

Numerical Simulation of Radial Manifold Microchannel Heat Sinks for Next-Generation Thermal Management in Data Centers and High-Flux Electronics

by

Faramarz Kahbandeh

A dissertation submitted to the Graduate Faculty of
Auburn University
in partial fulfillment of the
requirements for the Degree of
Doctor of Philosophy

Auburn, Alabama
August 8, 2026

Keywords: manifold microchannel heat sink, data center cooling, conjugate heat transfer, numerical simulation

Copyright 2026 by Faramarz Kahbandeh

Approved by

Dr. Daniel Harris, Chair, Associate Professor of Mechanical Engineering
Dr. Mehmet Arik, Co-Chair, Associate Professor of Mechanical Engineering
Dr. Mark Hoffman, Associate Professor of Mechanical Engineering
Dr. Yanzhao Cao, Professor of Mathematics and Statistics
Dr. Bart Prorok, Professor of Materials Engineering

Abstract

The widespread adoption of emerging technologies, including artificial intelligence (AI), augmented reality (AR), fifth-generation (5G) mobile networks, the Internet of Things (IoT), and cryptocurrency-based computing, has led to an unprecedented surge in global data generation and processing demands. Modern data centers are rapidly evolving to accommodate next-generation microprocessors characterized by increased transistor density, higher clock speeds, and enhanced computational capability. While these advances significantly improve electrical performance and functional integration, they simultaneously intensify heat generation due to escalating power densities. In extreme cases, localized heat fluxes can approach or exceed 1000 W/cm^2 . Currently, data centers account for approximately 2% of global electricity consumption, with nearly 40% of that energy dedicated solely to thermal management infrastructure. These trends underscore an urgent need for transformative cooling technologies capable of sustaining continued growth in computational performance while minimizing energy consumption and environmental impact.

Although air cooling remains the dominant thermal management approach in contemporary data centers due to its simplicity, low cost, and established infrastructure, its thermophysical limitations render it increasingly inadequate for high-density server environments. The relatively low thermal conductivity, density, and heat capacity of air constrain its ability to dissipate large heat fluxes effectively. As microprocessor thermal design powers continue to rise and rack-level power densities increase, conventional air-based strategies struggle to maintain acceptable junction temperatures, resulting in higher operational costs, reduced reliability, and limited scalability. Consequently, a paradigm shift toward liquid cooling solutions has become not only advantageous but inevitable for next-generation data center architectures.

This dissertation begins with a comprehensive literature review examining the evolution of electronic cooling technologies, highlighting the transition from air-based convection systems to advanced liquid cooling strategies. The review identifies critical challenges associated with

high heat flux removal, non-uniform temperature distributions, pumping power penalties, and system-level integration within data center infrastructure. Particular emphasis is placed on the thermophysical superiority of liquid coolants, whose higher thermal conductivity, density, and specific heat capacity enable significantly improved heat transport compared to air. Direct-to-chip cold plates, microchannel heat sinks, jet impingement systems, and immersion cooling architectures are evaluated in terms of performance, reliability, and energy efficiency.

Building upon this foundation, the research presented in this dissertation focuses on the design, analysis, and optimization of advanced liquid cooling configurations tailored for high-power electronics. Advanced computational fluid dynamics (CFD) techniques were employed to analyze the flow and heat transfer characteristics of novel manifold geometries intended to address the increasing thermal demands of modern data centers, artificial intelligence hardware, and high-performance computing systems. Through a systematic combination of fluid flow analysis, conjugate heat transfer modeling, and geometric optimization, the study establishes design strategies for improving thermal performance while maintaining acceptable hydraulic losses.

The first technical phase of this work investigated a novel radial manifold microchannel heat sink concept by considering only the fluid domain. The objective was to evaluate the influence of manifold curvature, channel length, and nozzle placement on coolant flow behavior and thermal performance. Numerical simulations demonstrated that introducing curvature into the manifold geometry generates secondary flow structures that enhance coolant mixing and improve heat transfer characteristics compared to conventional straight manifold designs. The study further identified optimal nozzle locations and manifold lengths that maximize thermal performance while minimizing pressure drop penalties.

Building upon these findings, the second phase of the research incorporated realistic thermal boundary conditions and employed a conjugate heat transfer (CHT) framework to simultaneously model heat conduction within the solid substrate and heat convection within the coolant. Parametric studies were performed to investigate the effects of geometric parameters,

including nozzle dimensions, microchannel aspect ratio, and manifold length, on pressure drop, thermal resistance, heat transfer coefficient, and temperature uniformity. The results demonstrated that careful optimization of manifold and microchannel geometries can significantly improve cooling effectiveness while maintaining manageable pumping power requirements. Furthermore, the analysis provided valuable insight into the interactions between flow distribution, thermal boundary layer development, and localized heat transfer mechanisms within the heat sink.

The final phase of the dissertation extended the conjugate heat transfer analysis to tapered curved manifold configurations. Two distinct tapering strategies were proposed and systematically evaluated to determine how the location and implementation of tapering influence thermo-hydraulic performance. Additional reference configurations, including straight manifolds, reduced-height manifolds, and non-tapered curved manifolds, were investigated to isolate the effects of tapering, curvature, and manifold dimensions. The results revealed that the placement of tapering within the manifold system plays a critical role in determining flow distribution and cooling effectiveness. In particular, tapering strategies that preserved the inlet and outlet boundary conditions demonstrated the ability to reduce hydraulic losses without sacrificing thermal performance, while alternative tapering approaches exhibited a more conventional trade-off between pressure drop and heat transfer enhancement.

Throughout the dissertation, CFD simulations were utilized to resolve complex three-dimensional flow structures, pressure distributions, and temperature fields within the manifold microchannel heat sinks. Detailed analyses of velocity streamlines, pressure contours, thermal boundary layers, and heat flux distributions provided fundamental insight into the physical mechanisms governing coolant transport and heat removal. These numerical investigations enabled the identification of design configurations that achieve enhanced thermal performance while maintaining favorable hydraulic characteristics.

This work establishes a comprehensive numerical framework for the design and optimization of advanced manifold microchannel heat sinks for next-generation electronic cooling

applications. The findings demonstrate that appropriately engineered manifold geometries can significantly reduce thermal resistance, improve temperature uniformity, and enhance heat transfer performance while limiting hydraulic penalties. Furthermore, the results provide practical design guidelines for the development of high-performance cooling technologies capable of supporting the rapidly increasing thermal requirements of emerging computing platforms.

Future research should focus on the experimental validation of the proposed manifold microchannel heat sink configurations and the numerical findings presented in this dissertation. Experimental investigations would provide valuable verification of the predicted flow and heat transfer behavior while identifying practical considerations associated with fabrication, measurement uncertainty, and operational reliability. Additional research may also explore the integration of the proposed geometries with two-phase cooling technologies, alternative working fluids, and advanced manufacturing techniques. Such studies would further expand the applicability of the proposed designs and contribute to the development of highly efficient and sustainable thermal management solutions for future electronic and data center cooling systems.

Artificial Intelligence (AI) Use Disclosure Statement

In the preparation of this dissertation, the following Artificial Intelligence (AI) tools were used: ChatGPT (OpenAI) and QuillBot. These tools were used primarily to improve grammar, refine academic writing, paraphrase text, and enhance the clarity and organization of the manuscript. The author acknowledges full responsibility for the intellectual content of this dissertation and has ensured that all AI-assisted sections were thoroughly reviewed, verified, and revised for technical accuracy, scientific correctness, and appropriate academic style. All AI-generated content was critically evaluated before incorporation into the final manuscript to maintain the scholarly integrity of this research.

Digital Accessibility Disclosure Statement

In the preparation of this dissertation, the following software tools were used to support document preparation, data processing, numerical simulation, and figure generation: ANSYS Fluent, MATLAB, Python, Microsoft Word, Microsoft Excel, and Overleaf ($\text{\LaTeX}$). These applications were used to perform computational analyses, process simulation data, generate figures and plots, and prepare the final manuscript in accordance with the university's formatting requirements.

This document was typeset with LuaLaTeX using an accessible $\text{\LaTeX}$ class that produces a natively tagged PDF conforming to PDF/UA-2 and targeting WCAG 2.1 Level AA. Headings, lists, tables, and mathematics are tagged automatically by the compiler; every figure carries author-written alternative text, every data table declares its header row, and complex tables additionally carry a screen-reader summary. The following digital accessibility tools were used to verify the result: PAC (PDF Accessibility Checker) and Adobe Acrobat.

The author acknowledges full responsibility for the intellectual content of this dissertation and has made a good-faith effort to comply with applicable digital accessibility requirements in publishing, wherein the nature of the content does not significantly change in order to do so. Furthermore, all text, figures, tables, and other document elements were reviewed and revised, where appropriate, to improve accessibility while preserving the technical accuracy and scientific integrity of the research presented.

Acknowledgments

First and foremost, I would like to express my deepest and most heartfelt gratitude to my parents. Their unwavering love, sacrifice, and lifelong dedication to my education and personal growth have been the foundation of every achievement in my life. They devoted their lives to creating opportunities for me, always placing my dreams above their own. Words cannot adequately express the extent of my appreciation for their patience, encouragement, and belief in me.

I extend my profound gratitude to my beloved wife, Saba, and my sweet daughter, Delanna. Their love, understanding, and endless patience carried me through the most demanding years of this journey. I will never forget the countless evenings and weekends that should have belonged to them but were instead spent in my office running simulations and writing manuscripts. Not once did they complain about my absence at home. Their quiet strength, encouragement, and sacrifice made it possible for me to remain focused and persistent during challenging times. I truly could not have completed this stage of my life without their extraordinary understanding and unconditional support. This accomplishment is as much theirs as it is mine.

I am sincerely grateful to my advisor, Dr. Daniel Harris, for his invaluable mentorship, guidance, and continuous support throughout my doctoral studies. His insight, patience, and high academic standards shaped not only this dissertation but also my development as a researcher and engineer. I deeply appreciate his trust in my abilities and the opportunities he provided for intellectual growth and professional advancement.

