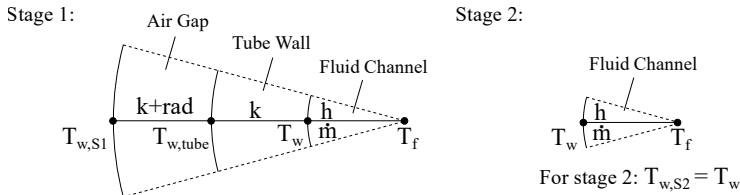


Figure 5.21: Schematic representation of heat transfer from base plate to fluid bulk with corresponding temperatures

The modeled geometry is identical to that of the prototype presented in the previous sections of this chapter. Water is considered as the working fluid with the corresponding mass flow rate, mass flux, and pressure as in the experiment. Temperature dependent fluid and material properties are defined in the model. The air gap of Stage 1 is considered by conduction and radiation heat transfer. The tube wall of Stage 1 is considered by conduction only. Heat transfer in the channels of Stages 1 and 2 is modeled by convection. The heat transfer coefficients are calculated on a correlation basis.

A schematic representation of the modeling approach is shown in Figure 5.22. The model works with an iterative approach. The iteration continues until the deviation $\Delta\dot{Q}$ of the energy balances is close to zero ($\dot{Q}_{k+rad} = \dot{Q}_h = \dot{Q}_m$ for Stage 1 and $\dot{Q}_h = \dot{Q}_m$

for Stage 2). The model is initialized with input parameters, such as wall temperatures ($T_{w,S1}$ and $T_{w,S2}$), fluid inlet temperature ($T_{f,in}$), mass flow rate ($\dot{m}$), fluid pressure (p), and fluid and material properties. The resulting fluid temperature T_f , vapor mass fraction $\dot{x}$ (Stages 1 and 2), and fluid channel wall temperature (only Stage 1) are used as input to recalculate the system of equations.

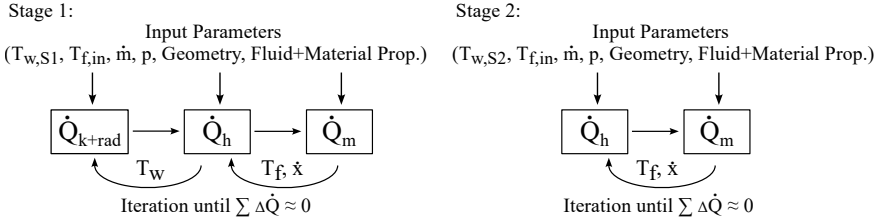


Figure 5.22: Schematic representation of the modeling approach

The energy balance by conductive and radiative heat transfer $\dot{Q}_{\lambda+rad}$ for Stage 1 across the air gap and the inner tube wall is as follows:

$$\begin{aligned} \dot{Q}_{k+rad} &= \frac{A_{w,S1}}{\frac{d_{w,S1}}{2k_{air}} \ln\left(\frac{d_{w,S1}}{d_{tube}}\right)} dT_1 + \frac{A_{w,S1}}{\frac{1}{\varepsilon_{w,S1}} + \frac{A_{w,S1}}{A_{w,tube}} \left(\frac{1}{\varepsilon_{w,tube}} - 1\right)} \sigma (T_{w,S1}^4 - T_{w,tube}^4) \\ &= \frac{A_{w,tube}}{\frac{d_{tube}}{2k_{tube}} \ln\left(\frac{d_{tube}}{d_w}\right)} dT_2 \end{aligned} \quad (5.7)$$

$$\text{with } dT_1 = T_{w,S1} - T_{w,tube} \quad \text{and} \quad dT_2 = T_{w,tube} - T_w$$

The temperatures $T_{w,S1}$, $T_{w,tube}$, and T_w are at the wall of the base plate drill hole, the outer wall of the inner tube, and the wall of the fluid channel. The areas $A_{w,S1}$ and $A_{w,tube}$ are the surface areas of the base plate drill hole wall of Stage 1 and the outer wall of the inner tube. The corresponding diameters are $d_{w,S1}$, which is the Stage 1 drill hole diameter in the base plate, and d_{tube} , which is the outer tube diameter of the inner tube. The corresponding emission coefficients are $\varepsilon_{w,S1}$ and $\varepsilon_{w,tube}$. The emission coefficients are assumed to be $\varepsilon_{w,S1} = \varepsilon_{w,tube} = 0.5$. The heat transferred by radiation across the air gap is less than 10% of the heat transferred by conduction and is therefore less significant for the total heat flow. The left and right sides of the equation are solved by an iterative approach using the method of least squares by adjusting the parameters $T_{w,tube}$ and T_w .

The energy balance by means of convective heat transfer $\dot{Q}_h$ in the fluid channel for the Stages 1 and 2 is calculated in the following way:

$$\dot{Q}_h = hA_s dT_3 \quad (5.8)$$

with $dT_3 = T_w - T_f$

The heat transfer coefficient is h and the surface area of the channels is A_s .

The energy balance by mass transport $\dot{Q}_m$ for single-phase and two-phase heat transfer is modeled as follows:

$$\dot{Q}_m = \dot{m}_1 c_{p,l} dT_4 + \dot{m}_1 \dot{x} h_v + \dot{m}_1 c_{p,v} dT_5 \quad (\text{for } 0 \leq \dot{x} \leq 1) \quad (5.9)$$

with $dT_4 = T_{f,l} - T_{f,in}$ and $dT_5 = T_{f,v} - T_s$

The corresponding vapor mass fraction ($\dot{x} = \dot{m}_v / \dot{m}_1$) is calculated from Equation 5.6, where $\dot{m}_1$ is the total mass flow.

The modeling of the heat transfer coefficient (h) used in Equation 5.8 depends on selecting the applicable heat transfer regime in the first step. Figure 5.23 shows the model criteria for this selection process. The heat transfer regime is selected based on the combination of wall temperature (T_w), saturation temperature (T_s), fluid temperature (T_f), critical temperature (T_{crit}), minimum temperature (T_{min}), and vapor mass fraction ($\dot{x}$). It is essential to select the appropriate heat transfer regime, as the heat transfer coefficient differs considerably between these different regimes. The physical effects that occur are also fundamentally different.

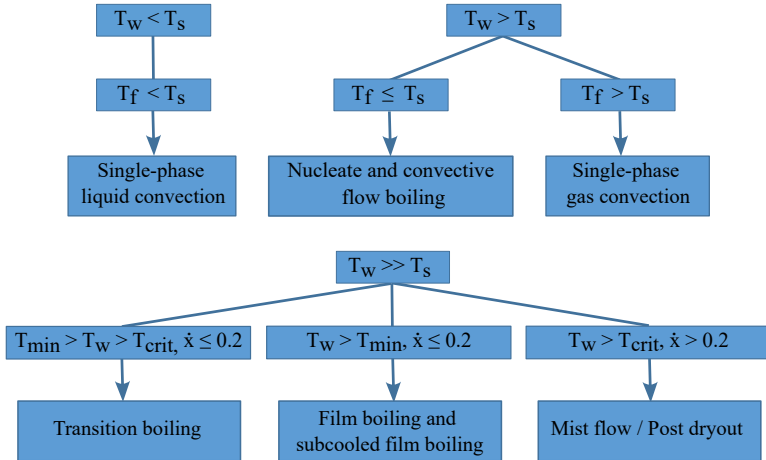


Figure 5.23: Model criteria for selection of heat transfer regime

After selecting the applicable heat transfer regime, in a second step each regime is dedicated to a specific calculation method for the heat transfer coefficient. Therefore, appropriate correlation-based models from the literature are used. The calculation methods used for the corresponding heat transfer regime, together with further assumptions made, are described in Table 5.3.

Heat transfer regime	Calculation method
Single-phase liquid convection	$Nu = 3.66$, according to Table 2.3, as laminar flow for $Re < 2300$, circular channel geometry
Single-phase gas convection	$Nu = 0.023 Re^{0.8} Nu^{0.4}$, according to Equation 2.18, as turbulent flow for $Re > 2300$, circular channel geometry
Nucleate and convective flow boiling	According to method after Kim/Mudawar [49], see Subsection 2.5.2. Subcooled part treated as saturated by assumption.
Transition boiling	Interpolation of $\dot{q}$ with logarithmic function between $\dot{q}_{crit}$ and $\dot{q}_{min}$ for given temperature between T_{crit} and T_{min} , according to Equation 2.47
Film boiling and subcooled film boiling	According to equations given in Subsection 2.5.4. Assumption for combined heat transfer coefficient h for saturated and subcooled film boiling: $h = 0.75 \alpha_{film} + 0.25 \alpha_{sub}$, with an average subcooling of 20 K
Mist flow, post dryout vaporization	Not applicable for most cases, as $\dot{x} < 0.2$ is expected for wall-dry out. The heat transfer coefficient is assumed to be the same as for film boiling.

Table 5.3: Dedication of heat transfer regimes to specific calculation methods for the heat transfer coefficient h

In order to model the boiling curve, the temperatures and heat fluxes at the critical and minimum heat flux points have to be known. The definitions of the corresponding model parameters, dT_{crit} and $\dot{q}_{crit}$, and dT_{min} and $\dot{q}_{min}$, are summarized in Table 5.4. These are based on literature data for water at atmospheric pressure.

Physical item	Calculation method
Critical heat flux $\dot{q}_{crit}$	Linear interpolation of $\dot{q}_{crit}$ for given mass flux and pressure with the 1995 look-up table for critical heat flux in tubes [27] and Equation 2.43 for correction of the tube diameter.
Excess temperature dT_{crit} for critical heat flux	According to Table 2.4, see also Section 2.5
Minimum heat flux $\dot{q}_{min}$ (Leidenfrost point)	$\dot{q}_{min} = h dT_{min}$; Combined heat transfer coefficient h for saturated and subcooled film boiling at Leidenfrost point multiplied by dT_{min}
Excess temperature dT_{min} for minimum heat flux	According to Table 2.4, see also Section 2.5

Table 5.4: Definition of model parameters dT_{crit} and $\dot{q}_{crit}$, and dT_{min} and $\dot{q}_{min}$

Figure 5.24 illustrates the modeled parameters and heat transfer regimes in the modeled boiling curve.

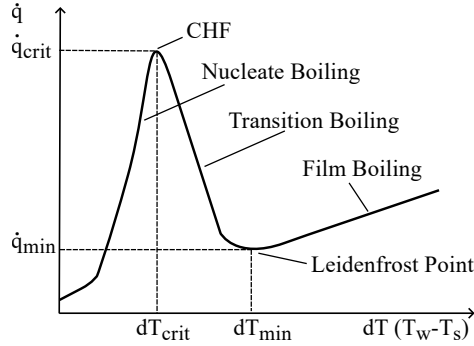


Figure 5.24: Definition of model parameters (modeled boiling curve)

The results of the model are presented in the next section, along with a comparison with the experimental data and the adjustment of the model.

5.4.2 Adjustment of the model and comparison with experimental data

Based on the developed physical model and with the help of the experimental data, it is now possible to include this information into the modeling process. For better fit to the experimental data, an adjusted model is derived to further characterize the heat transfer process. Therefore, each stage is discussed individually in the following.

As the physical model is based on the wall temperatures $T_{w,S1}$ and $T_{w,S2}$, which were not directly measured in the experiment, the measured temperature of the base plate has to be adapted in order to provide a correct comparison. This is due to the temperature drop in the base plate towards the evaporator channels, which is caused by the solid material's limited thermal conductivity. The relation of the base plate temperature T_{BP} , averaged across the measured temperature at six different positions, and the wall temperatures $T_{w,S1}$ and $T_{w,S2}$ is estimated by:

$$T_{BP} - T_{w,S1/S2} = \frac{\dot{Q}_{ex}(d_{x,BP}/2k_{BP})\ln(d_{x,BP}/d_{w,S1/S2})}{A_{x,BP}} \quad (5.10)$$

Here, $d_{x,BP}$ is the thickness of the base plate of 12 mm and $d_{w,S1/S2}$ is the diameter of the drill hole of Stage 1, respectively Stage 2. $A_{x,BP}$ is the imaginary cylindrical surface of a cylinder of the diameter $d_{x,BP}$ times the number of drill holes per stage, which is six for this prototype. For an example, applying this leads to a temperature drop of

6.3 K for Stage 1 and 8.4 K for Stage 2 for a heat extraction performance of 1000 W per stage with the geometry of the prototype at a temperature of 400 °C. Therefore, the experimentally measured base plate temperature is corrected correspondingly for further comparison. The procedure is a best guess according to the Fourier law and might deviate slightly from the real temperature drop in the experiment.

5.4.2.1 Stage 1 model

Figures 5.25(a)–(d) show a comparison of the heat flow results for the adapted model and the unadapted model with the experimental data for Stage 1 for different flow rates. The heat flow is plotted against the $T_{w,S1}$ temperature and the experimental temperature data have been corrected according to Equation 5.10 to allow a comparison.

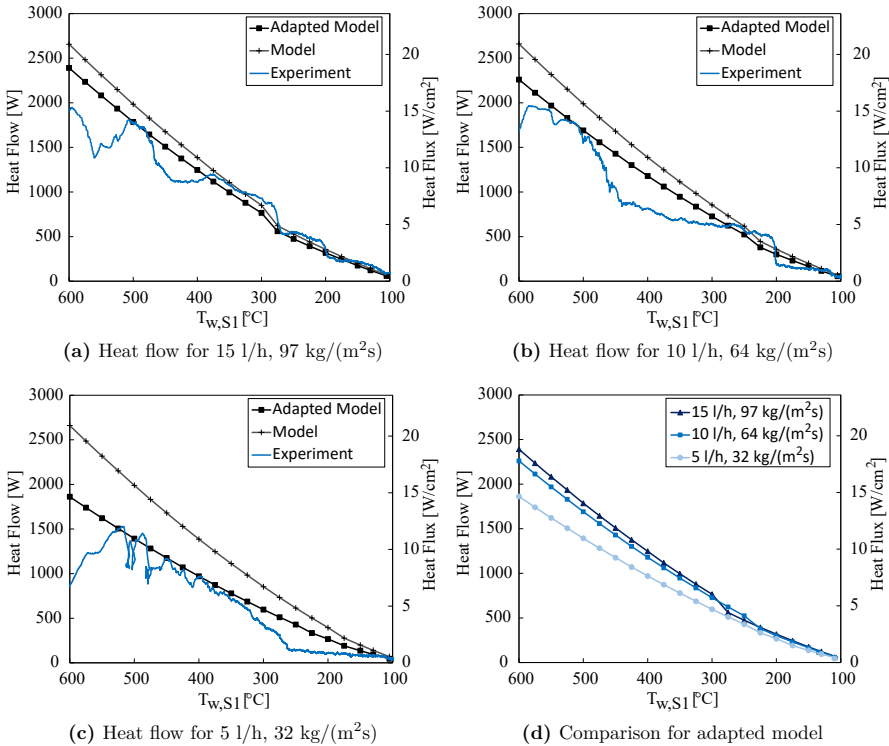


Figure 5.25: Comparison of heat flow results of adapted and unadapted model with experimental data for Stage 1

For better fit to the experimental data, the adapted model considers adjustment factors as presented in Table 5.5. Multiplying these factors by the results of the developed physical model yields the results of the adapted model.

Flow rate	15 l/h, 97 kg/(m ² s)	10 l/h, 64 kg/(m ² s)	5 l/h, 32 kg/(m ² s)
Adjustment factor	0.9	0.85	0.7

Table 5.5: Adjustment factors for the adapted model of Stage 1

The physical model is able to predict roughly the magnitude and the qualitative curve of the heat flow in dependence of the temperature. The adapted model is able to improve the fit to the experimental data. However, the models are not able to predict the fluctuations of the experimental data curve. The deviations between the models are dependent on the mass flow rates. The deviation is smaller for higher flow rates and larger for lower flow rates. Obviously, the impact of mass flow on the heat flow is higher than predicted by the used correlations. Figure 5.25(d) shows a comparison of the heat flow results of the adapted model for different flow rates.

The possible main reasons for the deviation between the developed models and the experimental data for Stage 1 are related to the following effects:

- Slight deviation of the air gap caused by manufacturing tolerances: 0.01 mm larger or smaller gap width results in approximately 5% more or less total heat flow
- Occurrence of boiling crises (partly) during start-up for a short period of time until the inner tube wall has cooled down to temperatures below T_{crit} (for deviations at temperatures $T_{w,S1} > 550$ °C)
- The measurement uncertainty of experimental results is about ± 131 W
- Possible maldistribution of fluid flow across channels
- Higher influence of phase separation in tubes due to gravity than considered in correlations
- Uncertainty of applied correlations from literature

The relatively large discrepancy between the model results and the experiment at $T_{w,S1}$ temperatures above 550 °C is likely due to the partial occurrence of the boiling crisis. At start-up, the inner tubes reach temperatures similar to those of the base plate (up to more than 600 °C), and the water entering the tubes is exposed to these severe conditions. During this initial phase, the inner tubes require a short amount of time to cool down and reach $T_w < T_{crit}$. Although the concept generally prevents the boiling crisis, this transient start-up effect is difficult to avoid.

Nonetheless, this effect is expected to be less noticeable with larger storage sizes because the thermal mass of the inner tubes becomes smaller in relation to the thermal mass of the storage system. Additionally, reducing the thermal mass of the inner tubes, e.g., by using a thinner tube wall, is one option to reduce this negative effect and improve performance directly after start-up. This results in faster cooling of the inner tube after startup and an earlier prevention of the boiling crisis.

5.4.2.2 Stage 2 model

Figures 5.26(a)–(d) show a comparison of the heat flow results for the adapted model and the unadapted model with the experimental data for Stage 2 for different flow rates. The heat flow is plotted against the $T_{w,S2}$ temperature and the experimental temperature data have been corrected according to Equation 5.10 to allow a comparison.

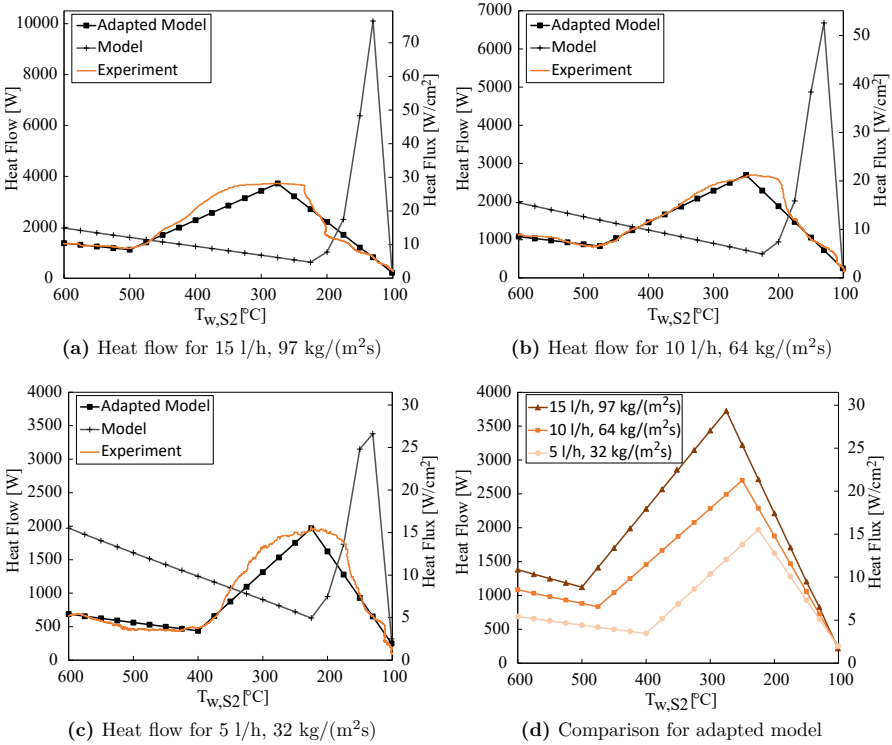


Figure 5.26: Comparison of heat flow results of adapted and unadapted model with experimental data for Stage 2

To better fit the experimental data, the adapted model takes into account adjustment factors as shown in Table 5.6. Adapting the developed physical model with these parameters yields the results of the adapted model. In addition to the adjustment factors, the adapted model considers a linear curve for the heat flow in the transition regime between T_{crit} and T_{min} . This is in contrast to a logarithmic relationship for the transition boiling regime in the literature and in the physical model. Furthermore, a linear curve is considered for the heat flow in the nucleate and convective boiling regime for $T_{w,S2} < T_{crit}$.

The developed physical model deviates clearly from the experimental data for Stage 2. Basically, the peak heat flux is shifted to lower values and the temperatures T_{crit} and

Flow rate	15 l/h, 97 kg/(m ² s)	10 l/h, 64 kg/(m ² s)	5 l/h, 32 kg/(m ² s)
$\dot{q}_{crit}$	29.3 W/cm ²	21.2 W/cm ²	15.5 W/cm ²
dT_{crit}	175 K	150 K	125 K
dT_{min}	400 K	375 K	300 K
Adjustment factor film boiling HTC	0.7	0.55	0.35

Table 5.6: Parameters for $\dot{q}_{crit}$, dT_{crit} , dT_{min} , and film boiling HTC factor for the adapted model of Stage 2

T_{min} are shifted to higher temperatures. Also, the film boiling heat transfer coefficient is overestimated in the model. Therefore, these parameters have been adjusted according to Table 5.6 for the adapted model. The adapted model is able to significantly improve the fit to the experimental data. However, despite the improvements done to enhance the model, a precise fit to the experimental data curve is not possible. Consequently, the adapted model can be considered more as a rough prediction of the temperature dependent heat flow. Obviously, there is a noticeable effect of the flow rate on the film boiling heat transfer compared to the correlations used in the literature and in the physical model. 5.26(d) shows a comparison of the heat flow results of the adapted model for different flow rates.

The possible main reasons for the deviation between the developed models and the experimental data for Stage 2 are related to the following effects:

- The assumption of equal channel wall temperature in circumferential and flow direction is too optimistic. Large temperature differences in the flow direction were observed in the experiments: This can lead to the occurrence of several boiling regimes at different locations at the same time, which is considered as the main reason for the shift of T_{crit} and T_{min} . Non-uniform temperature in the circumferential direction: High heat fluxes close to $\dot{q}_{crit}$ are not possible over the entire channel surface area due to the limited thermal conductivity of the base plate, which is considered as the main reason for a lower peak heat flow compared to the model.
- Possible maldistribution of fluid flow across channels
- Higher influence of phase separation in tubes due to gravity than considered in correlations
- The measurement uncertainty of experimental results is about ± 131 W
- Uncertainty of applied correlations from literature
- Film boiling correlation from literature of limited suitability for forced convection channel flow under the boundaries presented

5.4.3 Coupled operation of Stages 1 and 2

In real applications, the combined operation of Stage 1 and Stage 2 is important. This is particularly relevant for the controlled operation of a constant total heat flow over a wide operating temperature range. However, in the experiments, only single-stage operation with constant mass flow rates was performed. In coupled mode, the heat flows of Stages 1 and 2 are added to create a combined total heat flow. This is done under the assumption of negligible temperature differences between $T_{w,S1}$ and $T_{w,S2}$ for different heat flows. In real application, slight temperature differences will occur, but they will have a neglectable impact on the total heat flow.

Figure 5.27(a) shows the heat flow results for the coupled operation versus the temperature ($T_{w,S12}$) for a constant mass flow of 15 l/h (97 kg/(m²s)). The results are based on the adapted model presented in the previous subsections. The coupled heat flow of Stages 1 and 2 reaches a maximum of approximately 4.3 kW at 275 °C. In the high-temperature region, Stage 1 contributes the major part to the total heat flow, showing peak values of around 2.4 kW. In contrast, Stage 2 contributes the major part to the total heat flow in the mid and low temperature region, reaching peak values of around 3.7 kW.

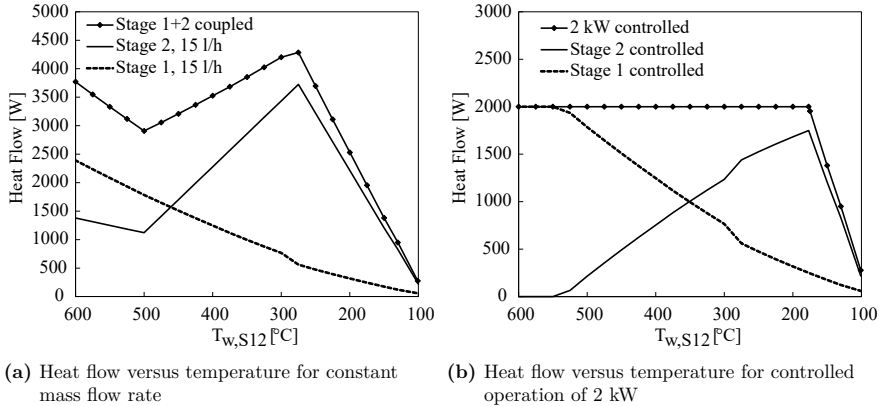


Figure 5.27: Results for coupled operation of Stages 1 and 2, based on the adapted model (15 l/h corresponds to 97 kg/(m²s))

Figure 5.27(b) shows the heat flow results versus temperature ($T_{w,S12}$) for controlled operation of 2 kW. Again, the results are based on the presented adapted model. A heat output of 2 kW can be achieved within the full operating temperature range, with a lower limit of 177 °C. For temperatures lower than that, the heat flow falls below 2 kW, even in coupled operation with full mass flow in each stage. In the high-temperature region and at start-up, only operation of Stage 1 is required. At a temperature of approximately 550 °C, Stage 2 is activated, and the flow rate gradually increases as the temperature decreases. The results show that the coupled and controlled operation provides a high specific heat

output over a wide operating temperature range. In addition, the thermomechanical load is significantly reduced because the required mass and heat flow of Stage 2 is minimized.

5.4.4 Upscaling of results to application scale

This subsection suggests and describes a practical scaling method for designing future concepts and prototypes. Furthermore, it derives an upscaling of the presented prototype results to application scale. These results are used in Chapter 6 for comparison of the developed thermal discharge concepts. The experimental and heat transfer calculation model results presented so far are based on a single prototype geometry. Therefore, an appropriate method is needed to estimate the heat flow for different storage dimensions.

For upscaling or downscaling of future prototypes, a dimensional analysis is suggested, which is based on the gained experimental data and modeling results. In this context, the ratio of a variable y is defined as the ratio of the y_{Scale_2} values to the y_{Scale_1} values of the variable y for the up- and/or downscaled size:

$$Ratio\ y = y_{Scale_2} / y_{Scale_1} \quad (5.11)$$

For engineering estimation purposes, up- and/or downscaling are considered valid when the following criteria are fulfilled:

1. Ratio A_s / Ratio A_f = 1.0 (Ratio of the total channel surface area A_s divided by the ratio of the total fluid cross flow area A_f)
2. Ratio G = 1.0 (Ratio of the mass flux G ($G = \dot{m}/A_f$))
3. Maintaining basic design parameters, such as fluid and material properties, the Stage 1 gap width, and the general arrangement of components

In other words, the mass flux and vapor mass fraction must be kept within the range of the conducted experiments simultaneously. This is essential for meaningful model outcomes. Following this procedure yields an estimate rather than an exact prediction of the actual physical numbers expected for thermal discharge characteristics.

For practical application and comparison of the developed thermal discharge concepts in the next chapter, the multi-stage results are scaled up to application scale size. Therefore, the base plate dimensions of the direct air discharge prototype at the application scale, described in Section 4.4, are considered.

Table 5.7 summarizes the selected, upscaled channel geometry. The design consists of an inside channel diameter of 6 mm for Stages 1 and 2 instead of 3 mm as in the experimental prototype. The gap width and tube wall thickness of Stage 1 are identical to those of the experimental prototype presented in Section 5.2. The number of channels per stage is increased to 15 for both stages. The base plate length considered is 450 mm, and the width

is 555 mm, which is identical to the air discharge prototype. With the selected number of channels and diameters, all fluid channels and electric heaters can be arranged in parallel in the base plate. Consequently, the storage system's volume is reduced compared to the direct air discharge concept because the air heat transfer section is not necessary.

Item	Specification
Channel hydraulic diameter	6 mm (Stages 1 and 2, each)
Number of channels	15 x (Stages 1 and 2, each)
Length (in base plate)	450 mm
Tube Stage 1	7 mm x 0.5 mm with 0.15 mm air gap

Table 5.7: Selected upscaled channel geometry, considering base plate dimensions identical to that of the direct air discharge application-scale prototype

Table 5.8 lists the ratios for upscaling of the experimental prototype results to the application scale, based on the selected channel geometry. The channel geometry was carefully selected to fulfill the criteria of the described scaling method. Therefore, the ratio of the total channel surface area (A_s) to the total flow cross section area (A_f) is 1.0. By maintaining an identical mass flux to that of the experiments, similar physical conditions for heat transfer are expected in terms of velocities, flow regimes, heat transfer regimes, and vapor mass fractions. This results in a factor of 10 for both the mass flow ($\dot{m}$) and the heat output ($\dot{Q}_{ex}$). The upscaled heat flow is calculated by multiplying this factor by the results of the adapted model detailed in the previous sections.

Item	Ratio/Factor
Ratio A_s	10.0
Ratio A_f	10.0
Ratio A_s / Ratio A_f	1.0
Ratio G	1.0
Ratio $\dot{m}$	10.0
Ratio $\dot{Q}_{ex}$	10.0

Table 5.8: Ratios for upscaling of experimental prototype results to the application scale for the multi-stage boiling discharge concept

Figures 5.28(a)–(b) show the heat flow results for the upscaling to application scale. These are based on the upscaled adapted model results, taking into account the geometry and ratios given in Tables 5.7 and 5.8. The results consider a coupled operation of Stages 1 and 2.

Figure 5.28(a) shows the heat flow results versus the temperature ($T_{w,S12}$) for a constant mass flow rate of 15 l/h (97 kg/(m²s)). The coupled heat flow of Stages 1 and 2 reaches a maximum of approximately 43 kW at 275 °C. In the high-temperature region, Stage 1

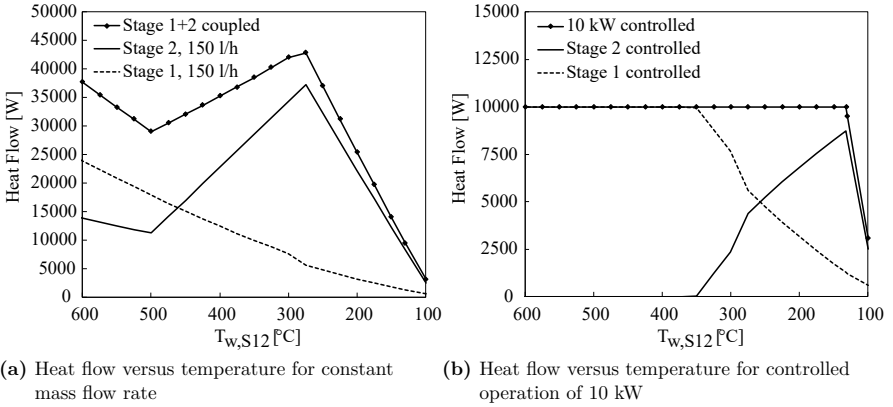


Figure 5.28: Results for coupled operation of Stages 1 and 2, based on the upscaling of the adapted model results to the application scale (150 l/h corresponds to $97 \text{ kg}/(\text{m}^2\text{s})$)

contributes the major part to the total heat flow, showing peak values of around 24 kW. In contrast, Stage 2 contributes the major part to the total heat flow in the mid and low temperature region, reaching peak values of around 37 kW.