I would also like to thank my committee members for their time, expertise, and thoughtful feedback. I am particularly grateful to Dr. Arik, Co-Chair of my committee, for his guidance and constructive suggestions. I extend my appreciation to Dr. Hoffman and Dr. Cao, of the Department of Mechanical Engineering and the Department of Mathematics and Statistics, respectively, for their valuable perspectives and support.

My sincere thanks go to the faculty members of the Department of Mechanical Engineering

at Auburn University who have contributed to my academic development. I am grateful to Dr. Flowers, Dr. Suhling, Dr. Khodadadi, and Dr. Ayasoufi for their instruction, encouragement, and professional example.

I would also like to express my appreciation to Dr. Mohammad Azarifar for his generous support and guidance. His willingness to share his experience and knowledge, especially as a graduate of the Department of Mechanical Engineering at Auburn University, was instrumental during critical stages of my research.

Finally, I am thankful to Auburn University and the Department of Mechanical Engineering for providing an environment that fosters innovation, perseverance, and academic excellence. This journey has been both challenging and transformative, and I am deeply grateful to everyone who played a role in helping me reach this milestone.

Table of Contents

Abstract	2
Artificial Intelligence (AI) Use Disclosure Statement	6
Digital Accessibility Disclosure Statement	7
Acknowledgments	8
List of Tables	13
List of Figures	14
List of Abbreviations	17
List of Symbols	20
Chapter 1 From Air to Liquid: The Evolution of Thermal Management for High-Density Electronics	23
1.1 Literature Review: Data Center Cooling Approaches	23
1.1.1 Motivation and Scope	23
1.1.2 Air Cooling: Architectures and Heat Removal Pathways	24
1.1.3 Liquid Cooling: Motivation and System Integration	25
1.1.4 Indirect Water-Cooled Data Centers	26
1.1.5 Direct-to-Chip Cooling with Cold Plates	27
1.1.6 Immersion Cooling	30
1.1.7 Comparative Discussion and Emerging Trends	32
1.1.8 Summary	33
1.2 Emerging Cold Plate Designs	34
1.2.1 Refinement of the Manifold Layout Design	35
1.2.2 Optimization of Key Geometric Parameters	37
1.3 Research Gaps	41
Chapter 2 Numerical Investigation of the Novel Radial Manifold Design Considering the Fluid Domain	44

2.1	Introduction	44
2.2	Physically Based Model	44
2.3	Numerical Modeling and Boundary Conditions	45
2.4	Mesh Independence Study and Validation	51
2.5	Results and Discussion	53
2.5.1	Effects of Nozzle Position	54
2.5.2	Effects of Channel Length	59
2.6	Conclusions	63
Chapter 3	Conjugate Heat Transfer Analysis and Geometric Optimization of a Radial Manifold Microchannel Heat Sink	66
3.1	Introduction	66
3.2	Simulation Method and Data Reduction	66
3.2.1	Physical Model and Three-Dimensional Numerical Simulation Method	66
3.2.2	Computational Domains and Boundary Conditions	69
3.2.3	Grid Independence Test and Validation	71
3.2.4	Data Reduction	76
3.3	Results and Discussion	77
3.3.1	Effects of the Nozzle-to-Inlet Manifold Length Ratio	80
3.3.2	Effects of the Microchannel Aspect Ratio	83
3.3.3	Effects of Manifold Length	87
3.4	Conclusions	93
Chapter 4	Numerical Investigation of Tapered Curved Radial Manifold Microchannel Heat Sinks Using Conjugate Heat Transfer	96
4.1	Introduction	96
4.2	Simulation Method and Data Reduction	96
4.2.1	Physical Model and Three-Dimensional Numerical Simulation Method	96
4.2.2	Boundary Conditions	101

4.2.3	Data Reduction	102
4.3	Results and Discussion	102
4.4	Conclusions	122
References		124
Appendix 1	Full-Scale Comparison of the Traditional and Radial Manifold Heat Sinks .	132
Appendix 2	Tapered Configurations Geometry and Flow Patterns in the Manifold Section	135

List of Tables

1.1	Studies optimizing the key geometric parameters of MMCHS	42
2.1	Geometric details of the unit manifold model	48
3.1	Thermophysical properties of the materials	68
3.2	Key dimensions of the curved manifold microchannel heat sink	72
3.3	Mesh independence study	74
4.1	Geometric features of the Group A and Group B tapered models	100

List of Figures

1.1	Air cooling configuration	25
1.2	RDHx liquid cooling architecture	28
1.3	Some cold plate configurations	29
1.4	Single-phase immersion cooling	31
1.5	Two-phase immersion cooling	33
1.6	Traditional architecture of an MMCHS	35
1.7	Computational domain of the MMCHS studied by Luo et al.	38
1.8	Computational domain of the MMCHS studied by Pan et al.	40
1.9	Schematic of the ribbed MMCHS studied by Chen et al.	41
2.1	Baseline and proposed heat sink geometries	46
2.2	Comparison between curved and straight manifolds	46
2.3	Unit manifold model geometry and boundary conditions	48
2.4	Mesh independence study	53
2.5	Validation against experimental data	54
2.6	Thermal resistance and pressure drop for the three manifold lengths	55
2.7	Effect of manifold channel length	56
2.8	Streamlines and velocity contours for the three nozzle positions	57
2.9	Temperature contours and two-dimensional streamlines	59
2.10	Wall heat flux and streamlines for the three manifold lengths	60
2.11	Thermal resistance as a function of pumping power	62
2.12	Thermal performance index for all manifold models	64
3.1	Computational domain for the fluid and solid regions	70
3.2	Local mesh of the computational domain	72
3.3	Validation against the experimental data of Drummond et al.	75

3.4	Pressure drop and thermal resistance for all models	79
3.5	Location of the defined cross-sectional planes	81
3.6	Static temperature contours for varying nozzle ratio	82
3.7	Pressure contours and surface streamlines for varying aspect ratio	85
3.8	Friction factor and heat transfer coefficient for varying aspect ratio	87
3.9	Base temperature contours for three manifold lengths	88
3.10	Flow streamlines in the middle manifold channel	89
3.11	Heat transfer coefficient for three manifold lengths	91
3.12	Mean absolute temperature deviation for three manifold lengths	93
4.1	Conventional and proposed tapered heat sink designs	97
4.2	Geometry of the tapered curved manifold	100
4.3	Direct comparison between Groups A and B	103
4.4	Group B configurations and their reference models	104
4.5	Group A configurations and their reference models	107
4.6	MATD for both group designs	110
4.7	Base temperature distribution for all configurations	114
4.8	Heat transfer coefficient for both group designs	115
4.9	Velocity contours of the Group A designs	117
4.10	Velocity contours of the Group B designs	119
4.11	Velocity streamlines of both group designs	121
A1.1	Flow pattern and structure of the TMM and R-RMM heat sinks	132
A1.2	Thermo-hydraulic performance of the RMM and TMM heat sinks	133
A1.3	Velocity contours at the center cross-section	134
A2.1	Structure of the A1 configuration	135
A2.2	Manifold height comparison for the A1 configuration	136
A2.3	Structure of the A2 configuration	136

A2.4	Manifold height comparison for the A2 configuration	136
A2.5	Structure of the A3 configuration	137
A2.6	Manifold height comparison for the A3 configuration	137
A2.7	Structure of the A4 configuration	138
A2.8	Manifold height comparison for the A4 configuration	138
A2.9	Structure of the A5 configuration	139
A2.10	Manifold height comparison for the A5 configuration	139
A2.11	Structure of the B1 configuration	140
A2.12	Manifold height comparison for the B1 configuration	140
A2.13	Structure of the B2 configuration	141
A2.14	Manifold height comparison for the B2 configuration	141
A2.15	Structure of the B3 configuration	142
A2.16	Manifold height comparison for the B3 configuration	142
A2.17	Structure of the B4 configuration	143
A2.18	Manifold height comparison for the B4 configuration	143
A2.19	Structure of the B5 configuration	144
A2.20	Manifold height comparison for the B5 configuration	144

List of Abbreviations

AI	Artificial Intelligence
AR	Augmented Reality
CDU	Coolant Distribution Unit
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
COP	Coefficient of Performance
CPU	Central Processing Unit
CRAC	Computer Room Air Conditioner
CRAH	Computer Room Air Handler
DI	Deionized (water)
DLC	Direct Liquid Cooling
FVM	Finite Volume Method
FWS	Facility Water System
GPU	Graphics Processing Unit
HBM	High-Bandwidth Memory
HPC	High-Performance Computing
IoT	Internet of Things
IT	Information Technology

MATD Mean Absolute Temperature Deviation

MCHS Microchannel Heat Sink

MMCHS Manifold Microchannel Heat Sink

PUE Power Usage Effectiveness

RDHx Rear-Door Heat Exchanger

RMM Radial Manifold Microchannel

RMMCHS Radial Manifold Microchannel Heat Sink

TCS Technology Cooling System

TMM Traditional Manifold Microchannel

TPI Thermal Performance Index

The nine radial manifold configurations studied in Chapter 2 are named by nozzle position (left, middle, or right edge) followed by manifold length (short, middle, or long):

L RMM Left-edge nozzle radial manifold microchannel

M RMM Middle nozzle radial manifold microchannel

R RMM Right-edge nozzle radial manifold microchannel

LS Left-edge nozzle in the shortest manifold studied

LM Left-edge nozzle in the middle-length manifold studied

LL Left-edge nozzle in the longest manifold studied

MS Middle nozzle in the shortest manifold studied

MM Middle nozzle in the middle-length manifold studied

- ML** Middle nozzle in the longest manifold studied
- RS** Right-edge nozzle in the shortest manifold studied
- RM** Right-edge nozzle in the middle-length manifold studied
- RL** Right-edge nozzle in the longest manifold studied