Figure 5.28(b) shows the heat flow results versus temperature ($T_{w,S12}$) for controlled operation of 10 kW. A heat output of 10 kW can be achieved within the full operating temperature range, with a lower limit of 132 °C. For temperatures lower than that, the heat flow falls below 10 kW, even in coupled operation with full mass flow in each stage. In the high-temperature region and at start-up, only operation of Stage 1 is required. At approximately 350 °C, Stage 2 is activated, and the flow rate gradually increases as the temperature decreases. The coupled and controlled operation provides a high specific heat output over a wide operating temperature range. Once again, the thermomechanical load is significantly reduced because the required mass and heat flow of Stage 2 is minimized.

6 Comparison of concepts and evaluation at vehicle system level

This chapter compares and evaluates the two developed heat extraction concepts, the direct air and the multi-stage boiling concept, at a vehicle system level. Therefore, the discharge level and the degree of storage utilization are analyzed and further discussed. This is followed by a system-level comparison of volume, weight, and electrical power requirements for different scenarios.

6.1 Discharge level and storage utilization

To compare the developed concepts, the multi-stage boiling and the direct air discharge concepts are evaluated on the basis of the application-scale prototype size. The application-scale prototype is described in Section 4.4 and has a storage capacity of 25 kWh. This capacity could be increased by raising the height and mass of the storage material without changing the base plate dimensions. Comparing the concepts at an application-scale storage system size is more meaningful than at a small-scale prototype size because the former is more applicable to real-world scenarios.

Figure 6.1 shows a comparison of heat flow results for the multi-stage boiling and the direct air discharge concepts versus temperature. These results are based on application-scale storage dimensions (base plate area $A_{ref} = 0.25 \text{ m}^2$). The plotted air discharge heat flow was calculated using the validated air calculation model detailed in Sections 4.1 and 4.4. Here, a constant fan power of 85 W was defined, which was equal to a 100% fan power rate of the installed fan in the experiments. The plotted multi-stage boiling heat flow was calculated using the methods described in Section 5.4, including a scale up of the validated heat transfer model results to the application scale. A constant mass flow rate of 150 l/h (97 kg/(m²s)) per stage was defined, corresponding to a scale up of the highest mass flow rate in the experiments.

The heat flow for air discharge reaches a maximum of approximately 24.5 kW, decreasing as the temperature ($T_{w,AirHEX}$) falls. In comparison, the coupled heat flow of evaporator Stages 1 and 2 reaches a maximum of about 43 kW at a temperature of 275 °C ($T_{w,S12}$). Throughout the full plotted temperature range, the coupled multi-stage heat flow exceeds the air discharge heat flow. However, this is different when operating Stages 1 and 2 individually. Operating only Stage 1 results in a lower heat flow compared to the air discharge concept throughout the entire plotted temperature range. Operating only Stage 2 results

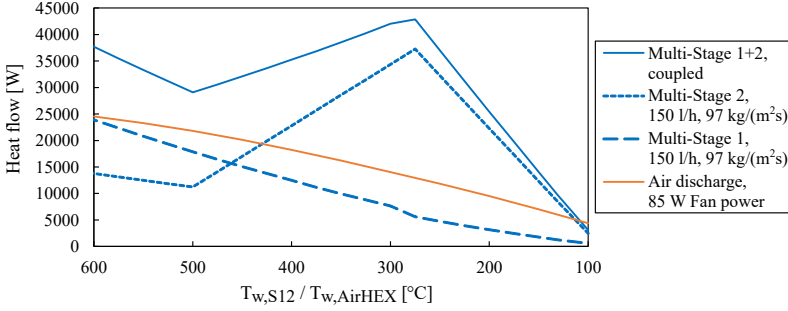


Figure 6.1: Comparison of heat flow versus temperature for multi-stage boiling and direct air discharge concepts based on application-scale storage size and developed calculation models

in a lower heat flow than the air discharge concept for temperatures above approximately 431 °C.

For the evaluation of the discharge level, the lower limit temperature is examined for a given heat flow. The lower limit temperature is defined as the lowest temperature for $T_{w,AirHEX}$, $T_{w,S1}$, and $T_{w,S2}$, at which the given heat flow can be achieved. This temperature definition is of practical relevance for a real application, as it defines the operating temperature range of the storage system. A reduction in the lower limit temperature yields a higher amount of heat that can be extracted and is therefore beneficial.

Figure 6.2 shows the results of the lower limit temperature versus heat flow for the multi-stage boiling and direct air discharge concepts. All heat flow results are based on the same reference base plate area ($A_{ref} = 0.25 \text{ m}^2$). Thus, a heat flow rate of 5 kW corresponds to a heat flux of 2 W/cm^2 at the base plate, 10 kW corresponds to 4 W/cm^2 , 15 kW corresponds to 6 W/cm^2 , and 20 kW corresponds to 8 W/cm^2 . In order to investigate the influence of the thermal resistance on the discharge performance, ideal ($R_{th} = ideal$) and real ($R_{th} = real$) condition cases were defined, and the results are plotted in Figure 6.2.

Under ideal conditions ($R_{th} = ideal$), there is zero thermal resistance from the base plate channel walls or air heat exchanger interface to the mPCM storage material. In this case, the total thermal resistance (R_{th}) is only affected by the thermal resistance in the heat transfer section. Therefore, the storage temperature is equal to the temperature at the interface with the heat transfer section ($T_{w,S12}$ and $T_{w,AirHEX}$). Thus, the thermal contact resistance at the mPCM-container interface and the thermal resistances due to conduction in solid parts are negligible. Consequently, the case of ideal conditions is purely theoretical to demonstrate the significant impact of thermal resistance on discharge performance.

Under real conditions ($R_{th} = real$), the thermal resistance between the mPCM and the base plate is similar to the thermal resistances observed in experiments with the bench-scale prototype. In this case, the total thermal resistance (R_{th}) is affected by all resistances from the mPCM to the discharge heat transfer fluid. As discussed and analyzed in Sections

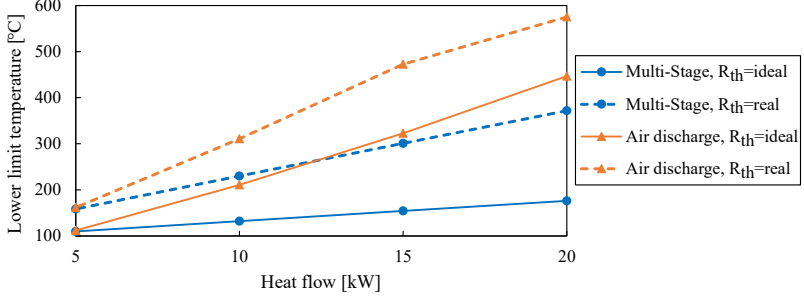


Figure 6.2: Lower limit temperature versus heat flow rate for multi-stage boiling and direct air discharge concepts ($A_{ref} = 0.25 \text{ m}^2$)

4.2 and 4.3, a large increase in thermal contact resistance at the mPCM-container interface occurs during solidification.

For the evaluation, only the thermal resistance at the lower limit temperature is relevant. The experimental results in Section 4.2 showed a temperature difference of around 60 K from the mPCM to the base plate with a constant heat flow of 1000 W over a wide temperature range in the solid state of the mPCM. Considering the reference base plate area of the prototype yields an effective resistance of $19.4 \text{ cm}^2\text{K/W}$ from the mPCM to the core base plate. This value is smaller than those determined for thermal contact resistance in Section 4.3. However, it takes into account the significant amount of sensible heat stored in storage components, such as the base plate, under transient conditions. Therefore, a thermal resistance of $20 \text{ cm}^2\text{K/W}$ is assumed from the mPCM to the core base plate for further estimations. Additionally, the thermal resistance from the core base plate to the heat transfer section interface ($T_{w,S12}$ and $T_{w,AirHEX}$) adds an additional 3–5 $\text{cm}^2\text{K/W}$, as estimated by Equation 5.10.

Depending on the configuration of future prototypes, the value for R_{th} could be somewhere between the real and ideal conditions discussed here, or even higher for an unfavorable design.

As plotted in Figure 6.2, the ideal thermal resistance case ($R_{th} = ideal$) results in a lower limit temperature of around $110 \text{ }^\circ\text{C}$ for both the multi-stage and the air discharge concepts, given a heat flow rate of 5 kW. In contrast, the real thermal resistance case ($R_{th} = real$) results in a lower limit temperature of approximately $160 \text{ }^\circ\text{C}$ for both concepts with the same heat flow rate. The lower limit temperature rises for both concepts and both considered resistance cases as the heat flow rate is increased. With higher heat flow rate, the selection of the heat extraction concept and the consideration of the thermal resistance case become more important. For a heat flow rate of 20 kW, the lower limit temperature is $176 \text{ }^\circ\text{C}$ with an ideal thermal resistance and the multi-stage boiling concept, but it increases to $372 \text{ }^\circ\text{C}$ with the real thermal resistance. For the air discharge concept, however, the lower limit temperatures are significantly higher. It is $454 \text{ }^\circ\text{C}$ for the ideal thermal resistance case and $577 \text{ }^\circ\text{C}$ for the real thermal resistance case.

For further evaluation and comparison of the heat extraction concepts, the storage utilization factor η_{SU} is introduced:

$$\eta_{SU} = \frac{\int_{T_{min,OP}}^{T_{max,OP}} c_p dt + \Delta h_f}{\int_{T_{amb}}^{T_{max,OP}} c_p dt + \Delta h_f} \quad (6.1)$$

The given equation is valid for $T_{max,OP} > \text{phase change temperature of mPCM} > T_{min,OP}$ and $T_{min,OP} > T_{amb}$. $T_{max,OP}$ and $T_{min,OP}$ are the maximum and minimum operating temperatures, respectively, and T_{amb} is the ambient temperature. The lower operating temperature ($T_{min,OP}$) corresponds to the lower limit temperature results illustrated in Figure 6.2. Thus, the storage utilization factor (η_{SU}) is the ratio of the extractable storage heat to the total available thermal energy for a given heat flow requirement.

Figure 6.3 shows the storage utilization factor (η_{SU}) versus heat flow for the multi-stage boiling and the direct air discharge concepts. A storage utilization of 100% corresponds to an ambient temperature of $T_{amb} = 20^\circ\text{C}$ in Equation 6.1. Again, the results are given for ideal and real thermal resistance cases. For the multi-stage boiling concept and the ideal thermal resistance case, the storage utilization is 89–94% over the full heat flow range shown in Figure 6.3. For the real thermal resistance case, the storage utilization is reduced to 72–90%. For the direct air discharge concept and the ideal resistance case, the storage utilization is 65–94%, and decreases to 51–90% for the real thermal resistance case. Consequently, the impact of the heat extraction concept and the thermal resistance case on storage utilization is relatively low at 5 kW (2 W/cm² corresponding to A_{ref}), but it increases significantly with higher heat flow rates.

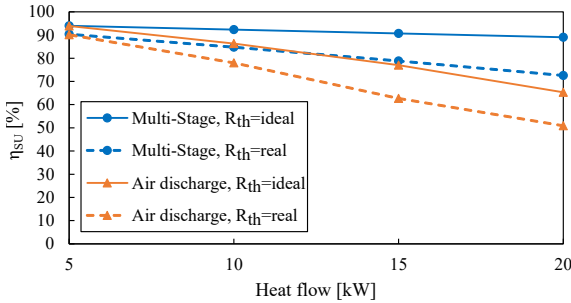


Figure 6.3: Storage utilization factor (η_{SU}) versus heat flow rate for multi-stage boiling and direct air discharge concepts ($A_{ref} = 0.25 \text{ m}^2$)

Figure 6.4 shows the improvement in storage utilization (η_{SU}) versus heat flow rate for the multi-stage boiling concept in comparison to the direct air discharge concept. Both concepts demonstrate similar storage utilization at lower heat flows of 5 kW (2 W/cm² corresponding to A_{ref}), with little influence from the thermal resistance case. As the

heat flow and heat flux increase, the multi-stage boiling concept's improvement in storage utilization becomes significant.

At a heat flow of 20 kW (8 W/cm² corresponding to A_{ref}), the improvement is 42% for the real resistance case and 36% for the ideal resistance case. Thus, the higher the required heat flux at the reference base plate area, the more advantageous the multi-stage boiling concept is compared to the direct air discharge concept. As an improvement in storage utilization allows for a reduction in the required storage material, the storage size and weight can be significantly reduced.

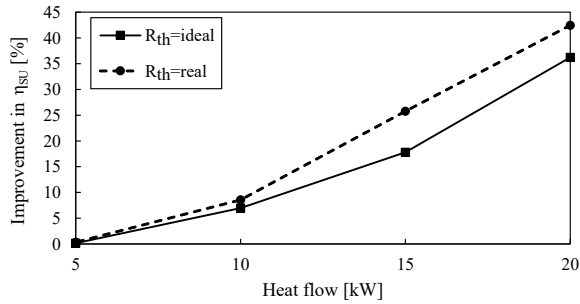


Figure 6.4: Improvement in storage utilization (η_{SU}) versus heat flow rate for the multi-stage boiling concept in comparison to the direct air discharge concept ($A_{ref} = 0.25 \text{ m}^2$)

6.2 Volume, weight, and electrical power requirements

6.2.1 Definition of scenarios and methodology

For the evaluation of the developed discharge concepts, the effects on volume, weight, and electrical power consumption of each system at the vehicle system level play a crucial role. So far, the focus of the investigations has been on the heat extraction process in the storage itself and has not considered such aspects. In addition to the thermal discharge of the storage system, the heat must be distributed throughout the vehicle cabin for heating. The required distribution system adds components, space, and weight. Even in the absence of a heat distribution system, an additional bypass system is required to control the outlet air temperature for the air discharge concept, and an air-heat exchanger and fan are required to transfer the extracted multi-stage system heat to the cabin air.

A valid comparison on a vehicle system level depends strongly on the individual vehicle size and detail design. Since this study does not focus on such a specific vehicle, but more on the fundamental context and important impact factors, different scenarios are defined, which consider a wide range of different possible vehicle types. For simplification,

three scenarios are defined, representing different vehicle sizes and distribution lengths. Table 6.1 shows the scenarios, the single-way distribution length and the corresponding exemplary vehicle types. The classification is made to show the general impact of the scenario and the distribution length, but is not intended to accurately represent the exact condition of an individual vehicle. With this in mind, all results presented later are an estimate rather than an exact representation of a real vehicle.

Scenario	Distribution length single-way	Exemplary vehicle system
1	0 m	Direct comparison without distribution system, any vehicle size
2	10 m	Small city bus, space station module
3	25 m	Train cabin, large city bus, short-haul aircraft

Table 6.1: Definition of scenarios for further comparison

For the direct air discharge concept, only the single path distribution length is used for further estimation. For the multi-stage boiling discharge concept, a water-glycol distribution system is considered. As this works in a closed circuit, twice the single-way distribution length is taken into account for further estimation.

Figure 6.5 shows a schematic of the air and water-glycol distribution systems. The air distribution system attributes to the direct air discharge concept, while the water-glycol distribution system attributes to the multi-stage boiling discharge concept. The TES arrangement, with its corresponding discharge system and components, is illustrated in more detail in Figures 3.5 and 3.7. The distribution systems shown in Figures 6.5(a)–(b) are only relevant for Scenarios 2 and 3. For Scenario 1, no distribution system is considered. These schematic distribution systems assume a centralized storage system located at the front or rear of the vehicle. There are many other ways to arrange the storage and its distribution system, such as a modularized storage approach. The distribution systems shown are simplified, and the results based on them are only indicative.

The direct air discharge distribution system consists mainly of pipes and air ducts. It is assumed that two separate air paths run along the vehicle as illustrated in Figure 6.5(a). The fan, bypass, and control valve are not shown, as they are considered to be part of the TES discharge system shown in Figure 3.5. For the multi-stage boiling discharge concept, the distribution system consists out of the water-glycol circuit piping, a liquid pump, and the air-water/glycol heat exchanger with associated air fans as illustrated in Figure 6.5(b).

The estimation of volume, weight, and power consumption is based on several assumptions and calculation steps. As a reference, the application scale prototype considered in the previous section for the storage utilization is chosen. It has a storage capacity of 25 kWh in the temperature range of 100–650 °C. As a reference for comparison, the volume and weight of the storage system without heat extraction system is considered. The

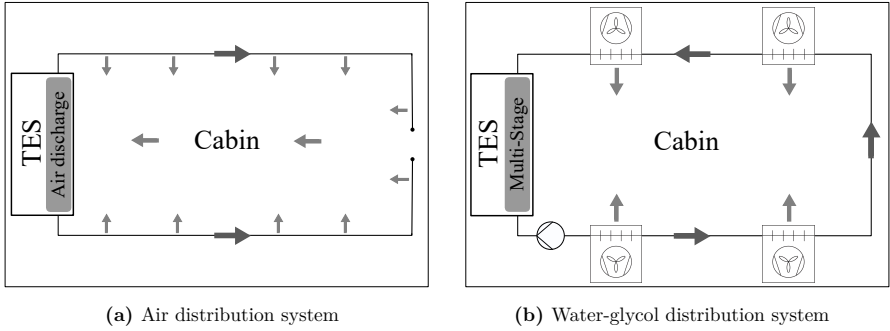


Figure 6.5: Schematic of the distribution systems

heat extraction and distribution systems are estimated based on an individual estimation of all components that are part of the direct air and multi-stage concepts, along with their corresponding distribution systems. The sizing of the components is based on an exemplary heat output of 10 kW for both concepts.

The volume, weight, and pressure drop estimation of the air heat transfer section is based on the presented lazy ruffled wavy fin design of the prototype presented in Section 4.4. The inlet and outlet diffusers are assumed to be made of stainless steel and, due to the high temperature, are surrounded by a 5 mm thick insulating layer. The bypass system is designed for an air velocity of 10 m/s at full mass flow. The required full mass flow is calculated assuming a temperature rise of 50 K from the storage inlet to the cabin outlet. This could be achieved with an air inlet temperature of 15 °C and an air outlet temperature of 65 °C. The actual bypass mass flow is then derived by subtracting the required mass flow through the air heat transfer section from the total mass flow. The distribution system is assumed to be circular and designed for an air velocity of 10 m/s at full mass flow. The full mass flow is present only at the inlet of the distribution system and decreases along the distribution length as hot air is supplied to the cabin. Therefore, an average air velocity of 6 m/s is assumed in the distribution piping for pressure drop estimation. The pressure drop is important for the estimation of the electrical power requirement of the fan.

Pressure drop estimation considers both the continuous increase in pressure loss in length of the flow path, as well as additional pressure drop due to sudden changes in the flow path, such as entrance and exit losses, diversions, and constrictions. The pressure drop estimation for the air heat transfer section was performed according to the methodology presented in Sections 4.1 and 4.4. For all other components, the pressure drop related to the Darcy friction factor f was calculated using Equation 2.28. The pressure drop related to the ζ -friction factor was calculated using Equation 2.35, with a ζ -factor of 1 assumed for the distribution system, 0.5 for the bypass system, and 1 for the inlet and outlet diffusers. The fan efficiency was assumed to be $\eta=0.3$, and the weight and volume were assumed to be the same as the model G3G190-RD45-03 (Ebm Papst). For the distribution and

bypass system, the piping and air ducts were assumed to be made out of common flexible thermoplastic elastomer (TPE) pipe materials, with a specific weight of 0.5 kg/m.

The weight and volume of the multi-stage boiling discharge concept was estimated based on the upscaled prototype presented in Section 5.4, along with reasonable engineering assumptions. The upscaled design is based on the results of the bench-scale prototype in Chapter 5. However, instead of one feed pump and one control valve, two individual pumps (Bosch PAD) without a control valve are considered for the boiling-condensation multi-stage loop. The water-glycol distribution system is designed for a fluid velocity of 1 m/s, and the fluid properties are based on a 50/50 water/glycol mixture. The corresponding mass flow is calculated under the assumption of a temperature difference of 5 K at the condenser inlet and outlet.

For the pressure drop calculation, a ζ -factor of 2 is assumed for abrupt pressure drops in the piping system and heat exchangers, while continuous pressure drops are estimated using Equation 2.28. The assumed material for the distribution pipes is ethylene propylene diene monomer rubber (EPDM), commonly used in vehicle cooling circuits, with a wall thickness of 3 mm. The fluid in the distribution system is driven by a liquid pump (Bosch PCE). No additional insulation is considered for the distribution system, while the condenser is surrounded by a 10 mm layer of insulation and the components of the multi-stage-loop outside of the main storage system have a 5 mm insulation thickness.

The air-water/glycol heat exchanger required to transfer the heat to the cabin is assumed to be an aluminum fin heat exchanger commonly used in vehicles. Therefore, a fin thickness of 0.1 mm, a fin height of 8 mm, a fin and channel length of 20 mm, a fin density of 600 fins per meter, and a plain fin design are considered. The height of the water/glycol fluid channels including the wall thickness is assumed to be 2 mm. The total heat exchanger size and weight are estimated by maintaining a 2:1 width to height ratio and stacking multiple layers of fins and channels. The weight was estimated by upscaling the measured weight of a real aluminum fin vehicle air-cooler in the lab. The heat transfer and pressure drop calculations are performed using the same equations as in Section 4.1. For the estimation, a temperature difference of 50 K between air inlet and water/glycol temperature is considered. Several air fans (e.g. EbmPapst 4118 N/2H7P), which are arranged in parallel, provide forced convection of the cabin air in the exchanger. The results provide a rough estimate rather than an exact heat exchanger design.

In Scenario 1, there is no distribution system and the condenser and air heat exchanger are integrated into a single heat exchanger. This is basically an air-based condenser with the working fluid of the multi-stage loop condensing on one side and air heat transfer on the other side. The design is based on a temperature difference of 75 K between the air inlet and the condensing process side. The geometry characteristics, fan selection, and calculation method are the same as for the air-water/glycol heat exchanger.

6.2.2 Results and discussion

The results presented in this section provide a rough estimate of the system volume, weight, and electrical power requirements for the scenarios defined in Table 6.1. While actual values may differ from real applications, the numbers provided indicate how the developed heat extraction concepts compare to each other at the system level and reveal important impact factors.

Figures 6.6(a)–(b) show the component weight and volume proportions for Scenario 2, for both the direct air system and the multi-stage boiling discharge system, as well as their respective vehicle distribution systems. The given percentage values illustrate how the volume and weight are distributed among the various components of the discharge and distribution systems. For Scenario 2, the single-way distribution distance is 10 m as given in Table 6.1, and the heat output rate is 10 kW.

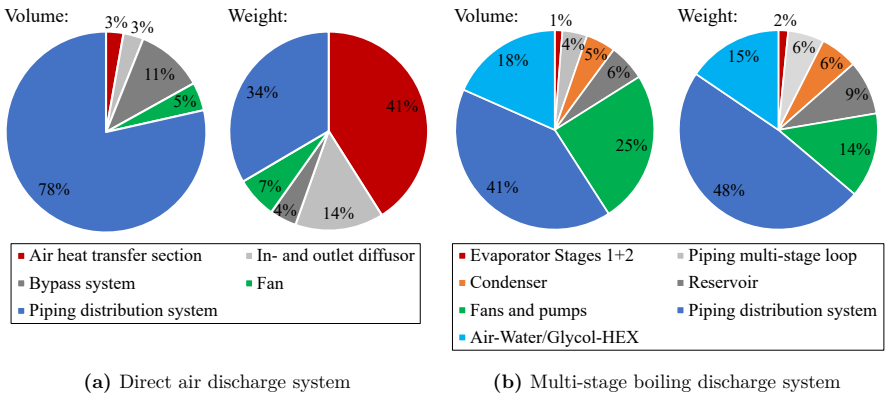


Figure 6.6: Component weight and volume proportions for Scenario 2

A large proportion of the volume and weight of both systems comes from the piping of the distribution systems. This is especially true for the volume of the air discharge system. The impact of the distribution system is even larger for Scenario 3, which has a single-way distribution length of 25 meters. For Scenario 1, which does not consider a distribution system, there is no such impact. The air heat transfer section has only a small portion of the volume of the air discharge system, while it has a large portion of the system weight. The evaporator Stages 1 and 2 represent only a small portion of the volume and weight of the multi-stage boiling discharge system, although they are the key technological part of the concept. In the multi-stage concept, more fans and pumps are involved than in the direct air discharge concept. As a result, the volume and weight proportions of the fans and pumps are significantly higher. In addition, the air-water/glycol heat exchangers have an apparent proportion of the system volume and weight of the multi-stage concept.

Of greater interest than the component proportions is a direct comparison of system volume and weight between the direct air and multi-stage boiling discharge concepts.

Figure 6.7 shows the normalized volume of both concepts based on a thermal output of 10 kW. The storage system without a heat extraction unit is shown in red and serves as a reference with a normalized volume of 1. In all three scenarios, the multi-stage design has a smaller volume than the air discharge design. This is true even for Scenario 1, which does not include a distribution system. Therefore, Scenario 1 can be regarded as a direct comparison of the two systems without the effect of a distribution system. The greater the distribution length, the larger the volume of both systems and the greater the difference between them. For Scenario 3, the total volume, including the heat extraction unit and distribution system, is reduced by over 90% for the multi-stage design compared to the air discharge design.

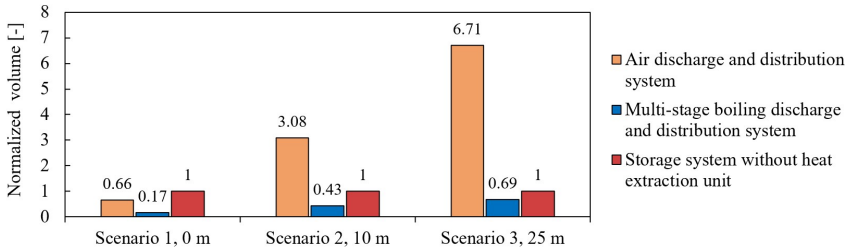


Figure 6.7: Normalized volume of direct air and multi-stage boiling discharge concepts including their distribution systems, based on 10 kW thermal design

Moreover, Figure 6.7 shows that the volume of the heat extraction unit with its corresponding distribution system has a significant volume impact compared to the storage system alone. Therefore, these findings and the appropriate design and selection of the heat extraction concept are of great practical importance for the real application.

Figure 6.8 shows the normalized weight of both concepts based on a thermal output of 10 kW. Once again, the storage system without a heat extraction unit is the reference, with a normalized weight of 1, and is shown in red. In all three scenarios, both concepts have significantly lower weights than the storage system. For Scenarios 2 and 3, the weight of the multi-stage concept and its distribution system is higher than that of the air discharge concept and its distribution system. For Scenario 1, the opposite is true. The main reason for this is that the water-glycol distribution system adds more weight due to the piping and the required amount of liquid, which is more than is needed for air distribution. However, the two heat extraction concepts differ much less in terms of system weight than in terms of system volume.

Based on the results, a reasonable optimization for weight reduction on a system level could be to combine the multi-stage boiling discharge concept with an air distribution system. This would then result in less weight even for Scenarios 2 and 3 compared to the results for the direct air discharge with distribution system. Considering this, the multi-stage concept can always provide a volume and weight advantage on a system level. Therefore, the use of the multi-stage design approach allows for efficient heat transfer into

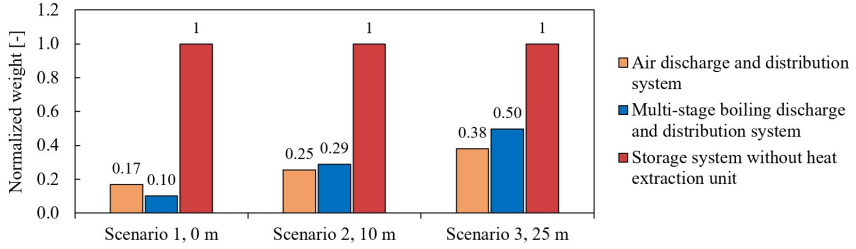


Figure 6.8: Normalized weight of direct air and multi-stage boiling discharge concepts including their distribution systems, based on 10 kW thermal design

a liquid and an air-based distribution system. As a result, it is a more flexible concept for integration into a vehicle system than the direct air discharge concept.

Furthermore, the results discussed in Figure 6.8 do not account for the increased storage utilization possible with the multi-stage system for heat extraction, as shown in Figure 6.3. For high specific heat output rates, the required storage capacity, and thus the weight of the storage system itself, can be significantly reduced. This results in reduced total weight when considering the sum of the storage system and the heat extraction unit with distribution system. However, at low specific heat output rates, there is no such benefit, as indicated in Figure 6.3.

Figure 6.9 illustrates the ratio of the estimated electrical power requirements for fans and pumps to the extracted heat flow at 10 kW ($P_{el}/\dot{Q}$). Results are given at the lower limit temperature for the three defined scenarios for the direct air and multi-stage boiling discharge designs. The power consumption is based on the individual consumption of all fans and pumps present in each scenario. For temperatures higher than the lower limit temperature, the ratio decreases, and for lower temperatures, the ratio increases. The ratio is in the order of 1% for both heat extraction designs. It increases for longer distribution distances and decreases for shorter distances. Consequently, both heat extraction concepts developed have the potential to provide efficient thermal discharge and heat supply on a system level.

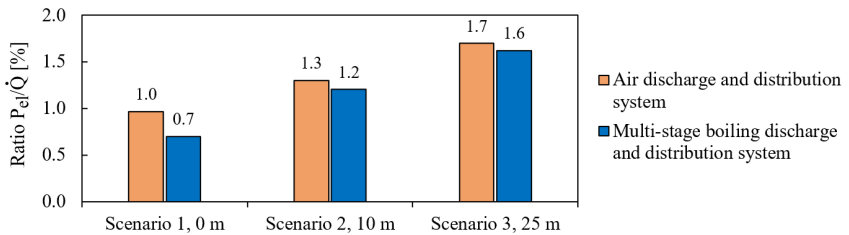


Figure 6.9: Ratio of estimated electrical power requirements for fans and pumps in relation to extracted heat flow at lower limit temperature

6.3 Evaluation of results

The discharge level and the degree of storage utilization have been evaluated for both the direct air and multi-stage boiling discharge concepts. In coupled operation, the multi-stage heat flow exceeds the direct air heat flow throughout the considered operating temperature range. The heat flux, thermal resistance, and discharge concept applied have a strong impact on the discharge level and the storage utilization. The higher the required heat flux at the reference base-plate area, the more advantageous the multi-stage boiling concept becomes compared to the direct air-discharge concept in terms of heat-extraction performance. At heat fluxes below 2 W/cm^2 , there is no such advantage, and therefore, the application of an air discharge concept becomes reasonable. By keeping thermal resistances low, the storage utilization can be significantly increased. The thermal contact resistance at the mPCM-container interface has a major impact and should therefore be reduced for future application.