List of Symbols

A_{hs}	Heat sink base surface area (m ²)
C_p	Specific heat capacity (J/(kg · K))
$C_{p,f}$	Specific heat capacity of the fluid domain (J/(kg · K))
D_h	Hydraulic diameter (m)
f	Darcy friction factor
h	Convective heat transfer coefficient (W/(m ² · K))
h_b	Substrate height (mm)
h_c, h_{mc}	Microchannel height (mm)
h_{in}	Manifold inlet height (mm)
h_m	Manifold height (mm)
h_n	Slot nozzle height (mm)
h_u	Unit model height (mm)
k	Thermal conductivity (W/(m · K))
k_f	Thermal conductivity of the fluid domain (W/(m · K))
k_s	Thermal conductivity of the solid domain (W/(m · K))
l_{in}	Manifold inlet length (mm)
l_n	Slot nozzle length (mm)
l_{out}	Manifold outlet length (mm)

l_u	Unit model length (mm)
l_{ws}	Wall spacing length (mm)
L_{ave}	Average microchannel length (m)
$\dot{m}$	Mass flow rate (kg/s)
N	Total number of manifold channels
P	Pressure (Pa)
P_{in}	Inlet pressure (Pa)
P_{out}	Outlet pressure (Pa)
P_p	Pumping power (W)
q''	Heat flux (W/m ²)
$\bar{q}$	Average heat removed by the heat sink (W)
$\bar{q}_C$	Average heat transfer rate of the curved manifold configuration (W)
$\bar{q}_S$	Average heat transfer rate of the straight manifold configuration (W)
r	Radial distance (mm)
Re	Reynolds number
R_t, R_{tot}	Total thermal resistance (K/W)
T	Temperature (K)
$T_{base,ave}$	Average temperature of the heat sink base (K)
$T_{base,max}$	Maximum temperature of the heat sink base (K)
$T_{base,min}$	Minimum temperature of the heat sink base (K)

T_f	Temperature in the fluid domain (K)
$T_{f,ave}$	Average fluid temperature (K)
$T_{hs,max}$	Maximum heat sink base temperature (K)
T_{in}	Working fluid inlet temperature (K)
T_s	Temperature in the solid domain (K)
t_c	Microchannel wall thickness (mm)
t_m	Manifold wall thickness (mm)
$\mathbf{u}$	Velocity vector field (m/s)
u_c	Average fluid velocity at the microchannel inlet (m/s)
$\dot{V}$	Volumetric flow rate (m ³ /s)
w_c	Wall spacing length (mm)
α	Ratio of slot nozzle length to manifold inlet length
β	Aspect ratio of microchannel height to width
ΔP	Pressure drop across the domain (Pa)
ΔP_C	Pressure drop of the curved manifold configuration (Pa)
ΔP_S	Pressure drop of the straight manifold configuration (Pa)
μ	Dynamic viscosity (kg/(m · s))
ρ	Density (kg/m ³)
ρ_f	Fluid domain density (kg/m ³)

Chapter 1

From Air to Liquid: The Evolution of Thermal Management for High-Density Electronics

1.1 Literature Review: Data Center Cooling Approaches

1.1.1 Motivation and Scope

The continued advancement of high-performance computing is compelling a fundamental transformation in data center infrastructure, most notably in their thermal management architectures. Rapid growth in artificial intelligence (AI), high-performance computing (HPC), cloud services, and edge applications has increased both server utilization and component-level power density. Data center operation with cooling systems that consume up to 40% of total facility energy is increasingly considered unsustainable [1]. As a result, thermal management has become a primary constraint on data center performance, reliability, and energy efficiency. The pressing need for energy-efficient cooling technologies is not only driven by operational considerations but also plays a crucial role in mitigating the environmental footprint of the data industry [2]. Historically, most facilities relied on air-based cooling because of its simplicity and established supply chains; however, rising rack heat loads and localized hot spots increasingly push air systems toward practical limits. The thermophysical properties of air are no longer sufficient to meet the thermal demands of modern data centers. In comparison, the heat removal capacity of air is roughly 37% that of water [3]. Industry surveys show that direct liquid cooling adoption is growing but remains gradual, with many operators considering partial deployment as density rises [4], [5].

This literature review categorizes data center cooling strategies into four major classes: (i) air cooling, (ii) liquid cooling at the facility or room level, (iii) direct-to-chip cooling (cold plates), and (iv) immersion cooling. For each category, the review summarizes governing heat-transfer mechanisms, typical architectures, operational advantages, and key limitations that motivate continued research and development. It should be noted that each of these subcategories

represents a broad and extensively studied research area [6], [7].

1.1.2 Air Cooling: Architectures and Heat Removal Pathways

Air cooling removes heat primarily through forced convection at server heat sinks, followed by the transport of heated air through the room to cooling units such as air-to-liquid coils or direct-expansion refrigeration systems. Figure 1.1 illustrates conventional air-cooled data centers in which Computer Room Air Handlers (CRAHs) or Computer Room Air Conditioners (CRACs) supply chilled air through raised-floor plenums and perforated tiles, although overhead supply configurations are also used in some facilities. Server fans draw this cooled air from the cold aisle and circulate it across electronic components, often equipped with heat sinks to enhance heat dissipation from high-power devices. The heated air is then expelled through the rear of the server chassis into the hot aisle, establishing a structured airflow pattern that reduces mixing between supply and exhaust air streams. The warm air is subsequently returned to the cooling units, where the absorbed heat is rejected through external heat rejection systems such as condensers, dry coolers, cooling towers, or evaporative cooling systems. This recirculating airflow forms the foundation of traditional air-based thermal management in data centers. The effectiveness of air cooling depends strongly on (i) minimizing bypass air and recirculation, (ii) delivering adequate mass flow to the highest-power servers, and (iii) maintaining acceptable inlet temperatures and humidity within recommended envelopes [8], [9].

A major research and industry focus has been hot-aisle or cold-aisle containment, which reduces mixing losses and stabilizes server inlet temperatures [8]. That study proposes two strategies to address inefficient airflow and the resulting formation of local hotspots in servers: (a) regulating airflow distribution and (b) reducing the distance traveled by the supplied cold air. In-row coolers are also employed as “air-side boosters” that intercept heat close to the racks, reducing the burden on room-level cooling and enabling higher return air temperatures. These strategies can improve energy efficiency by allowing higher chilled-water temperatures and greater economizer hours, but they remain fundamentally limited by air-side heat capacity and

volumetric flow constraints at high rack densities.

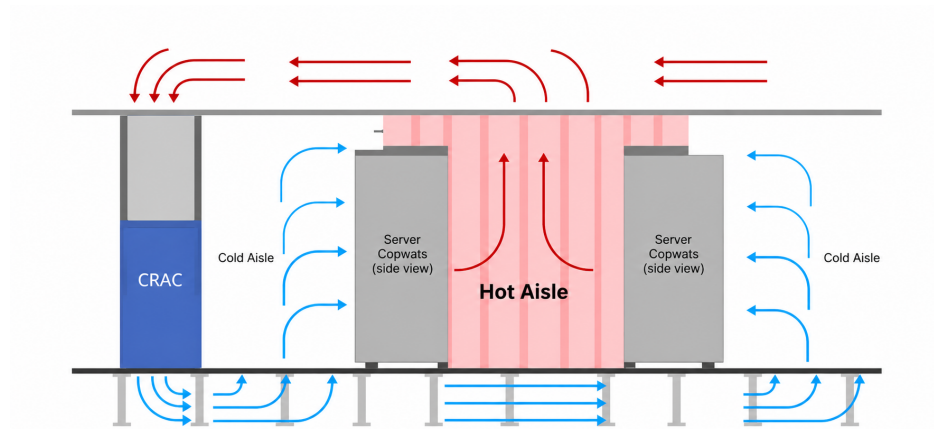


Figure 1.1: Air cooling configuration reproduced from [10].

Air is a weak heat transport medium compared with liquids; as rack power and chip heat flux rise, required airflow rates increase substantially, creating challenges in fan power, acoustic limits, flow distribution, and floor-space utilization. In addition, localized hot spots on heterogeneous compute boards (for example, GPUs, high-bandwidth memory, and voltage regulators) can exceed the capability of air-cooled heat sinks even when average rack power appears manageable. These constraints are a primary driver behind the industry shift toward direct liquid cooling for high-density deployments [6], [9].

1.1.3 Liquid Cooling: Motivation and System Integration

The limitations of air cooling in modern data centers underscore the need for liquid cooling. When processor thermal design power exceeds approximately 250–280 W and rack-level heat loads surpass 100 kW, conventional air-cooling approaches become insufficient to maintain recommended operating temperatures [11]. Liquid cooling of microelectronic components began to gain significant attention in the mid-1980s, when companies such as IBM, Honeywell, Sperry-Univac, Control Data, and Hitachi introduced indirectly water-cooled mainframe computers [12]. Since then, liquid cooling has been widely adopted in large computer systems and has evolved into four primary categories:

- indirect water-cooled systems that act as a transition from air cooling;
- direct liquid-cooled systems (cold plates);
- open-bath single-phase immersion cooling;
- sealed two-phase immersion cooling utilizing engineered dielectric liquids.

1.1.4 Indirect Water-Cooled Data Centers

One of the most practical pathways for transitioning existing data centers toward water cooling is the integration of rear-door heat exchangers (RDHx), shown in Figure 1.2. In 2005, IBM introduced the first water-cooled RDHx system, sometimes referred to as a “sidecar”, demonstrating notable energy savings. RDHx units are typically mounted on the rear of server racks and operate similarly to radiators, using internal coils to transfer heat from the exhaust air to circulating water. These systems are commonly classified as passive RDHx; however, some designs incorporate suction fans to compensate for pressure losses, forming active RDHx configurations. In such systems, chilled water, conditioned above the dew point by coolant distribution units (CDUs), removes heat from the warm exhaust air generated by IT equipment. A key advantage of RDHx technology is that, under ideal conditions, it can eliminate the need for hot-aisle containment or return plenums. In practice, RDHx units absorb a portion of rack-level heat, thereby reducing the load on the primary refrigeration system. Additionally, RDHx systems help mitigate airflow issues such as hot-air recirculation and cold-air bypass by rapidly capturing exhaust heat and limiting mixing with ambient air. This can lead to lower inlet air temperatures and also presents opportunities for waste heat recovery [11].

While most legacy facilities already employ chilled water for CRAH coils, modern designs increasingly pursue warmer liquid loops (including warm-water cooling) to reduce compressor work and improve opportunities for heat recovery [7].