For further comparison, the volume, weight, and electric power requirements were analyzed. On a system level, the distribution system plays an important role in the heat supply of the vehicle. The greater the distribution length, the larger the volume and weight requirement. In terms of system volume, the multi-stage boiling concept is advantageous in all considered scenarios compared to the air discharge concept. In terms of weight, the multi-stage boiling concept is only advantageous if no water-glycol distribution system is needed or if the benefits of increased storage utilization allow for a relevant reduction in storage weight. Therefore, an optimization in overall system weight is possible by extracting the storage heat via the multi-stage boiling concept and transferring it via a heat exchanger directly to an air distribution system instead of a water-glycol distribution system. The electrical power requirements for fans and pumps are moderate for all scenarios considered, allowing for efficient thermal discharge and heat supply at the vehicle system level for both concepts.

This study aims at the fundamentals and technological approach of the heat extraction of a high-temperature mPCM thermal energy storage and does not consider a detailed cost analysis of the developed concepts. For the analysis of the resulting costs per kW and kWh, the total lifetime and operating conditions have to be taken into account. These are highly application dependent and have not been defined at this stage. These include the manufacturing costs of all components of the storage and heat extraction system, total quantities, operation and maintenance costs, energy costs for electrical charging, and more. All parts of the heat extraction and distribution system can be realized by state-of-the-art components and manufacturing technologies. A detailed cost analysis has to be considered in future studies.

7 Conclusions and outlook

This thesis focuses on the development and investigation of heat extraction concepts for thermal energy storage based on metallic phase change materials in mobile applications. Two main concepts were selected for further prototyping, simulation, and experimental investigation based on a conceptual development. The conclusions of this work are correspondingly split into the results of the developed direct air and multi-stage boiling discharge concepts. Finally, there is an outlook on future optimization, investigation, and application potentials.

7.1 Conclusions

The main research questions stated in Chapter 1 that motivated the research of this thesis were:

How can a high-temperature metallic phase change thermal energy storage be discharged in the context of a mobile application?

What are important physical effects and performance characteristics of the thermal discharge of such a storage system?

Which functional and technical design characteristics lead to an efficient thermal discharge system?

The conceptual development, prototyping, simulation, and experiments were conducted and evaluated with respect to these questions.

The direct air discharge concept was considered to be the simplest approach and directly transfers the heat to the cabin air. The experimental investigation of a bench- and application-scale prototype was assisted by a conjugate CFD simulation and the development of a heat transfer calculation model.

The following main conclusions were drawn from the direct air discharge results:

1. Experiments were carried out successfully across an operating temperature range of 650–100 °C, with high level of functionality and controllability. A proof of concept of the air discharge concept was achieved.
2. High specific discharge performance was realized at high storage temperatures, while the performance tends to become insufficient at storage temperatures below 200–300 °C.

3. The conjugate numerical simulation showed good agreement with the experimental data for the air discharge time and heat flow, indicating the model's successful validation. The mPCM solidifies faster near to the air inlet.
4. Based on the simulation and experiments, a strong effect of the phase change on the thermal contact resistance at the AlSi_{12} interface was observed. The contact resistance is lowest in the liquid state of the AlSi_{12} , increases significantly during the phase change, and continues to increase with decreasing temperature. The temperature-dependent values of the thermal contact resistance were determined.
5. The accuracy of the developed heat transfer calculation model results in comparison with the experimental data implies that the model can be considered validated and used to estimate the heat flow. It provides a relatively simple and efficient method for designing and estimating future prototypes.
6. Plain and lazy ruffled wavy fins showed to be the most effective air heat transfer structure in terms of heat transfer and required electrical power of the fan. A ratio of around 1% of the fan's electric power requirement to the total extracted heat flow ($P_{el} / \dot{Q}_{ex}$) at the lower limit temperature proved to be a constructive selection criterion.

The second concept developed was considered to be more sophisticated and is based on a novel multi-stage boiling approach. It addresses a practical engineering challenge, which is known as the boiling crisis, and has significant implications for energy efficiency and real-world applications. The integration of a two-stage boiling process within a TES system represents a novel approach. Heat extraction was achieved through a closed-loop boiling and condensing cycle using water as the working fluid, allowing heat to be transferred to a vehicle liquid cooling circuit. The evaporator consists of two distinct boiling stages. Stage 1 uses tubes, which are mechanically decoupled from the storage system, while Stage 2 works with direct contact fluid channels. The experimental investigation of a bench-scale prototype was assisted by the simulation of a 2D-temperature profile. Based on the experimental results, a heat transfer calculation model was derived.

The following main conclusions were drawn from the multi-stage boiling discharge results:

1. Experiments were conducted successfully across an operating temperature range of 650–100 °C. Based on the experimental results, a proof of concept of the novel multi-stage heat extraction concept was achieved.
2. High specific heat discharge rates were observed at high, medium, and low storage temperatures, with peak values exceeding 20 W/cm². The results demonstrate that the system effectively utilizes both latent and sensible heat of the storage material, resulting in high efficiency and low pumping power requirements.
3. The occurrence of the boiling crisis was observed at high temperatures in the direct contact fluid channels of Stage 2. In contrast, the Stage 1 tube arrangement effectively prevented the boiling crisis.

4. The multi-stage approach showed an improved thermal performance by a factor of two in the liquid state of the AlSi_{12} compared to a direct contact boiling heat transfer. At the same time, the temperature gradients in the storage system were reduced by more than 220 K. This feature can extend the lifespan of the storage system, as rapid thermal transients can cause thermomechanical damage to mechanical components due to high stresses and thermostructural fatigue.
5. Under pressure-tight conditions, the absolute pressure in the discharge system remained below 1.4 bar, which is uncritical for real-world vehicle applications.
6. The developed heat transfer calculation model was adapted to the experimental data, allowing for estimation of the heat flow. The results demonstrate that coupled, controlled operation achieves a high specific heat output over a wide operating temperature range.

For further evaluation, the two developed and investigated thermal discharge concepts were compared to each other on a vehicle system level. The heat flux, thermal resistance, and applied discharge concept strongly impact the discharge level and degree of storage utilization. The higher the required heat flux at the reference base plate area, the more advantageous the multi-stage boiling concept becomes compared to the direct air discharge concept in terms of heat extraction performance. In terms of system volume and weight, the vehicle heat distribution system plays an important role. The multi-stage boiling concept is capable of efficiently transferring heat to an air- and/or water-glycol-based distribution system. In contrast, the direct air discharge concept is limited to an air-based distribution system. Therefore, the multi-stage boiling concept offers greater flexibility in terms of the distribution system. Without a distribution system, the multi-stage concept is more beneficial in terms of system volume and weight compared to the direct air discharge system. Both concepts allow for efficient thermal discharge with regard to electrical power requirements for fans and pumps.

In conclusion, the main research questions were answered. For the first time, this thesis demonstrated the functionality, performance, and efficiency of two different approaches to heat extraction for metallic latent heat thermal energy storage systems. It also characterized the complex physics of the heat extraction process. The novel technical concepts proposed and the new physical insights gained during this research extend the state of the art and provide the basis for further development in this field.

7.2 Outlook

Although the results of this thesis extend the state of the art, future development and investigation studies have several issues to consider.

First of all, a major improvement with regard to thermal discharge is expected by reducing the thermal contact resistance at the mPCM interface, which applies to all concepts proposed in this work. This could be achieved by adapting the containment geometry

or the arrangement of storage components to create a pressure boundary between the mPCM and the surrounding components. Suggestions for improvement include adding fin structures to the housing in contact with the mPCM or integrating the heat extraction system directly into the mPCM without other solid component layers in between. The multi-stage boiling concept could be further improved by increasing the peak heat flow of Stage 1 to match the values of Stage 2 by optimizing the gap width at the inner tube intersection. The principle of the novel multi-stage boiling concept can be flexibly adapted to other geometries, storage concepts, and operating temperature ranges. The results gained are also the basis for further development of single-stage boiling concepts, such as direct boiling or resistance boiling, which were introduced in Chapter 3. The air discharge system's thermal performance could be further enhanced by using nickel instead of stainless steel as the fin material due to nickel's higher thermal conductivity.

So far, experiments with the multi-stage concept have only been performed with constant mass flow rates and individual operation of the stages. Future experiments need to implement coupled, controlled operation of both stages to achieve constant heat flow across the entire operating temperature range. Additionally, the complex boiling phenomena observed require further experimentation and simulation studies.

The final step for a future application in a vehicle would then be to implement and test the storage system and heat extraction unit in a real vehicle environment with all relevant scenarios that can occur. Depending on the individual vehicle system and boundary conditions, further adjustments in size and geometry may be necessary. In the short-run, the storage system has the greatest potential in battery-electric buses and trains. Future studies need to evaluate the technology's potential in other fields, such as stationary energy storage, electric aircraft (e.g., drones and urban air mobility vehicles), and spacecraft (e.g., satellites and space stations).

Appendix

A Relevant fluid and material properties

A.1 Material description of components

Bench-scale air discharge and multi-stage discharge prototypes:

Geometry	Material	Properties used in this thesis are based on data from
Steel container	Stainless steel alloy 1.4980	[113]
Graphite container	Graphite R7660	DLR internal data set
Graphite bottom plate	Graphite R7510	DLR internal data set
Base plate	Stainless steel alloy 1.4980	[113]
Fins (only air discharge prototype)	Stainless steel alloy 1.4980	[113]
Bottom cover plate (only air discharge prototype)	Stainless steel alloy 1.4301	[25]
Tubes Stages 1 and 2 (only multi-stage prototype)	Stainless steel alloy 1.4571	[25]

Table A.1: Material description of bench-scale air discharge and multi-stage discharge prototypes

Application-scale air discharge prototype:

Geometry	Material	Properties used in this thesis are based on data from
Base plate	Stainless steel alloy 1.4828	[25]
Fins	Stainless steel alloy 1.4404	[25]
Bottom cover plate	Stainless steel alloy 1.4301	[25]

Table A.2: Material description of application-scale air discharge prototype

Other materials for theoretical investigations:

The material properties of nickel (Ni201) are taken from [114] and of aluminum are taken from [61].

A.2 Storage material

The temperature-dependent material properties of AlSi_{12} used in this study are based on DLR project report data [87]. The density of AlSi_{12} ranges from 2646 kg/m^3 at 25°C to 2421 kg/m^3 at 650°C .

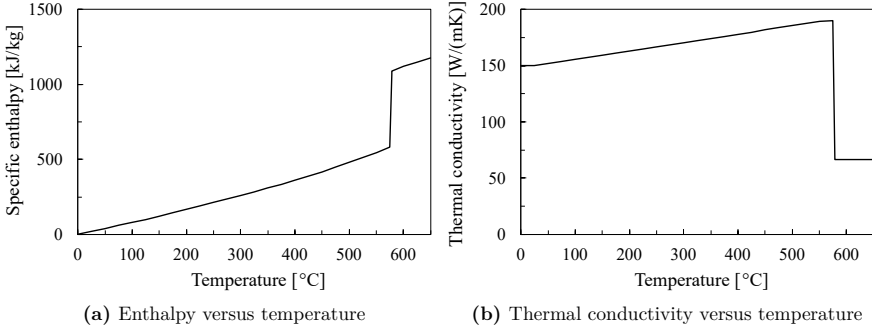


Figure A.1: Selected material properties of the storage material AlSi_{12} , based on data from [87]

A.3 Heat transfer fluids

The temperature-dependent fluid properties of water and air used in this thesis are based on data from [42]. At atmospheric pressure, the density of water changes from 958 kg/m^3 to 0.6 kg/m^3 during the phase change from liquid to vapor, and its thermal conductivity changes from 0.68 W/(mK) to 0.02 W/(mK) [42]. The fluid properties of water-glycol mixture are based on data from [97].

Water:

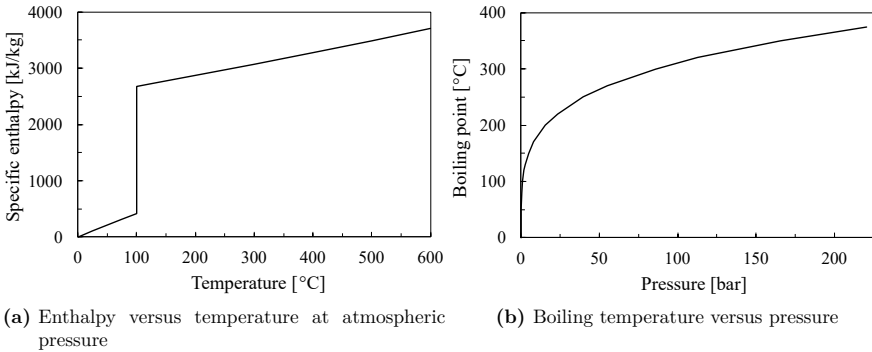


Figure A.2: Selected fluid properties of water, based on data from [42]

Air:

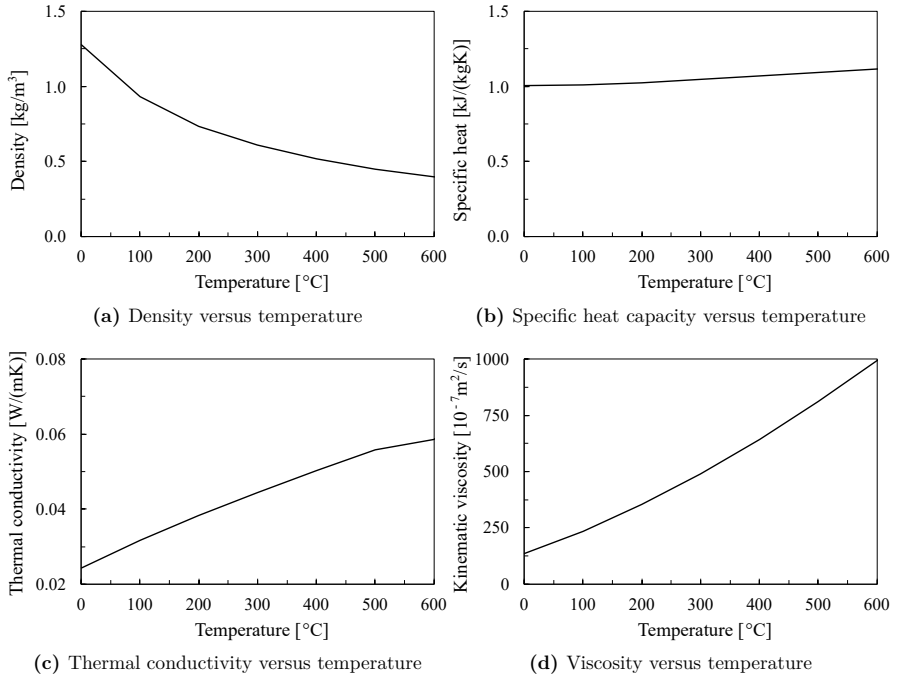


Figure A.3: Selected fluid properties of air, based on data from [42]

B Supplemental information on measurement uncertainties

B.1 Theoretical basis

The measurement uncertainties were determined in two steps:

1. Determination of the uncertainties of the single component measured variables based on manufacturer's data sheets and calibration reports, such as temperature, mass flow, absolute pressure, and differential pressure.
2. Determination of the measurement uncertainty based on the combination and propagation of the aforementioned uncertainties for heat flow or combined measured variables.

The combination and propagation of measurement uncertainties were conducted in accordance with the JCGM guidelines [41]. The combined uncertainty (u_c) of a measured variable (y) is calculated as the square root of the sum of the squares of the uncorrelated single-component uncertainties (u_i):

$$u_c(y) = \sqrt{\sum_{i=1}^N u_i^2(y)} \quad (\text{B.1})$$

The propagation of uncertainties for uncorrelated input quantities (x_i) is given by:

$$u_c(y) = \sqrt{\sum_{i=1}^N \left(\frac{\delta f}{\delta x_i} \right)^2 u^2(x_i)} \quad (\text{B.2})$$

The specified uncertainties were rounded up to the specified decimal place.

B.2 Direct air discharge – Bench-scale prototype

This section is a supplement to Subsection 4.2.3.

The uncertainty of $T_{air,out}$ and T_{BP} based on propagation of single temperature measurement uncertainties is:

$$u_c(T_{air,out}) = u_c(T_{21}, T_{22}, T_{23}) = 0.7 \text{ K}$$

$$u_c(T_{BP}) = u_c(T_{31}, T_{32}, T_{33}, T_{41}, T_{42}, T_{43}) = 0.5 \text{ K}$$

Thermocouple	Reference PT100/type-S ^{a)} [K]	Calibration setup ^{b)} [K]	Regression error ^{c)} [K]	Combined uncertainty ^{d)} [K]
T_0	0.04	0.1	0.17	0.2
T_{11}, T_{12}, T_{13}	1.0	0.1	<0.14	<1.1
T_{21}, T_{22}, T_{23}	1.0	0.1	<0.17	<1.1
$T_{31}, T_{32}, T_{33}, T_{41}, T_{42}, T_{43}$	1.0	0.1	<0.97	<1.4
T_{mPCM}	1.0	0.1	0.34	1.1

^{a)} The uncertainty of the Pt-100 and Typ-S reference sensors is based on the latest calibration report and is given for the relevant temperature range of the thermocouple in the experiments.

^{b)} Additional error due to calibration setup.

^{c)} The regression error caused by the smoothing function of the reference versus the measured temperatures was evaluated as the root mean square error (RMSE).

^{d)} The combined uncertainty is calculated using Equation B.1.

Table B.1: Uncertainties of single temperature measurements at various positions for bench-scale prototype experiments

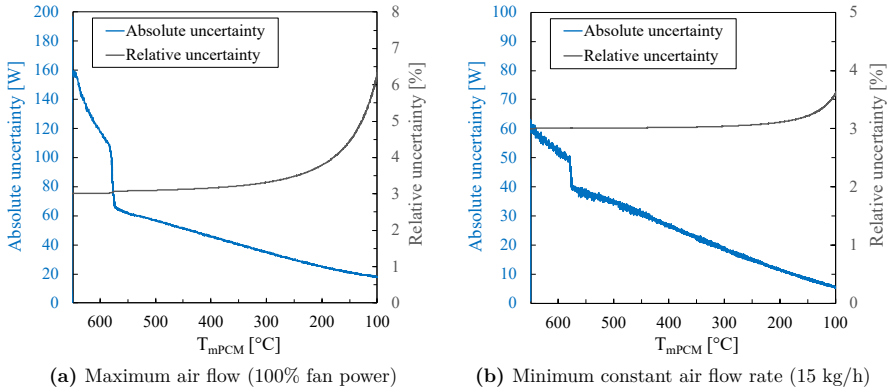


Figure B.1: Absolute and relative uncertainty of experimental heat flow versus mPCM temperature for bench-scale prototype experiments

B.3 Direct air discharge – Application-scale prototype

This section is a supplement to Subsection 4.4.3.

The uncertainty of $T_{air,out}$ and $T_{w,AirHEX}$ temperatures based on propagation of single temperature measurement uncertainties is:

$$u_c(T_{air,out}) = u_c(T_{21}, T_{22}, T_{23}, T_{24}, T_{25}) = 0.7 \text{ K}$$

$$u_c(T_{w,AirHEX}) = 0.8 \text{ K}$$

Thermocouple	Reference PT100/type-S ^{a)} [K]	Calibration setup ^{b)} [K]	Regression error ^{c)} [K]	Combined uncertainty ^{d)} [K]
T_0	0.05	0.1	0.19	0.3
$T_{11}, T_{12}, T_{13}, T_{14}, T_{15}$	1.0	0.1	<0.36	<1.1
$T_{21}, T_{22}, T_{23}, T_{24}, T_{25}$	1.0	0.1	<1.32	<1.7
$T_{31}, T_{32}, T_{33}, T_{34}, T_{35}, T_{36}$	1.0	0.1	<0.46	<1.2
$T_{41}, T_{42}, T_{43}, T_{44}, T_{45}, T_{46}$	1.0	0.1	<0.83	<1.4

^{a)} The uncertainty of the Pt-100 and Typ-S reference sensors is based on the latest calibration report and is given for the relevant temperature range of the thermocouple in the experiments.

^{b)} Additional error due to calibration setup.

^{c)} The regression error caused by the smoothing function of the reference versus the measured temperatures was evaluated as the root mean square error (RMSE).

^{d)} The combined uncertainty is calculated using Equation B.1.

Table B.2: Uncertainties of single temperature measurements at various positions for application-scale prototype experiments

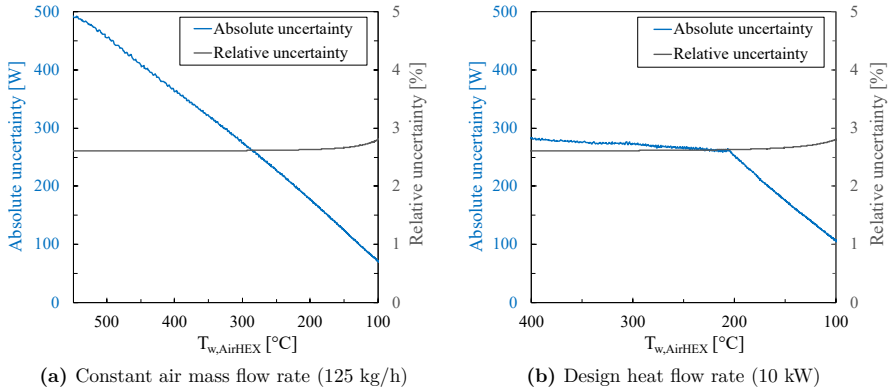


Figure B.2: Absolute and relative uncertainty of experimental heat flow versus $T_{w,AirHEX}$ temperature for application-scale prototype experiments

B.4 Multi-stage boiling discharge prototype

This section is a supplement to Subsection 5.2.4.

The uncertainty of the base plate temperature (T_{BP}) based on propagation of single temperature measurement uncertainties is:

$$u_c(T_{BP}) = u_c(T_{40}, T_{41}, T_{42}, T_{43}, T_{44}, T_{45}) = 0.5 \text{ K}$$

Thermocouple	Reference PT100/type-S ^{a)} [K]	Calibration setup ^{b)} [K]	Regression error ^{c)} [K]	Combined uncertainty ^{d)} [K]
T_1	0.04	0.1	0.07	0.13
T_{21}	0.04	0.1	0.07	0.13
T_{22}	0.04	0.1	0.07	0.13
T_{23}	0.04	0.1	0.07	0.13
$T_{40}, T_{41}, T_{42}, T_{43}, T_{44}, T_{45}$	1.0	0.1	<0.77	<1.3
T_5 (T_{mPCM})	1.0	0.1	0.14	1.1

- ^{a)} The uncertainty of the Pt-100 and Typ-S reference sensors is based on the latest calibration report and is given for the relevant temperature range of the thermocouple in the experiments.
- ^{b)} Additional error due to calibration setup.
- ^{c)} The regression error caused by the smoothing function of the reference versus the measured temperatures was evaluated as the root mean square error (RMSE).
- ^{d)} The combined uncertainty is calculated using Equation B.1.

Table B.3: Uncertainties of single temperature measurements at various positions for multi-stage prototype experiments

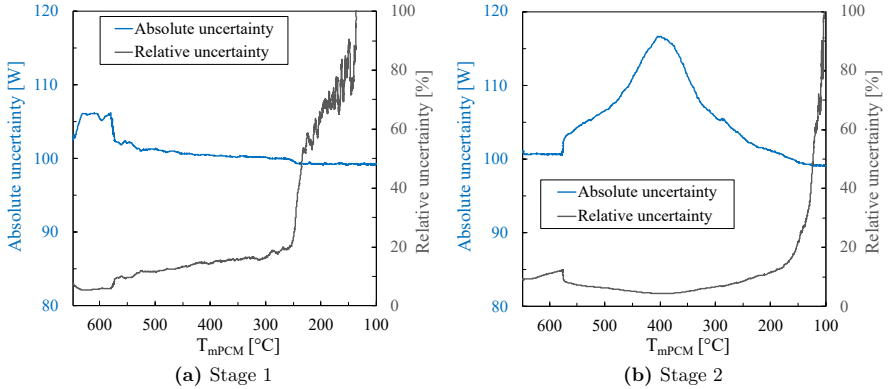


Figure B.3: Absolute and relative uncertainty of experimental heat flow for Stages 1 and 2 versus mPCM temperature for multi-stage prototype experiments (for 10 l/h, 64 kg/(m²s))

C Grid convergence study for numerical simulations

The discretization error of the numerical mesh used in Section 4.3 is estimated by a grid convergence study. A state-of-the-art method to estimate the discretization error is the Grid Convergence Index (GCI), which is based on the Richardson extrapolation [92, 93]. In this work, the step-by-step method for determining the GCI according to Celik [13] is applied. In this approach, the mesh is refined in a structured way by using a constant refinement factor, and the extrapolated values for the field variables and the discretization errors are estimated.

- Step 1: The global grid size h is obtained from the volume of the cells ΔV_i using the formula:

$$h = \left[\frac{1}{N} \sum_{i=1}^N (\Delta V_i) \right]^{1/3} \quad (\text{C.1})$$

- Step 2: Three different grid sizes h_1 , h_2 , and h_3 are selected using a constant grid refinement factor r ($h_1 < h_2 < h_3$, and $r_{21} = h_2/h_1$, $r_{32} = h_3/h_2$). Simulations are performed on all three meshes and the resulting values of the relevant field variables ϕ are reported.
- Step 3: The apparent order p of the method is calculated using the expression:

$$p = \frac{1}{\ln(r_{21})} |\ln|\varepsilon_{32}/\varepsilon_{21}| + q(p)|, \quad (\text{C.2})$$

$$q(p) = \ln \left(\frac{r_{21}^p - s}{r_{32}^p - s} \right), \quad (\text{C.3})$$

$$s = 1 \cdot \text{sign}(\varepsilon_{32}/\varepsilon_{21}), \quad (\text{C.4})$$

with $\varepsilon_{32} = \phi_3 - \phi_2$ and $\varepsilon_{21} = \phi_2 - \phi_1$

- Step 4: The extrapolated values are calculated using the following equations:

$$\phi_{ext}^{21} = (r_{21}^p \phi_1 - \phi_2) / (r_{21}^p - 1) \quad (\text{C.5})$$

$$\phi_{ext}^{32} = (r_{32}^p \phi_2 - \phi_3) / (r_{32}^p - 1) \quad (\text{C.6})$$

- Step 5: The approximate error and the extrapolated relative error are calculated as follows:

$$e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right| \quad (C.7)$$

$$e_{ext}^{21} = \left| \frac{\phi_{ext}^{12} - \phi_1}{\phi_{ext}^{12}} \right| \quad (C.8)$$

The GCI fine grid convergence index is calculated as:

$$GCI_{fine}^{21} = \frac{1.25e_a^{21}}{r_{21}^p - 1} \quad (C.9)$$

The factor 1.25 is a safety factor that provides a sufficient safety margin for grid convergence studies based on three grids [94]. To check for asymptotic convergence, it is observed whether the value of the field variable tends toward the extrapolated value as the mesh is refined. Mesh selection is based on computational capacity and a reasonable amount of accuracy. Table C.1 shows the results of the calculation procedure for the three selected grids. The field variables ϕ_1 , ϕ_2 , and ϕ_3 result from numerical simulations.

Parameter	Unit	$\dot{Q}_{ex}$	$\dot{Q}_{ex}$
		$T_{mPCM} = 600 \text{ }^\circ\text{C}$	$T_{mPCM} = 300 \text{ }^\circ\text{C}$
r_{21}	[-]	2.0	2.0
r_{32}	[-]	2.0	2.0
ϕ_1	[W]	1771.25	702.97
ϕ_2	[W]	1774.84	704.52
ϕ_3	[W]	1785.65	709.85
p	[-]	1.59	1.78
ϕ_{ext}^{21}	[W]	1769.46	702.33
ϕ_{ext}^{32}	[W]	1769.46	702.33
e_a^{21}	[%]	0.20	0.22
e_a^{32}	[%]	0.61	0.76
e_{ext}^{21}	[%]	0.10	0.09
e_{ext}^{32}	[%]	0.30	0.31
GCI_{fine}^{21}	[%]	0.13	0.11
GCI_{medium}^{32}	[%]	0.38	0.39

Table C.1: Estimation of the discretization error of the conjugate numerical simulation under steady state conditions and 15 kg/h air mass flow rate

References

- [1] J. D. Anderson. *Computational fluid dynamics: The basics with applications*. McGraw-Hill International, New York, 2006. ISBN 0071132104.
- [2] R. Anderson, L. Bates, E. Johnson, and J. F. Morris. Packed bed thermal energy storage: A simplified experimentally validated model. *Journal of Energy Storage*, 4:14–23, 2015. doi:10.1016/j.est.2015.08.007.
- [3] C. E. Andraka. Dish stirling advanced latent storage feasibility. *Energy Procedia*, 49:684–693, 2014. doi:10.1016/j.egypro.2014.03.074.
- [4] ANSYS, Inc. *Ansys Fluent Theory Guide*. Release 2022 R2, 2022.
- [5] H. D. Baehr and K. Stephan. *Wärme- und Stoffübertragung*. Springer, Berlin, Heidelberg, 2013. ISBN 978-3-642-36557-7. doi:10.1007/978-3-642-36558-4.
- [6] V. Becattini, L. Geissbühler, G. Zanganeh, A. Haselbacher, and A. Steinfeld. Pilot-scale demonstration of advanced adiabatic compressed air energy storage, Part 2: Tests with combined sensible/latent thermal-energy storage. *Journal of Energy Storage*, 17:140–152, 2018. doi:10.1016/j.est.2018.02.003.
- [7] G. K. Bloch. *An Experimental Study on Vertical Subcooled Flow Boiling Under the Influence of Turbulence and Secondary Flows*. Dissertation, Technische Universität München, 2016.
- [8] F. P. Bowden, D. Tabor, and F. Palmer. The friction and lubrication of solids. *American Journal of Physics*, 19(7):428–429, 1951. doi:10.1119/1.1933017.
- [9] N. Breidenbach, C. Martin, H. Jockenhöfer, and T. Bauer. Thermal energy storage in molten salts: Overview of novel concepts and the DLR test facility TESIS. *Energy Procedia*, 99:120–129, 2016. doi:10.1016/j.egypro.2016.10.103.
- [10] L. A. Bromley. Heat transfer in stable film boiling. *Chemical Engineering Progress*, (46):221–227, 1950.
- [11] A. Bruch, J. F. Fourmigué, and R. Couturier. Experimental and numerical investigation of a pilot-scale thermal oil packed bed thermal storage system for CSP power plant. *Solar Energy*, 105:116–125, 2014. doi:10.1016/j.solener.2014.03.019.
- [12] J. Buschbeck, S. Yang, S. von Mach, S. Donner, and W. Kraft. Thermal high performance storage systems for battery electrical rail vehicles. *ZEVrail*, (146): 43–49, 2022.