Simon et al. [13] evaluated the feasibility of RDHx in high-capacity data centers using computational fluid dynamics (CFD) simulations and found that RDHx systems can significantly

reduce cooling loads on CRAC and CRAH units while maintaining acceptable server inlet temperatures. Their results showed that passive RDHx systems may experience airflow leakage, which can be mitigated by using active RDHx with auxiliary fans, ultimately achieving overall energy savings. Kumar et al. [14] demonstrated the practical deployment of RDHx combined with direct evaporative cooling in a Tier 3 data center in Bangalore, achieving a power usage effectiveness (PUE) as low as 1.06 under favorable conditions, although air leakage and operational adjustments increased the annualized PUE to 1.38. Meanwhile, Gao et al. [15] investigated both steady-state and transient performance of RDHx systems using a three-dimensional room-level model and showed that integrating RDHx with conventional air cooling significantly improves thermal stability, reduces temperature fluctuations during load variations or CRAC failures, and enhances the overall reliability of data center cooling operations.

Coolant distribution units are widely reported as the interface between facility water systems and IT-level liquid loops. CDUs provide pumping, pressure control, filtration, temperature regulation, and often isolation via heat exchangers. Practical deployment requires attention to redundancy and fault response because loss of liquid flow can lead to rapid IT over-temperature excursions, that is, reduced thermal ride-through compared with air-cooled rooms [16]. Industry guidance emphasizes reliability engineering, instrumentation, and leak management as central design requirements for mainstream adoption [17].

1.1.5 Direct-to-Chip Cooling with Cold Plates

Direct-to-chip cooling replaces (or supplements) air heat sinks on high-power components with liquid-cooled cold plates attached to processors, accelerators, and sometimes memory modules. Heat is transferred from the silicon package through a thermal interface material into the cold plate, and then convected into the circulating coolant. Direct-to-chip cooling is increasingly positioned as a transformative approach for high-density computing, enabling improved energy efficiency and supporting higher rack power densities [2], [6]. Direct liquid cooling involves a complex heat exchange loop consisting of pumps, piping, manifolds, fluid

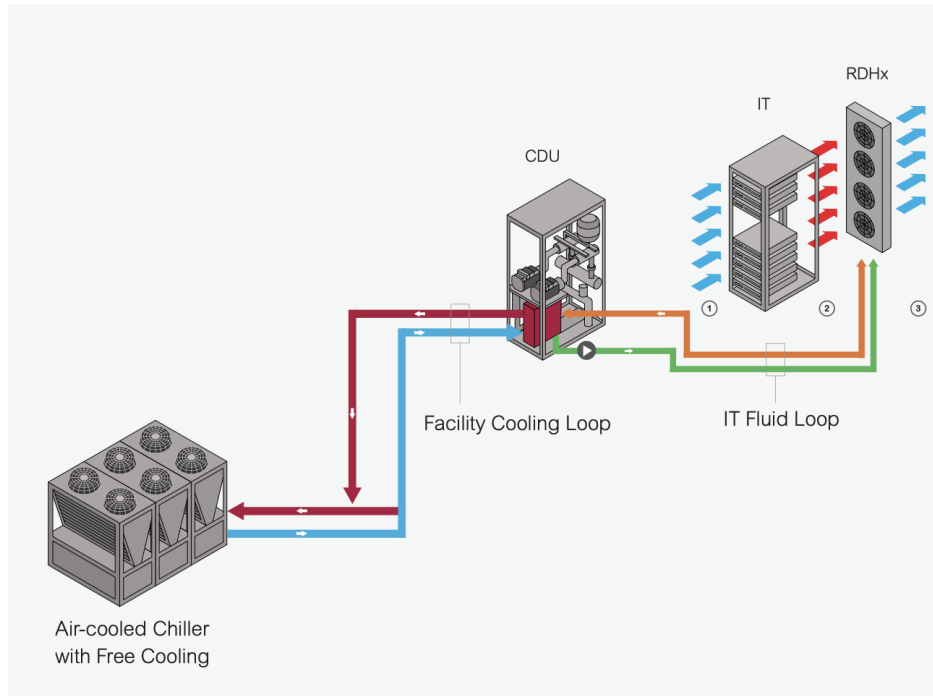


Figure 1.2: RDHx liquid cooling architecture reproduced from [18].

connections, and terminators, with the CDU serving as the central control element that enables heat transfer between the technology cooling system (TCS) and the facility water system (FWS) [11]. Standardization efforts such as those within the Open Compute Project aim to reduce fragmentation and enable interoperable ecosystems for cold plates and liquid-cooled server designs [19].

Cold plates are commonly designed with microchannels, where reducing the hydraulic diameter enhances heat transfer by better utilizing the thermal capacity of the TCS. However, smaller channel dimensions also lead to increased pressure drop between the inlet and outlet. Consequently, optimizing the trade-off between improved thermal performance and hydraulic losses is a critical design consideration. Additionally, finer microchannel geometries necessitate effective filtration of the TCS to prevent clogging [11].

Cold plate designs incorporating jet impingement cooling have gained increasing attention due to their high heat transfer potential. Early designs typically introduced coolant from the top of the cold plate to promote impingement on a heat spreader integrated with various mi-

crochannel geometries, such as rectangular, serpentine, wavy, spiral, zigzag, or pin-fin structures. However, these initial configurations often produced low-velocity, unstructured jets, resulting in non-uniform cooling performance. To address this limitation, manifold-based jet impingement designs were developed, significantly improving flow distribution and thermal performance. In such systems, coolant enters through a manifold and is evenly distributed to an array of nozzles, which direct high-velocity jets onto the heat spreader surface. The fluid then travels a short distance through interconnected microchannels or pin-fin structures before being collected by an outlet manifold. This approach enhances heat transfer by leveraging the high local coefficients of jet impingement while simultaneously reducing pressure drop due to the shortened flow path within the microchannels, making manifold heat sinks an effective solution for high-performance cooling applications [20], [21], [22].

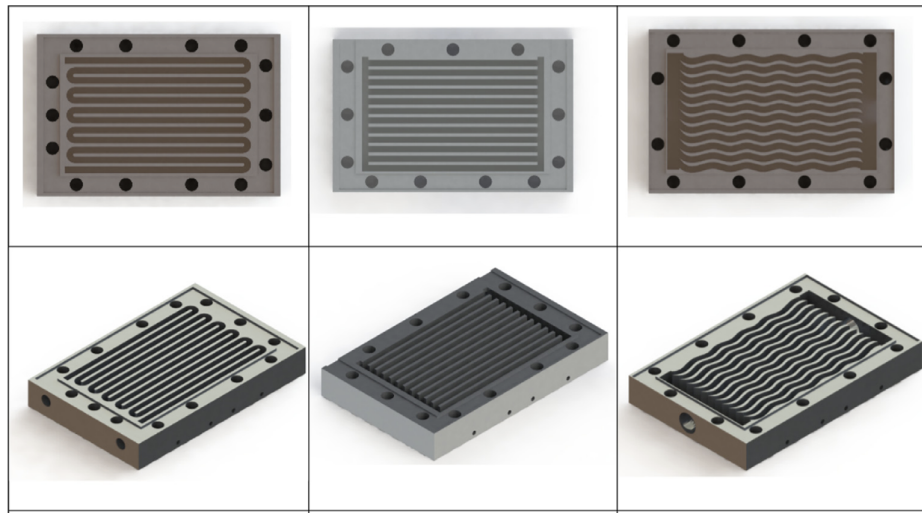


Figure 1.3: Some cold plate configurations reproduced from [23].

Single-phase direct-to-chip systems use a liquid coolant (commonly water or water-glycol in non-dielectric loops) that remains liquid throughout operation. Their advantages include mature components, simpler controls, and relatively straightforward reliability engineering. Thermal performance depends on cold plate microstructure (channels, jets, pin-fins), flow rate, pressure drop, and the quality of the thermal interface. System-level benefits include reduced reliance on high-volume airflow and potential increases in coolant supply temperature, which

can improve facility-side energy performance [7], [24].

Two-phase direct-to-chip cooling leverages boiling within the cold plate or evaporator to achieve high effective heat transfer coefficients and more uniform temperature control at high heat flux. Recent studies report hybrid deployments that integrate two-phase direct-to-chip loops into otherwise air-cooled server infrastructures, indicating practical retrofit pathways for incremental adoption [5]. However, two-phase systems introduce additional considerations: pressure control, flow stability, vapor management, condenser design, and long-term reliability under transient workloads. The literature increasingly frames two-phase direct-to-chip cooling as a candidate for extreme heat flux and next-generation accelerators, while acknowledging higher system complexity [4].

Across both single-phase and two-phase cold plate systems, recurring implementation challenges include leak risk management, connector reliability, serviceability (quick disconnects, maintenance workflows), materials compatibility (corrosion, erosion), and monitoring of flow, pressure, temperature, and coolant quality. Industry guidelines highlight the importance of design validation, integration logistics, and standardized interfaces to accelerate adoption [17], [19].

1.1.6 Immersion Cooling

Direct liquid cooling in data centers is widely recognized for its high heat removal capability achieved through cold plates and evaporators attached to microprocessors. However, it introduces several challenges, including leakage risks, complex maintenance, detailed system management, and reliance on CDUs for pumping and flow distribution through manifolds. Additionally, auxiliary air cooling is still required for peripheral components and condensation control, which can increase overall PUE. Immersion cooling extends this concept by fully submerging servers in dielectric liquids, allowing all components to be in direct contact with the coolant. This approach has also been explored in other high-density applications such as solar photovoltaics, batteries, and power transformers. The concept of dielectric cooling dates back

to early developments by IBM in the 1960s, with immersion cooling introduced shortly thereafter. Recent studies highlight immersion cooling as a highly efficient solution, offering benefits such as energy savings, reduced infrastructure footprint, simplified installation and maintenance, and lower environmental impact due to reduced noise and emissions. Furthermore, two-phase immersion cooling provides stable and uniform temperature control, eliminating hotspots and minimizing performance degradation. Owing to these advantages, the immersion cooling market is expected to grow significantly in the coming years [25], [26], [27], [28], [29], [30], [31].