- [13] I. Celik, U. Ghia, P. Roache, and C. Freitas. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *ASME Journal of Fluids Engineering*, 130(7):078001, 2008. doi:10.1115/1.2960953.
- [14] J. G. Collier and J. R. Thome. *Convective Boiling and Condensation*. Clarendon Press, Oxford, 3th edition, 1994. ISBN 0198562829.
- [15] Deutsche Raumfahrtagentur im DLR. Infused Thermal Solutions (ITS) Forschungsvorhaben: Latentwärmespeicher eingebettet in additiv gefertigte Integralstrukturen zur passiven Thermalkontrolle. Available online: <https://www.dlr-innospaace.de/gefoerderte-projekte/infused-thermal-solutions/> (accessed: Aug. 26, 2025).
- [16] N. Dicke. *Untersuchung und Optimierung von Konzepten thermischer Hochleistungsspeicher*. Masterthesis, Universität Stuttgart, 2019.
- [17] V. Dreißigacker. Solid media thermal energy storage system for heating electric vehicles: Advanced concept for highest thermal storage densities. *Applied Sciences*, 10(22):8027, 2020. doi:10.3390/app10228027.
- [18] V. Dreißigacker and S. Belik. High temperature solid media thermal energy storage system with high effective storage densities for flexible heat supply in electric vehicles. *Applied Thermal Engineering*, 149:173–179, 2019. doi:10.1016/j.applthermaleng.2018.12.026.
- [19] V. Dreißigacker and L. Hofer. High-performance solid medium thermal energy storage system for heat supply in battery electric vehicles: Proof of concept and experimental testing. *Applied Sciences*, 12(21):10943, 2022. doi:10.3390/app122110943.
- [20] P. Drögemüller. The use of hitran wire matrix elements to improve the thermal efficiency of tubular heat exchangers in single and two-phase flow. *Chemie Ingenieur Technik*, 87(3):188–202, 2015. doi:10.1002/cite.201400081.
- [21] R. Fukahori, T. Nomura, C. Zhu, N. Sheng, N. Okinaka, and T. Akiyama. Thermal analysis of Al-Si alloys as high-temperature phase-change material and their corrosion properties with ceramic materials. *Applied Energy*, 163:1–8, 2016. doi:10.1016/j.apenergy.2015.10.164.
- [22] A. Giovannelli and M. Anser Bashir. Charge and discharge analyses of a PCM storage system integrated in a high-temperature solar receiver. *Energies*, 10(12):1943, 2017. doi:10.3390/en10121943.
- [23] V. Gnielinski. New equations for heat and mass transfer in turbulent pipe and channel flow. *International Chemical Engineering*, (Vol. 16, No. 2):359–368, 1976.
- [24] D. Gorenflo. Behältersieden. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.

-
- [25] Granta Design Limited. *Cambridge Engineering Selector (CES), Version 2019, Database: JAHM Curve Data*. 2022.
 - [26] D. C. Groeneveld. Inverted annular and low quality film boiling: A state-of-the-art report. *Proceedings of the 1st Intern. Workshop on Fundamental Aspects of Post-dryout Heat Transfer, Salt Lake City*, 1984.
 - [27] D. C. Groeneveld, L. Leung, P. L. Kirillov, V. P. Bobkov, I. P. Smogalev, V. N. Vinogradov, X. C. Huang, and E. Royer. The 1995 look-up table for critical heat flux in tubes. *Nuclear Engineering and Design*, 163(1-2):1–23, 1996. doi:10.1016/0029-5493(95)01154-4.
 - [28] D. C. Groeneveld, L. Leung, A. Z. Vasic, Y. J. Guo, and S. C. Cheng. A look-up table for fully developed film-boiling heat transfer. *Nuclear Engineering and Design*, 225(1):83–97, 2003. doi:10.1016/S0029-5493(03)00149-3.
 - [29] H. Großmann. *Pkw-Klimatisierung: Physikalische Grundlagen und technische Umsetzung*. Springer, Berlin, Heidelberg, 2013. ISBN 978-3-642-39840-7. doi:10.1007/978-3-642-39841-4.
 - [30] M. Gumus. Reducing cold-start emission from internal combustion engines by means of thermal energy storage system. *Applied Thermal Engineering*, 29(4):652–660, 2009. doi:10.1016/j.applthermaleng.2008.03.044.
 - [31] D. D. Hall and I. Mudawar. Critical heat flux (CHF) for water flow in tubes—I. Compilation and assessment of world CHF data. *International Journal of Heat and Mass Transfer*, 43(14):2573–2604, 2000. doi:10.1016/S0017-9310(99)00191-X.
 - [32] L. Heber. *Effizienzsteigerung von Nutzfahrzeugen durch den Einsatz von Thermoelektrischen Generatoren – Konzeption, Optimierung und Validierung*. Dissertation, Universität Stuttgart, 2023.
 - [33] S. Heim and L. Komogowski. Analysis of technology potential of a metal-based latent heat storage for backup process steam supply. *Chemie Ingenieur Technik*, 93(4):605–617, 2021. doi:10.1002/cite.202000169.
 - [34] O. Herbst. Kritische Siedezustände. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.
 - [35] J. E. Hesselgreaves. *Compact heat exchangers: Selection, design and operation*. Pergamon, 1st edition, 2001. ISBN 0080428398.
 - [36] C. Huber. *Phase Change Material in Battery Thermal Management Applications*. Dissertation, Technische Universität München, 2017.
 - [37] R. A. Huggins. *Energy storage*. Springer, New York and Heidelberg, 2010. ISBN 978-1-4419-1023-3. doi:10.1007/978-1-4419-1024-0.

- [38] N. I. Ibrahim, F. A. Al-Sulaiman, S. Rahman, B. S. Yilbas, and A. Z. Sahin. Heat transfer enhancement of phase change materials for thermal energy storage applications: A critical review. *Renewable and Sustainable Energy Reviews*, 74:26–50, 2017. doi:10.1016/j.rser.2017.01.169.
- [39] F. P. Incropera. *Fundamentals of heat and mass transfer*. John Wiley & Sons, Hoboken NJ, 6th edition, 2007. ISBN 0471457280.
- [40] N. R. Jankowski and F. P. McCluskey. A review of phase change materials for vehicle component thermal buffering. *Applied Energy*, 113:1525–1561, 2014. doi:10.1016/j.apenergy.2013.08.026.
- [41] Joint Committee for Guides in Metrology. JCGM 100:2008 Evaluation of measurement data - Guide to the expression of uncertainty in measurement. Available online: http://www.bipm.org/utls/common/documents/jcgm/JCGM_100_2008_E.pdf (accessed: Dec. 4, 2024).
- [42] S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors). *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 11th edition, 2013. ISBN 978-3-642-19980-6. doi:10.1007/978-3-642-19981-3.
- [43] S. G. Kandlikar, M. Shoji, and V. K. Dhir. *Handbook of phase change: Boiling and condensation*. Taylor & Francis, Philadelphia, Pa., 1999. ISBN 1560326344.
- [44] T. G. Karayiannis and M. M. Mahmoud. Flow boiling in microchannels: Fundamentals and applications. *Applied Thermal Engineering*, 115:1372–1397, 2017. doi:10.1016/j.applthermaleng.2016.08.063.
- [45] W. Kast. Druckverlust in durchströmten Rohren. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.
- [46] A. Katsaounis and M. Kind. Wärmeübergang beim Strömungssieden nach der Siedekrise. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.
- [47] W. M. Kays and A. L. London. *Compact heat exchangers*. Krieger Publishing Company, 3th edition, 1998. ISBN 9780070334182.
- [48] M. M. Kenisarin. High-temperature phase change materials for thermal energy storage. *Renewable and Sustainable Energy Reviews*, 14(3):955–970, 2010. doi:10.1016/j.rser.2009.11.011.
- [49] S.-M. Kim and I. Mudawar. Review of databases and predictive methods for heat transfer in condensing and boiling mini/micro-channel flows. *International Journal of Heat and Mass Transfer*, 77:627–652, 2014. doi:10.1016/j.ijheatmasstransfer.2014.05.036.

-
- [50] M. Kind. Strömungssieden unterkühlter Flüssigkeiten. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.
 - [51] M. Klein Altstedde, F. Rinderknecht, and H. Friedrich. Integrating phase-change materials into automotive thermoelectric generators. *Journal of Electronic Materials*, 43(6):2134–2140, 2014. doi:10.1007/s11664-014-2990-z.
 - [52] K. Knapp, K. Dubil, T. Wetzel, and B. Dietrich. Assesment of the heat transfer enhancement potential of periodic open cellular structures (POCS). *Proceedings of the 17th International Heat Transfer Conference, IHTC-17, Cape Town, South Africa*, 2023. doi:10.1615/IHTC17.370-20.
 - [53] M. Kober. *Thermoelektrische Generatoren zur Effizienzsteigerung von Kraftfahrzeugen – Methode zur ganzheitlichen Entwicklung und Bewertung*. Dissertation, Universität Stuttgart, 2020.
 - [54] L. Komogowski, K. Riedl, S. Beer, and H. Maier. Flexibilisierung von Kraftwerken durch thermische Energiespeicher auf Basis metallbasierter Phasenwechselmaterialien. *50. Kraftwerkstechnisches Kolloquium*, 2018.
 - [55] E. Korin, R. Reshef, D. Tshernichovsky, and E. Sher. Improving cold-start functioning of catalytic converters by using phase-change materials. *SAE Technical Paper Series*, 1998. doi:10.4271/980671.
 - [56] J. P. Kotzé. *Thermal energy storage in metallic phase change materials*. PhD thesis, Stellenbosch University, Stellenbosch, 2014.
 - [57] W. Kraft. *Konzeption, Auslegung, Demonstration und Bewertung neuartiger metallischer Latentwärmespeicher als Heizsystem für batterieelektrische Busse*. Dissertation, Universität Stuttgart, (to be submitted).
 - [58] W. Kraft and M. K. Altstedde. Use of metallic phase change materials (mPCM) for heat storage in electric- and hybrid vehicles. *6th Hybrid and Electric Vehicles Conference (HEVC 2016), London*, 2016. doi:10.1049/cp.2016.0974.
 - [59] W. Kraft, V. Stahl, and P. Vetter. Thermal storage using metallic phase change materials for bus heating – State of the art of electric buses and requirements for the storage system. *Energies*, 13(11):3023, 2020. doi:10.3390/en13113023.
 - [60] W. Kraft, P. Vetter, and V. Stahl. Wärmespeichervorrichtung. DE102020107464A1, 2021.
 - [61] H. Kuchling. *Taschenbuch der Physik*. Fachbuchverlag Leipzig im C. Hanser Verl., München and Wien, 18th edition, 2004. ISBN 3-446-22883-7.
 - [62] H. Kurtz. *Energieversorgungssysteme für die Raumfahrt–Vorlesungsmanuskript*. Institut für Raumfahrtssysteme, Universität Stuttgart, 2010.

- [63] T. Lanz, V. Jilg, W. Kraft, P. Vetter, and B. S. Volk. Experimental evaluation of a heat transport system for a high-temperature storage unit. *Proceedings of the 5th World Congress on Mechanical, Chemical, and Material Engineering (MCM'19)*, 2019. doi:10.11159/htff19.161.
- [64] E. Laurien and H. Oertel. *Numerische Strömungsmechanik*. Lehrbuch. Springer Vieweg, Wiesbaden, 5th edition, 2013. ISBN 978-3658031442.
- [65] J. G. Leidenfrost. On the fixation of water in diverse fire. *International Journal of Heat and Mass Transfer*, 9(11):1153–1166, 1966. doi:10.1016/0017-9310(66)90111-6.
- [66] C. Luo, P. Xie, G. Chen, L. Mao, L. Liu, L. Jin, Z. Cheng, J. Xu, and G. Qiao. Prototype design and experimental study of a metal alloy-based thermal energy storage system for heat supply in electric vehicles. *Journal of Energy Storage*, 51: 104393, 2022. doi:10.1016/j.est.2022.104393.
- [67] F. Ma and P. Zhang. Investigation on the performance of a high-temperature packed bed latent heat thermal energy storage system using Al-Si alloy. *Energy Conversion and Management*, 150:500–514, 2017. doi:10.1016/j.enconman.2017.08.040.
- [68] C. V. Madhusudana and L. S. Fletcher. Contact heat transfer - the last decade. *AIAA Journal*, 24(3):510–523, 1986. doi:10.2514/3.9298.
- [69] C. Maradiya, J. Vadher, and R. Agarwal. The heat transfer enhancement techniques and their thermal performance factor. *Beni-Suef University Journal of Basic and Applied Sciences*, 7(1):1–21, 2018. doi:10.1016/j.bjbas.2017.10.001.
- [70] R. Maurus. *Bestimmung des Blasenverhaltens beim unterkühlten Strömungssieden mit der digitalen Bildfolgenanalyse*. Dissertation, Technische Universität München, 2002.
- [71] A. A. Minea (editor). *Advances in new heat transfer fluids: From numerical to experimental techniques*. CRC Press Taylor & Francis Group, Boca Raton and London and New York, 2017. ISBN 978-1-4987-5185-8.
- [72] K. Mollenhauer (editor). *Handbuch Dieselmotoren*. Springer, Berlin, 2nd edition, 2002. ISBN 3540412395.
- [73] H. Nazir, M. Batool, F. J. Bolivar Osorio, M. Isaza-Ruiz, X. Xu, K. Vignarooban, P. Phelan, Inamuddin, and A. M. Kannan. Recent developments in phase change materials for energy storage applications: A review. *International Journal of Heat and Mass Transfer*, 129:491–523, 2019. doi:10.1016/j.ijheatmasstransfer.2018.09.126.
- [74] F. Nees. Wärmeübertragungssystem und Verfahren zum Betreiben eines Wärmeübertragungssystems. *DE102024124165A1*, 2026.