In single-phase immersion cooling, shown in Figure 1.4, the coolant is engineered with a sufficiently high boiling temperature to avoid phase change under operating conditions. These systems typically operate at atmospheric pressure, enabling simpler maintenance and easier component replacement. However, additional measures are required to minimize fluid losses and remove entrained air and moisture. The presence of contaminants or dissolved gases can alter the thermophysical properties of the coolant; therefore, immersed IT components must be free of coatings or materials that could degrade fluid quality. Although single-phase systems are generally more cost-effective than two-phase alternatives, they still require filtration and fluid management due to gradual losses from diffusive evaporation [32], [33]. The United States Department of Energy recognizes immersion as a direct-liquid-cooling class and discusses its role in high-performance computing and energy-efficient facility design [7].

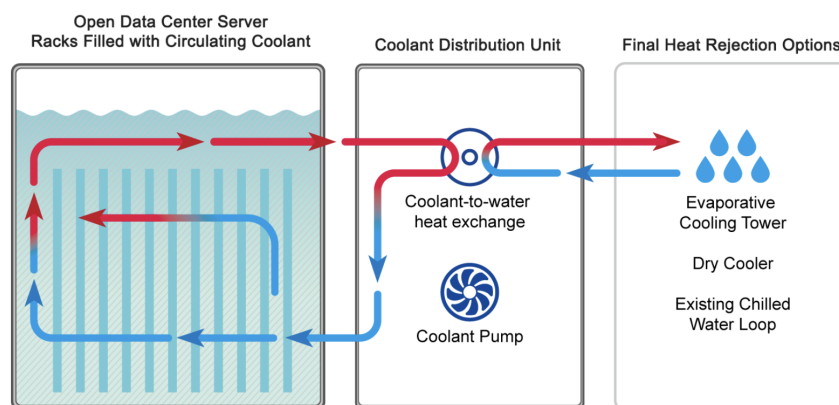


Figure 1.4: Single-phase immersion cooling architecture reproduced from [34].

Two-phase immersion, shown in Figure 1.5, provides enhanced heat transfer coefficients and improved temperature uniformity across heated surfaces. These benefits arise from isothermal nucleation and the high latent heat of vaporization associated with phase change. However, this approach introduces challenges related to flow instabilities, which can cause pressure and temperature fluctuations, flow reversal, and potentially lead to surface superheating and burnout [35], [36]. Two-phase immersion relies on boiling of a dielectric working fluid at component surfaces. Vapor rises and condenses on a heat exchanger surface, returning as liquid, enabling high heat flux handling with potentially simpler pumping requirements. In practice, two-phase immersion introduces additional challenges including fluid management, condensation capacity, containment, and fluid cost considerations. Recent reviews summarize benefits and barriers, emphasizing manufacturability, reliability, and compatibility as central research needs [37].

While immersion cooling can offer compelling thermal performance and mechanical simplicity through fan reduction, operational concerns remain: hardware compatibility (materials, coatings, seals), contamination control, maintenance procedures, component replacement workflows, and supplier qualification for immersion-ready IT equipment. As immersion deployments expand, the literature increasingly highlights the need for standardized test methods, validated component compatibility databases, and clear environmental, health, and safety protocols [7], [38].

1.1.7 Comparative Discussion and Emerging Trends

Cooling technology selection is increasingly determined by a combination of rack density targets, retrofit constraints, water and energy availability, and reliability requirements. Air cooling remains dominant for conventional racks and mixed-density deployments, but direct-to-chip liquid cooling is widely regarded as the most practical pathway for scaling high-density computing within existing data center paradigms [7], [9]. Immersion cooling provides an alternative route for very high densities and for facilities designed around immersion operations; however, ecosystem maturity and service workflows remain key adoption barriers [7], [38].

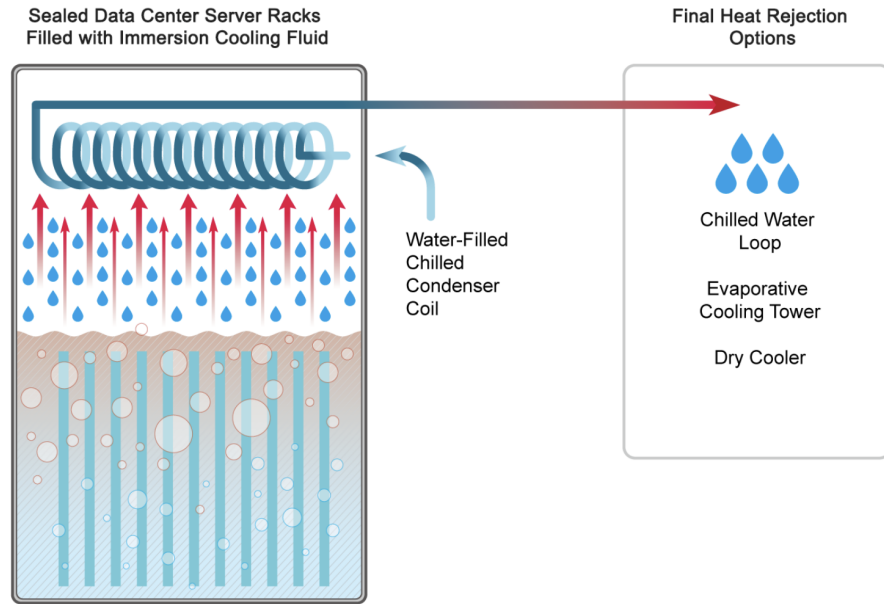


Figure 1.5: Two-phase immersion cooling architecture reproduced from [34].

A notable trend is hybridization: many operators pursue partial liquid cooling, cooling the highest-power components with cold plates while retaining air cooling for remaining board-level heat loads and facility compatibility. The literature also points to increasing interest in warmer coolant supply temperatures and waste heat reuse enabled by liquid cooling architectures, which can reduce cooling energy and improve total site sustainability [24]. In parallel, ongoing standardization initiatives and guidelines aim to reduce deployment friction by addressing interoperability, validation, and logistics [17].

1.1.8 Summary

The literature indicates a clear transition from purely air-cooled facilities toward liquid-enabled cooling ecosystems. Air cooling remains relevant due to simplicity and entrenched infrastructure, but is increasingly constrained by high power densities. Facility-level liquid loops and CDUs provide the backbone for scalable liquid deployments. Direct-to-chip cold plates, both single-phase and two-phase, offer a targeted, retrofit-friendly pathway to cool the dominant heat sources (CPUs and GPUs) with improved efficiency. Immersion cooling provides a high-

performance alternative with distinct operational trade-offs. Collectively, these approaches form a continuum of solutions that can be tailored to specific density, efficiency, and reliability objectives, motivating continued research in heat transfer enhancement, flow distribution, reliability engineering, and system-level optimization.

1.2 Emerging Cold Plate Designs

In 1981, Tuckerman and Pease [39] first proposed microchannels as an advanced liquid cooling solution after they achieved a very high heat flux of 790 W/cm^2 from an integral water-cooled microchannel with a temperature rise of 71°C . The heat sink achieved a high heat transfer coefficient of $11.1 \text{ W}/(\text{cm}^2 \cdot \text{K})$, albeit with a substantial pressure drop of 214 kPa. Since then, several researchers have carried out comprehensive and detailed investigations in the field to improve the thermal and flow characteristics of microchannel heat sinks. Despite the excellent heat transfer performance of traditional microchannel heat sinks, some shortcomings limit their application in engineering practice [40]. Some promising techniques to attain better results include diverging and converging channels [40], [41], wavy channels [42], [43], multi-layer channels [44], [45], pin-fins [46], [47], nanofluid coolants [48], [49], and manifold microchannel heat sink systems [50], [51]. It is worth noting that among all of these, implementing a manifold system is one of the approaches that has the most remarkable effect [40]. Manifold microchannel systems, which contain an additional layer above traditional microchannels, decrease the excessive pressure drop brought on by long flow length, facilitating more efficient working fluid distribution into microchannel sections. In 1991, Harpole and Eninger [52] were the first to develop a numerical model of a manifold microchannel heat sink (MMCHS).

The MMCHS has been extensively studied using both experimental methods and numerical simulations for a range of applications beyond electronics cooling, including solar cells [53], [54], electric vehicles [55], [56], and battery pack thermal management [57]. Although certain studies have explored the use of various refrigerants [58], [59] or nanofluids as working fluid [60], [61], [62], [63] to enhance the efficiency of MMCHS, most performance improvement efforts

can broadly be divided into two main categories: refinement of the manifold layout design, and optimization of key geometric parameters such as microchannel depth, width, and manifold structure [40].

1.2.1 Refinement of the Manifold Layout Design

Escher et al. [64], [65], whose design is shown in Figure 1.6, numerically studied the thermal and hydrodynamic performance of a unit cell of the heat sink with respect to the independent design parameters, and then experimentally investigated six different MMCHS prototypes with different numbers of manifolds and varying channel widths of the heat transfer structure. They reported a total thermal resistance of $0.09 \text{ K} \cdot \text{cm}^2/\text{W}$ with a pressure drop of 0.22 bar for their best design, which used 12.5 manifold systems and $25 \mu\text{m}$ wide microchannels.

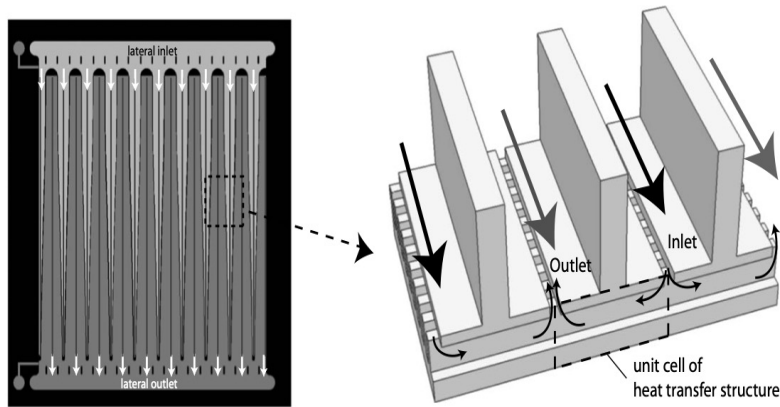


Figure 1.6: Traditional architecture of an MMCHS reproduced from [64].