-
- [75] F. Nees. Wärmeübertragungssystem mit einer Wärmequelle und einer Wärmesenke und Verfahren zum Betreiben eines Wärmeübertragungssystems. *German Patent Application DE102025102423A1* (Due date of publication: Jul. 23, 2026), 2025.
- [76] F. Nees and Y. S. Pai. Conjugate heat transfer analysis of the transient thermal discharge of a metallic latent heat storage system. *Journal of Physics: Conf. Ser.*, 2766(1):012212, 2024. doi:10.1088/1742-6596/2766/1/012212.
- [77] F. Nees, W. Kraft, V. Stahl, and P. Vetter. Impact of transient storage temperature on heat transport system design for a thermal energy storage unit in vehicle applications. *15th International Conference on Energy Storage (ENERSTOCK), Virtual Conference*, 2021.
- [78] F. Nees, A. Katourtzidis, W. Kraft, V. Stahl, and P. Vetter. Experimental investigation of thermal discharge performance of a metallic latent thermal energy storage system. *Proceedings of the 8th World Congress on Mechanical, Chemical, and Material Engineering (MCM'22), Prague, Czech Republic*, 2022. doi:10.11159/htff22.139.
- [79] F. Nees, J. Feine, A. Katourtzidis, V. Stahl, and W. Kraft. Heat extraction characteristics of a small scale metallic latent heat thermal energy storage system. *Proceedings of the 17th International Heat Transfer Conference, IHTC-17, Cape Town, South Africa*, 2023. doi:10.1615/IHTC17.430-40.
- [80] F. Nees, L. Ignatzi, and T. Siefkes. Experimental demonstration of a multi-stage boiling heat extraction concept for high-temperature metallic phase change thermal energy storage in electric vehicles. *Journal of Energy Storage*, 120:116427, 2025. doi:10.1016/j.est.2025.116427.
- [81] P. M. Norris, M. C. Hastings, and W. J. Wepfer. An experimental investigation of liquid coolant heat transfer in a diesel engine. *SAE Technical Paper Series*, 1989.
- [82] S. Nukiyama. The maximum and minimum values of the heat Q transmitted from metal to boiling water under atmospheric pressure. *International Journal of Heat and Mass Transfer*, 9(12):1419–1433, 1966. doi:10.1016/0017-9310(66)90138-4.
- [83] C. Oshman, C. Hardin, J. Rea, M. L. Olsen, N. Siegel, G. Glatzmaier, P. Parilla, D. Ginley, and E. S. Toberer. Design of a thermosyphon-based thermal valve for controlled high-temperature heat extraction. *Applied Thermal Engineering*, 126: 1141–1147, 2017. doi:10.1016/j.applthermaleng.2017.01.038.
- [84] F. Ostermann. *Anwendungstechnologie Aluminium*. Springer Vieweg, Berlin, Heidelberg, 2014. ISBN 978-3-662-43807-7. doi:10.1007/978-3-662-43807-7.
- [85] H. J. Paus. *Physik in Experimenten und Beispielen*. Hanser, München, 2nd edition, 2002. ISBN 3446221352.
- [86] S. Pischinger, P. Genender, S. Klopstein, and D. Hemkemeyer. Aufgaben beim Thermomanagement von Hybrid- und Elektrofahrzeugen. *ATZ Automobiltechnische Zeitschrift*, 116(4):54–59, 2014. doi:10.1007/s35148-014-0382-6.

- [87] A. J. Rawson, C. Villada Vargas, W. Kraft, T. Bähr, H. A. Grübbel, F. Kargl, N. Navarrete Argiles, F. Nees, V. Stahl, and P. Vetter. Auszüge aus dem Projekt LatHe.Go. *DLR Projektbericht*, 2023.
- [88] A. J. Rawson, W. Kraft, T. Gläsel, and F. Kargl. Selection of compatible metallic phase change materials and containers for thermal storage applications. *Journal of Energy Storage*, 2020. doi:10.1016/j.est.2020.101927.
- [89] J. E. Rea, C. J. Oshman, A. Singh, J. Alleman, P. A. Parilla, C. L. Hardin, M. L. Olsen, N. P. Siegel, D. S. Ginley, and E. S. Toberer. Experimental demonstration of a dispatchable latent heat storage system with aluminum-silicon as a phase change material. *Applied Energy*, 230:1218–1229, 2018. doi:10.1016/j.apenergy.2018.09.017.
- [90] J. E. Rea, E. S. Toberer, and N. P. Siegel. Guidelines for phase change material selection based on a holistic system model. *Solar Energy Materials and Solar Cells*, 208:110422, 2020. doi:10.1016/j.solmat.2020.110422.
- [91] M. Reichler. *Theoretische Untersuchungen zur Kühlleistungssteigerung durch innovative Kühlsysteme für Brennstoffzellen-Elektrofahrzeuge*. Dissertation, Universität Stuttgart, 2009.
- [92] L. F. Richardson. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam. *Transactions of the Royal Society of London Ser. A*, (210):307–357, 1910. doi:10.1098/rsta.1911.0009.
- [93] L. F. Richardson. The deferred approach to the limit. *Transactions of the Royal Society of London Ser. A*, (226):299–361, 1927. doi:10.1098/rsta.1927.0008.
- [94] P. J. Roache. Verification of codes and calculations. *AIAA Journal*, 36(5):696–702, 1998. doi:10.2514/2.457.
- [95] M. M. Sarafraz, M. R. Safaei, M. Goodarzi, B. Yang, and M. Arjomandi. Heat transfer analysis of Ga-In-Sn in a compact heat exchanger equipped with straight micro-passages. *International Journal of Heat and Mass Transfer*, 139:675–684, 2019. doi:10.1016/j.ijheatmasstransfer.2019.05.057.
- [96] A. Schweizer. Zeta Werte von Rohrleitungskomponenten. Available online: https://www.schweizer-fn.de/zeta/start_zeta.php (accessed: May 5, 2025).
- [97] A. Schweizer. Wasser mit Frostschutzmittel in Abhängigkeit der Temperatur und Konzentration. Available online: https://www.schweizer-fn.de/stoff/kuehlwasser/kuehlw_start.php (accessed: Aug. 26, 2025).
- [98] P. J. Shamberger and N. M. Bruno. Review of metallic phase change materials for high heat flux transient thermal management applications. *Applied Energy*, 258: 113955, 2020. doi:10.1016/j.apenergy.2019.113955.

-
- [99] B. Snaith, P. W. O’Callaghan, and S. D. Probert. Interstitial materials for controlling thermal conductances across pressed metallic contacts. *Applied Energy*, 16(3): 175–191, 1984. doi:10.1016/0306-2619(84)90019-9.
- [100] J. Spurk and N. Aksel. *Strömungslehre*. Springer, Berlin, Heidelberg, 2019. ISBN 978-3-662-58763-8. doi:10.1007/978-3-662-58764-5.
- [101] V. Stahl, Y. Shi, W. Kraft, T. Lanz, P. Vetter, R. Jemmali, F. Kessel, and D. Koch. C/C–SiC component for metallic phase change materials. *International Journal of Applied Ceramic Technology*, 17(5):2040–2050, 2020. doi:10.1111/ijac.13570.
- [102] V. Stahl, A. J. Rawson, W. Kraft, P. Vetter, F. Nees, and F. Kargl. Thermal contact resistance: Implications for thermal energy storage systems. *15th International Conference on Energy Storage (ENERSTOCK), Virtual Conference*, 2021.
- [103] V. Stahl, W. Kraft, F. Nees, J. Feine, A. Katourtzidis, N. Navarrete, and C. Villada. Graphite based encapsulation concept for a high temperature metallic latent thermal energy storage system. *Proceedings of the 16th International Conference on Energy Storage (ENERSTOCK)*, 2024. doi:10.5281/zenodo.13790499.
- [104] V. Stahl, W. Kraft, and F. Lanzerath. Computational modelling of thermal loads on a heat storage for battery electric buses over service life. *Proceedings of the 38th International Electric Vehicle Symposium and Exhibition (EVS38) Goteborg, Sweden*, 2025.
- [105] P. Stephan. Wärme und verschiedene Arten der Wärmeübertragung. In S. Kabelac, M. Kind, H. Martin, D. Mewes, K. Schaber, and P. Stephan (editors), *VDI-Wärmeatlas*. Springer Vieweg, Berlin and Heidelberg, 2013. ISBN 978-3-642-19980-6.
- [106] M. Sterner and I. Stadler. *Energiespeicher: Bedarf, Technologien, Integration*. Springer Vieweg, Berlin, 2014. ISBN 978-3-642-37379-4. doi:10.1007/978-3-642-37380-0_10.
- [107] A. Thess, F. Trieb, A. Wörner, and S. Zunft. Herausforderung Wärmespeicher. *Physik Journal*, (14(2)):33–39, 2015.
- [108] A. Thielmann, A. Sauer, R. Isenmann, and M. Wietschel. *Technology roadmap energy storage for electric mobility 2030*. Fraunhofer Institute for Systems and Innovation Research ISI, Karlsruhe, 2013.
- [109] J. Tombrink, H. Jockenhöfer, and D. Bauer. Experimental investigation of a rotating drum heat exchanger for latent heat storage. *Applied Thermal Engineering*, 183: 116221, 2021. doi:10.1016/j.applthermaleng.2020.116221.
- [110] S. Trevisan, Y. Jemmal, R. Guedez, and B. Laumert. Packed bed thermal energy storage: A novel design methodology including quasi-dynamic boundary conditions and techno-economic optimization. *Journal of Energy Storage*, 36:102441, 2021. doi:10.1016/j.est.2021.102441.

- [111] H. Tschöke. *Die Elektrifizierung des Antriebsstrangs*. Springer, Wiesbaden, 2015. ISBN 978-3-658-04643-9. doi:10.1007/978-3-658-04644-6.
- [112] G. Urschitz, H. Walter, and J. Brier. Experimental investigation on bimetallic tube compositions for the use in latent heat thermal energy storage units. *Energy Conversion and Management*, 125:368–378, 2016. doi:10.1016/j.enconman.2016.05.054.
- [113] Valbruna. Valbruna alloy 286 / 1.4980, Werkstoffdatenblatt. Available online: https://www.valbruna.ch/wp-content/themes/valbruna-ch/assets/pdf/Werkstoff_Datenblatt_1.4980.pdf (accessed: Aug. 26, 2025).
- [114] VDM Metals. VDM Nickel 200 and 201, Material data sheet no. 1101, Revision 02, 2022. Available online: https://www.vdm-metals.com/fileadmin/user_upload/Downloads/Data_Sheets/Data_Sheet_VDM_Nickel_200_201.pdf (accessed: Aug. 26, 2025).
- [115] M. Vehviläinen, R. Lavikka, S. Rantala, M. Paakkinen, J. Laurila, and T. Vainio. Setting up and operating electric city buses in harsh winter conditions. *Applied Sciences*, 12(6):2762, 2022. doi:10.3390/app12062762.
- [116] Verein Deutscher Ingenieure. *VDI-Richtlinie 2221: Entwicklung technischer Produkte und Systeme - Modell der Produktentwicklung*. Beuth Verlag, Berlin, 2019.
- [117] Verein Deutscher Ingenieure. *VDI-Richtlinie 2206: Entwicklung mechatronischer und cyber-physischer Systeme*. Beuth Verlag, Berlin, 2021.
- [118] K. Vignarooban, X. Xu, A. Arvay, K. Hsu, and A. M. Kannan. Heat transfer fluids for concentrating solar power systems – A review. *Applied Energy*, 146:383–396, 2015. doi:10.1016/j.apenergy.2015.01.125.
- [119] M. C. Vlachou, J. S. Lioumbas, and T. D. Karapantsios. Heat transfer enhancement in boiling over modified surfaces: A critical review. *Interfacial Phenomena and Heat Transfer*, 3(4):341–367, 2015. doi:10.1615/InterfacPhenomHeatTransfer.2016014133.
- [120] J. Vogel, J. Felbinger, and M. Johnson. Natural convection in high temperature flat plate latent heat thermal energy storage systems. *Applied Energy*, 184:184–196, 2016. doi:10.1016/j.apenergy.2016.10.001.
- [121] W. Wagner. *Wärmeträgertechnik mit organischen Fluiden*. Vogel, Würzburg, 8th edition, 2011. ISBN 3834332046.
- [122] R. L. Webb. Performance evaluation criteria for use of enhanced heat transfer surfaces in heat exchanger design. *International Journal of Heat and Mass Transfer*, 24(4):715–726, 1981. doi:10.1016/0017-9310(81)90015-6.
- [123] R. Winterton. Where did the ditrus and boelter equation come from? *International Journal of Heat and Mass Transfer*, 41(4-5):809–810, 1998. doi:10.1016/S0017-9310(97)00177-4.

-
- [124] P. Xie, L. Jin, G. Qiao, C. Lin, C. Barreneche, and Y. Ding. Thermal energy storage for electric vehicles at low temperatures: Concepts, systems, devices and materials. *Renewable and Sustainable Energy Reviews*, 160:112263, 2022. doi:10.1016/j.rser.2022.112263.
- [125] Q. L. Yue, C. X. He, H. R. Jiang, M. C. Wu, and T. S. Zhao. A hybrid battery thermal management system for electric vehicles under dynamic working conditions. *International Journal of Heat and Mass Transfer*, 164:120528, 2021. doi:10.1016/j.ijheatmasstransfer.2020.120528.
- [126] B. Zhang, L. Cheng, Y. Jiang, and Z. Li. Numerical study on thermal contact resistance of metals. *Proceedings of the 17th International Heat Transfer Conference, IHTC-17, Cape Town, South Africa*, 2023. doi:10.1615/IHTC17.70-70.
- [127] Z. Zhang, W. Li, C. Zhang, and J. Chen. Climate control loads prediction of electric vehicles. *Applied Thermal Engineering*, 110:1183–1188, 2017. doi:10.1016/j.applthermaleng.2016.08.186.
- [128] Z. Zhang, D. Wang, C. Zhang, and J. Chen. Electric vehicle range extension strategies based on improved AC system in cold climate – A review. *International Journal of Refrigeration*, 88:141–150, 2018. doi:10.1016/j.ijrefrig.2017.12.018.
- [129] Z. Zou, Z. Zhu, Q. Zhang, and P. Yang. Numerical investigation on the performance of a new type of perforated-serrated fins for plate-fin heat exchangers in cryogenic helium systems. *Cryogenics*, 136:103761, 2023. doi:10.1016/j.cryogenics.2023.103761.

List of publications, patents, and awards

The peer-reviewed publications, patents, and awards listed below, authored in my capacity as corresponding author, pertain to the present PhD thesis:

Publications

F. Nees, L. Ignatzi, and T. Siefkes. Experimental demonstration of a multi-stage boiling heat extraction concept for high-temperature metallic phase change thermal energy storage in electric vehicles. *Journal of Energy Storage*, vol. 120, p. 116427, 2025. doi:10.1016/j.est.2025.116427.

(Chapters 3 and 5)

F. Nees and Y. S. Pai. Conjugate heat transfer analysis of the transient thermal discharge of a metallic latent heat storage system. *Journal of Physics: Conf. Ser.*, vol. 2766, no. 1, p. 12212, 2024. doi:10.1088/1742-6596/2766/1/012212.

(Chapter 4)

F. Nees, J. Feine, A. Katourtzidis, V. Stahl, and W. Kraft. Heat extraction characteristics of a small scale metallic latent heat thermal energy storage system. *Proceedings of the 17th International Heat Transfer Conference, IHTC-17, Cape Town, South Africa*, 2023. doi:10.1615/IHTC17.430-40.

(Chapter 4)

F. Nees, A. Katourtzidis, W. Kraft, V. Stahl, and P. Vetter. Experimental Investigation of Thermal Discharge Performance of a Metallic Latent Thermal Energy Storage System. *Proceedings of the 8th World Congress on Mechanical, Chemical, and Material Engineering (MCM'22), Prague, Czech Republic*, 2022. doi:10.11159/htff22.139.

(Chapter 4)

F. Nees, W. Kraft, V. Stahl, and P. Vetter. Impact of transient storage temperature on heat transport system design for a thermal energy storage unit in vehicle applications. *15th International Conference on Energy Storage (ENERSTOCK), Virtual Conference*, 2021.

(Chapter 3)

Patents

F. Nees. Wärmeübertragungssystem mit einer Wärmequelle und einer Wärmesenke und Verfahren zum Betreiben eines Wärmeübertragungssystems. *German patent application DE102025102423A1 (Due date of publication: Jul. 23, 2026)*, 2025.

(Chapter 3)

F. Nees. Wärmeübertragungssystem und Verfahren zum Betreiben eines Wärmeübertragungssystems. *DE102024124165A1*, 2026.

(Chapters 3 and 5)

Awards

Best Paper Award: F. Nees. *8th World Congress on Mechanical, Chemical, and Material Engineering (MCM'22)*, Prague, Czech Republic, 2022.