Boteler et al. [66] conducted a numerical investigation of a single-phase MMCHS, modeling the full manifold length along with multiple microchannels. Their results demonstrated a 97% reduction in overall pressure drop compared to a conventional straight MCHS, under equivalent thermal performance conditions. Li et al. [67] introduced a tapered-type MMCHS aimed at enhancing temperature uniformity and minimizing hot spot temperatures. Using CFD simulations, they investigated the impact of geometric parameters by defining several dimensionless ratios.

Their findings revealed that the width inclination ratio of the inlet manifold had a more significant effect on coolant flow distribution than the height inclination ratio.

Tang et al. [40], [68] conducted a numerical investigation on the influence of two key dimensionless parameters in a diverging and converging MMCHS. The results indicated that combining a moderately diverging manifold with a converging channel configuration yielded optimal heat transfer efficiency at a fixed pumping power. Specifically, the diverging manifold reduced thermal resistance by 11.1% compared to a uniform baseline, while the converging channel design achieved an even greater reduction of 13.5%. In a subsequent experimental study building upon their numerical findings, the authors validated their design using ammonia as the working fluid. Under single-phase flow conditions, both thermal resistance and pressure drop remained relatively constant and showed minimal sensitivity to changes in heat flux. However, once the heat flux exceeded the boiling threshold, significant improvements in heat transfer were observed, accompanied by a notable increase in pressure drop.

Pu et al. [69] performed single-phase numerical simulations to assess the performance of two diverging and converging MMCHS designs: one oriented horizontally along the z -axis and the other vertically along the y -axis. The study revealed substantial improvements in both flow and temperature distribution uniformity. The horizontal design improved flow uniformity by an average of 81.44%, while the vertical configuration achieved an even greater enhancement of 83.65%. Similarly, temperature uniformity improved by 57.71% and 62.13% for the horizontal and vertical layouts, respectively. However, despite these significant gains, the system exhibited high pressure drops, indicating a need for further optimization to balance uniformity improvements with manageable hydraulic losses.

Luo et al. [70] conducted a numerical study to investigate the effect of different inlet-to-outlet width ratios (γ) on subcooled flow boiling in an MMCHS; their computational domain is shown in Figure 1.7. Their results indicated that a ratio of $\gamma \leq 1$ is not advisable, as it leads to relatively high thermal resistance. Conversely, while ratios greater than 2 improved thermal performance, they significantly increased pressure drop. Based on their analysis, an optimal

manifold width ratio lies between 1 and 2, offering a favorable balance for heat transfer.

Jung et al. [71] experimentally evaluated the thermal performance of an embedded MMCHS using single-phase water as the working fluid, over a range of volumetric flow rates and applied heat fluxes at the heat sink base. Their results demonstrated a heat removal capacity of 250 W/cm^2 with a pressure drop of just 2.6 kPa and a maximum base temperature of 90°C at a flow rate of 0.1 L/min. Wei et al. [72] numerically investigated a complete silicon-based MMCHS with a large footprint. They concluded that while the plenum in the complete cooler model effectively distributes fluid almost evenly among the manifold channels, each of these manifold channels exhibits non-uniform fluid distribution across the microchannels. Gilmore et al. [73] proposed two compact and adaptable manifold configurations using two-dimensional topology optimization, employing proportionally sized obstructions to regulate flow distribution at the inlet of each manifold channel. Both designs demonstrated excellent flow uniformity, with the mini-baffle configuration exhibiting a lower pressure drop. The authors noted that further investigation is required to determine the optimal obstruction geometry for practical applications. Kong et al. [74] experimentally studied the thermal and fluid behavior of an MMCHS featuring tapered inlets and outlets, which was fabricated using additive manufacturing techniques. The investigation spanned flow rates ranging from 100 to 400 g/min. Their results showed a significant decrease in thermal resistance, reaching as low as 0.21 K/W at the highest flow rate of 400 g/min. However, this enhancement was accompanied by a trade-off, as the pressure drop increased by approximately eleven times compared to the lowest evaluated flow rate.

1.2.2 Optimization of Key Geometric Parameters

Wei et al. [75] conducted experimental studies to evaluate the impact of varying microchannel aspect ratios in an MMCHS. Their findings indicated that increasing the aspect ratio leads to the formation of larger stagnant regions within the channels, and that an optimal aspect ratio of 3 yields the most favorable performance. Chen et al. [76] carried out an extensive numerical study to evaluate the flow and heat transfer performance of an MMCHS with five

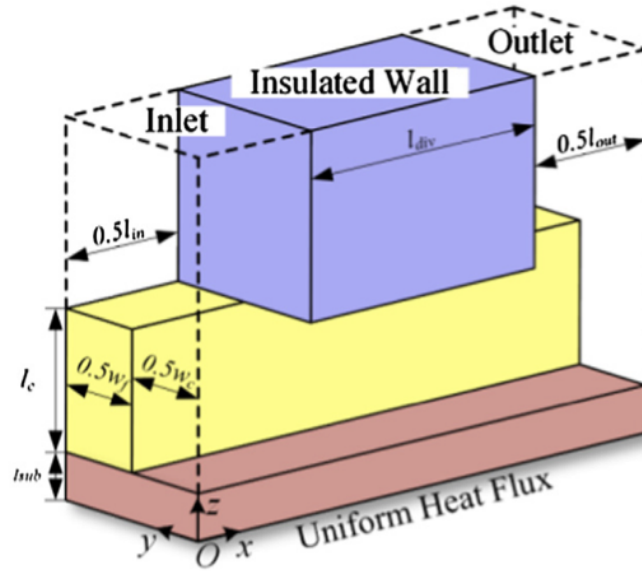


Figure 1.7: Computational domain of the MMCHS reported by [70].

different microchannel configurations. That research emphasizes the importance of geometric optimization in enhancing the thermal management effectiveness of MMCHS, particularly for high-power electronic systems. The findings show that staggered fins can reduce thermal resistance compared to straight fins, though this comes at the cost of increased pressure drop. Notably, a staggered pin-fin configuration was found to reduce pressure drop by 35.4% and thermal resistance by 21.6% compared to traditional parallel-fin layouts, demonstrating a promising strategy for improving the overall efficiency of MMCHS.

Kang et al. [77] conducted numerical simulations to analyze the heat transfer behavior of MMCHS systems featuring different interface configurations. While their study did not include pressure drop data, the results indicated that eliminating internal interfaces or enhancing their thermal conductivity could reduce thermal resistance by as much as 9.0%. Additionally, these modifications led to notable improvements in temperature uniformity across both the bottom wall and the entire heat sink, underscoring the critical role of interface optimization in enhancing heat sink performance.

Gilmore et al. [21], [53] numerically investigated a new design of MMCHS by eliminating

ineffective nozzle constrictions beneath the slot nozzles at the manifold channels. They reported a substantial improvement in pressure drop of 57% in their best study model. They numerically evaluated their design using two separate models for the microchannel and the manifold. They subsequently proposed a topology-optimized MMCHS design featuring intricate pin structures and constrictions aimed at enhancing jet impingement and mitigating streamwise fluid heating. Although these complex geometries are not compatible with current microfabrication technologies, the designs demonstrated notable performance improvements, achieving a 17% reduction in pressure drop and a 22.4% decrease in thermal resistance.

Pan et al. [78], [79], [80] investigated how the thermophysical properties of both the working fluid and the heat sink influence the optimal aspect ratio for performance; their computational domain is shown in Figure 1.8. The findings indicate that the optimal aspect ratio increases with higher thermal conductivity of the channel walls and greater dynamic viscosity of the working fluid, whereas it decreases with an increase in the specific heat capacity and thermal conductivity of the working fluid in rectangular-shaped MMCHS. They further extended their simulations to evaluate the effects of pin-fin and staggered pin-fin configurations within a manifold system, incorporating two different types of manifold dividers. The results showed that the staggered configuration offered superior heat transfer performance compared to the conventional rectangular manifold design. However, this thermal improvement came with a higher pressure drop, highlighting a trade-off between enhanced heat transfer and increased hydraulic resistance. These findings underscore the importance of balancing thermal efficiency against pressure drop when designing MMCHS.

Yang et al. [81] implemented bifurcated structures within an MMCHS to enhance its thermal performance. They numerically analyzed the influence of various geometric parameters, including bifurcation length, width, and the aspect ratio of the manifold channels. Their results indicated that the total thermal resistance of the heat sink could be reduced by up to 43.2% in the optimized configuration, particularly when the inlet-to-outlet width ratio of the manifold channels was set to 1/3. However, the authors also acknowledged that the hydraulic performance

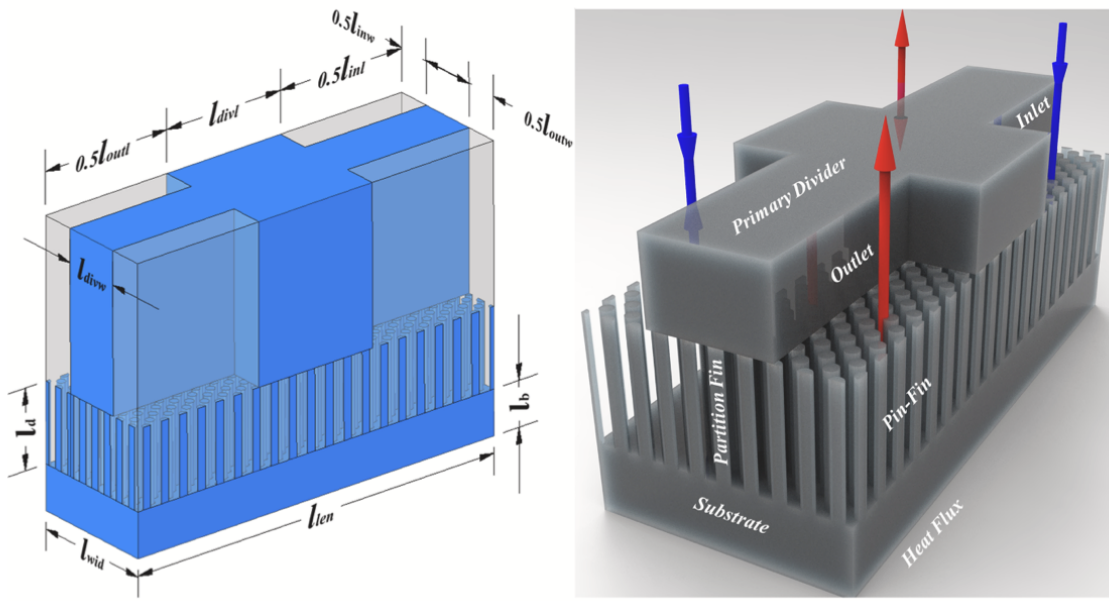


Figure 1.8: Computational domain of the MMCHS reported by [78], [79], [80].

may deteriorate due to the increased flow resistance introduced by longer bifurcations.

Kim et al. [55] numerically investigated the impact of various geometric parameters within both the manifold and microchannel sections of an MMCHS designed for thermal management in electric vehicles. Their findings revealed that variations in microchannel height had a negligible influence on pumping power. However, reducing the microchannel height led to a 5.2% improvement in thermal resistance under the higher volumetric flow rate conditions considered in their study. Yang et al. [82] proposed a novel microchannel configuration for MMCHS by incorporating secondary oblique channels and examined the influence of various geometric parameters through numerical simulations. Their results showed that the optimized design could reduce the pressure drop by 1.91% and simultaneously lower the thermal resistance by 19.15% compared to a conventional manifold microchannel heat sink at a Reynolds number of 295.

Chen et al. [83] conducted a numerical investigation to evaluate the impact of various rib shapes on the performance of MMCHS; their geometry is shown in Figure 1.9. Their results showed that the inclusion of ribs led to a significant increase in pressure drop, ranging from 55.7%

to 111.4% compared to a smooth-walled MCHS. Among the rib geometries studied, rectangular, triangular, and trapezoidal ribs caused the most pronounced rise in pressure drop, whereas the elliptical rib configuration resulted in the lowest pressure penalty under identical Reynolds number conditions. Table 1.1 summarizes representative studies reported in the literature along with their key findings and conclusions.

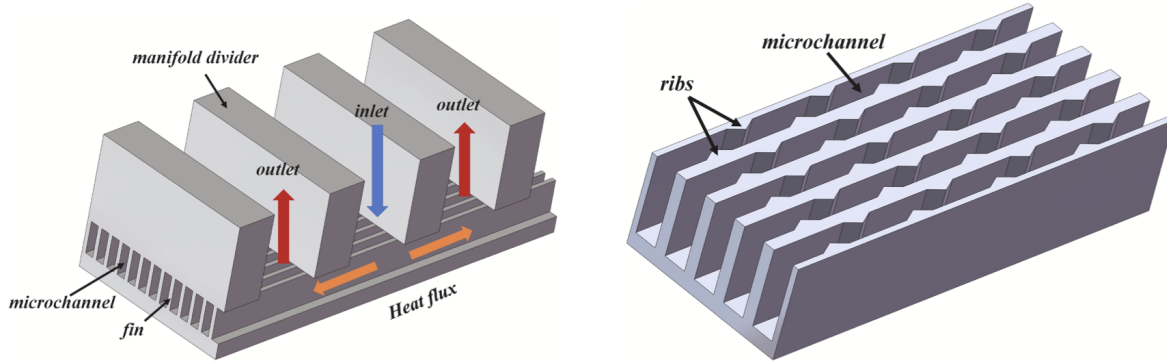


Figure 1.9: Schematic of the ribbed MMCHS reported by [83].

1.3 Research Gaps

Although substantial advancements have been achieved in liquid cooling technologies and MMCHS, several challenges remain, including improving coolant flow distribution, reducing pumping power requirements, and maintaining high thermal performance under increasingly demanding operating conditions. The literature demonstrates that MMCHS provide significant thermo-hydraulic advantages over conventional MCHS; however, most existing designs are based on traditional configurations employing straight manifold channels with rectangular cross-sections, occasionally incorporating tapered, converging, or diverging geometries. In these designs, the coolant generally follows a straight flow path through the manifold, limiting opportunities for enhanced mixing and flow redistribution.

To address these limitations, the present work introduces a novel radial manifold microchannel heat sink (RMMCHS), in which curved manifold channels are integrated above the cold plate. The curvature of the flow path generates centrifugal forces and secondary flow struc-

Table 1.1: Summary of studies that optimized the key geometric parameters of MMCHS. A dash indicates that the quantity was not reported.

Author	Investigation	Geometric feature	Thermal improvement (%)	Hydraulic improvement (%)
Chen et al. [76]	Numerical	Staggered pin-fin	21.6	35.4
Kang et al. [77]	Numerical	Different interface configurations	9.0	—
Gilmore et al. [21], [53]	Numerical	Eliminating ineffective nozzle walls	22.4	17.0
Yang et al. [81]	Numerical	Different aspect ratios	43.2	—
Kim et al. [55]	Numerical	Different microchannel heights	5.2	—
Yang et al. [82]	Numerical	Secondary oblique channels	19.5	1.91
Chen et al. [83]	Numerical	Various rib shapes	25.1	−111.4

tures that promote fluid mixing and potentially improve heat transfer performance compared to conventional MMCHS designs [84]. Numerical simulations were conducted to comprehensively investigate the flow behavior and thermo-hydraulic performance of the proposed design. Particular emphasis was placed on evaluating the effects of the slot nozzle-to-manifold inlet length ratio, the microchannel aspect ratio, and the manifold channel length on pressure drop, thermal resistance, and temperature uniformity, with the objective of developing an advanced cooling strategy suitable for next-generation high-power electronic and data center applications.

Chapter 2

Numerical Investigation of the Novel Radial Manifold Design Considering the Fluid Domain

2.1 Introduction

This chapter investigates the performance of the proposed radial manifold microchannel heat sink in comparison with the traditional straight manifold microchannel configuration by considering fluid flow behavior exclusively within the fluid domain. The analysis focuses on understanding the influence of manifold curvature on flow distribution, pressure characteristics, pumping power, and heat transfer enhancement mechanisms. In particular, the effects of varying manifold channel lengths and different nozzle placements along the inlet channel are systematically examined. By evaluating these geometric variations, the study aims to reveal how curvature-induced secondary flows, flow development, and jet impingement behavior contribute to the thermo-hydraulic performance of the radial manifold design relative to the traditional straight manifold configuration.

2.2 Physically Based Model

Figure 2.1(a) illustrates the schematic configuration of the traditional Z-type trapezoidal manifold microchannel heat sink (TMM). Building upon this baseline design, a novel configuration is introduced by incorporating curvature into the manifold sections, as depicted in Figure 2.1(b). The primary objective of this modification is to enhance flow distribution and heat transfer performance through the induction of secondary flow structures. To systematically evaluate the thermal performance of the proposed radial manifold microchannel heat sink (RMM), numerical simulations were conducted for both the traditional straight manifold and the newly developed curved manifold configurations under identical geometric dimensions, operating conditions, and boundary constraints.

The results, presented in Figure 2.2(a), indicate that the curved manifold configuration achieves approximately 10% lower thermal resistance at higher flow rates compared to the

straight manifold design. This reduction in thermal resistance highlights the effectiveness of the curved geometry in improving heat removal capabilities. Moreover, Figure 2.2(b) provides further insight into the underlying flow physics by illustrating the velocity streamlines within both configurations. In the curved manifold, the flow exhibits pronounced swirling behavior in both the inlet and outlet channels, whereas the straight manifold is characterized by predominantly linear streamlines with minimal secondary motion. The presence of these swirl-induced secondary flows enhances mixing, disrupts thermal boundary layer development, and promotes more uniform coolant distribution across the microchannels.

To further investigate the influence of manifold geometry on thermo-hydraulic performance, the numerical study was extended to include multiple design variations, comprising three distinct manifold lengths and three different nozzle positions for each configuration. This parametric analysis enables a comprehensive assessment of the combined effects of geometric curvature, flow distribution, and jet impingement location on overall heat sink performance. The results provide deeper insight into the interplay between manifold design parameters and cooling efficiency, offering valuable guidelines for the optimization of advanced microchannel heat sink architectures.

2.3 Numerical Modeling and Boundary Conditions

Figure 2.3 presents the computational domain used in this study and highlights the key geometric parameters that define the novelty in the heat sink configuration. A detailed summary of the principal dimensions associated with the curved manifold design, along with their corresponding equivalent values, is provided in Table 2.1. To systematically investigate the influence of manifold geometry on thermo-hydraulic performance, a parametric variation was introduced by modifying the radius of the inner curved channel, which represents one of the outlet manifolds. Specifically, the radius was varied over a range from 1 mm to 7.75 mm, enabling a comprehensive assessment of the effect of manifold length on flow distribution and heat transfer characteristics.

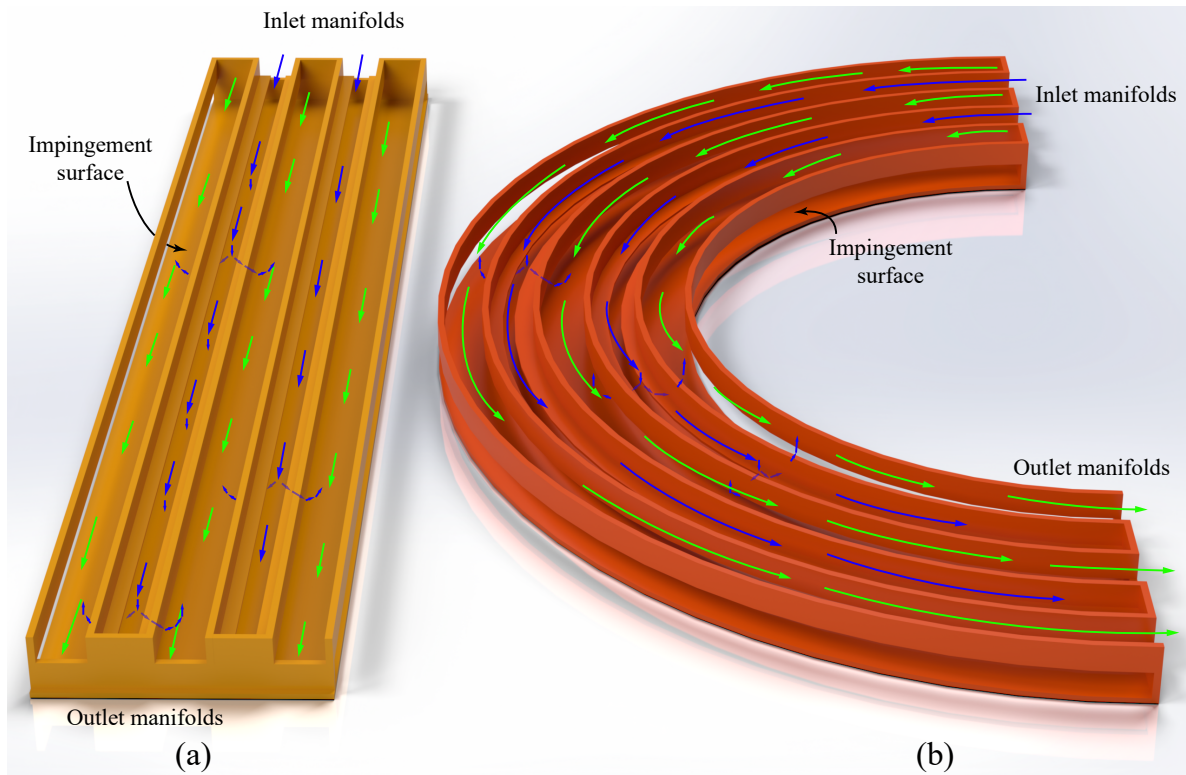


Figure 2.1: (a) Z-type TMM heat sink and (b) the novel RMM heat sink design.

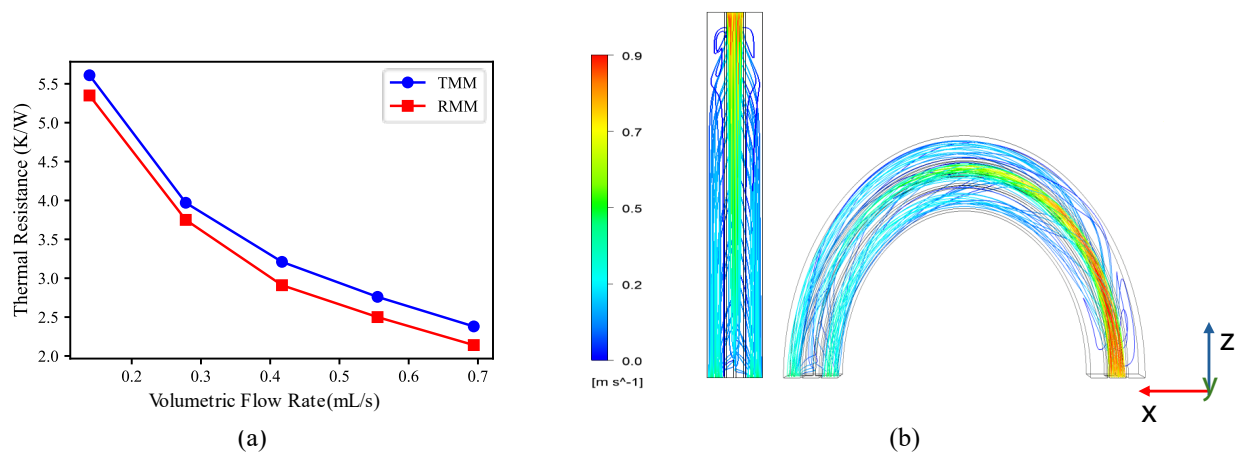


Figure 2.2: Comparison between curved and straight manifolds: (a) variation of thermal resistance with volumetric flow rate and (b) typical velocity streamlines in the manifold channels.

In addition to geometric variations in manifold length, the spatial positioning of the inlet nozzle was also examined as a critical design parameter. The nozzle was located at three distinct positions along the inlet channel, namely the left edge, the middle, and the right edge, to evaluate its impact on coolant distribution, jet impingement behavior, and overall thermal performance. This approach allows for a detailed understanding of how flow entry conditions influence the effectiveness of heat removal within the microchannel network.

Deionized (DI) water was selected as the working fluid for all simulations due to its favorable thermophysical properties, including high specific heat capacity and thermal conductivity, as well as its widespread use in electronic cooling applications. A constant temperature boundary condition of 80 °C was imposed at the base of the heat sink, representing the maximum allowable operating temperature for the electronic chip considered in this study [22], [85]. This boundary condition ensures that the simulation reflects realistic thermal constraints encountered in practical applications. All remaining surfaces, illustrated in gray in Figure 2.3, were treated as adiabatic to eliminate external heat losses, while no-slip boundary conditions were applied to all boundary interfaces to accurately capture viscous effects and realistic flow behavior within the system.

These modeling assumptions and parametric variations establish a robust framework for evaluating the combined effects of manifold geometry, nozzle placement, and flow characteristics on the overall performance of the proposed heat sink design.

The Reynolds number, which characterizes the flow regime within the manifold channels, is defined as

$$\text{Re} = \frac{4 \dot{m}}{\pi D_h \mu}, \quad (2.1)$$

where $\dot{m}$ represents the mass flow rate within an individual manifold channel, μ denotes the dynamic viscosity of the working fluid, and D_h is the hydraulic diameter of the channel. The

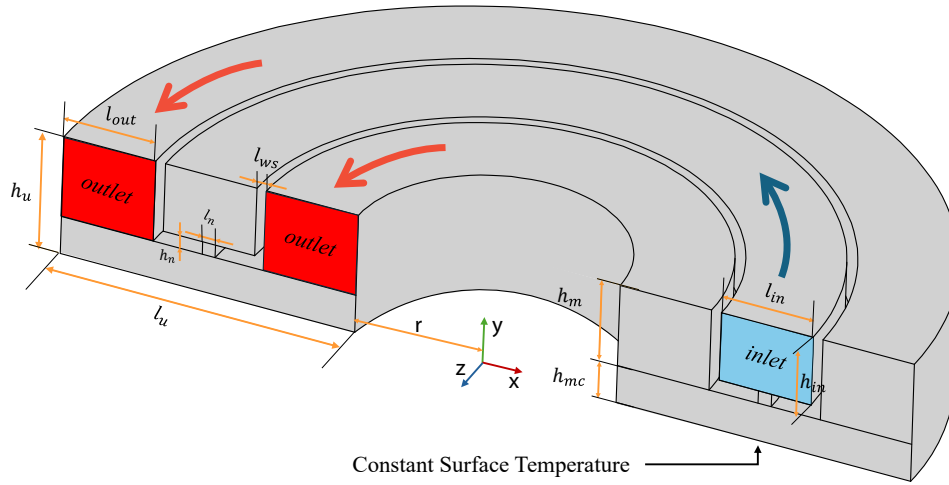


Figure 2.3: Unit manifold model geometry, geometric parameters, and boundary conditions.

Table 2.1: Geometric details of the unit manifold model.

Parameter	Definition	Dimension (mm)
l_{in}	Inlet length	0.7
l_{out}	Outlet length	0.7
l_n	Nozzle length	0.1
l_{ws}	Wall spacing length	0.075
l_u	Unit model length	2.25
h_m	Manifold height	0.65
h_{in}	Inlet height	0.55
h_{mc}	Microchannel height	0.3
h_n	Nozzle height	0.1
h_u	Unit model height	0.95
r	Radial distance	1, 4.65, 7.75

hydraulic diameter, which accounts for non-circular channel geometries, is defined as

$$D_h = \frac{2 l_{in} h_{in}}{l_{in} + h_{in}}. \quad (2.2)$$

In this expression, l_{in} and h_{in} correspond to the width and height of the inlet channel, respectively. This formulation enables an effective representation of flow characteristics in rectangular microchannels commonly used in cold plate designs.

Assuming a uniform distribution of flow across all manifold channels, the mass flow rate for an individual channel can be determined from the total volumetric flow rate using

$$\dot{m} = \frac{\dot{V}_{tot}}{N} \rho, \quad (2.3)$$

where $\dot{V}_{tot}$ is the total volumetric flow rate supplied to the manifold, ρ is the fluid density, and N represents the total number of manifold channels, which is set to 12 in the present study. The total volumetric flow rate within the main feeder channel varies from 0.1 L/min to 0.6 L/min. Under these conditions, the Reynolds number within each manifold channel remains below 1717, which is within the laminar regime and below the typical transitional threshold of approximately 2000.

At the outlet of the computational domain, a pressure outlet boundary condition with zero gauge pressure is imposed. This assumption ensures a stable and physically consistent representation of the outflow conditions while allowing the internal flow field to develop naturally based on the imposed inlet conditions and geometric configuration.

The governing equations in this study are solved using the finite volume method (FVM), which ensures conservation of mass, momentum, and energy over discrete control volumes. Numerical simulations are carried out using *ANSYS Fluent 2023 R1* to analyze three-dimensional fluid flow and heat transfer within the proposed heat sink configurations. To simplify the problem while maintaining physical accuracy, the following assumptions are applied:

- The flow and heat transfer processes are steady-state.

- The flow regime is laminar, and the coolant is treated as an incompressible Newtonian fluid.
- The thermophysical properties of the coolant are assumed to remain constant throughout the domain.

Under these assumptions, the governing equations describing the conservation of mass, momentum, and energy are expressed as follows. The continuity equation is

$$\nabla \cdot \mathbf{u} = 0, \quad (2.4)$$

the momentum equation is

$$\rho(\mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla P + \nabla \cdot (\mu \nabla \mathbf{u}), \quad (2.5)$$

and the energy equation is

$$\rho C_p(\mathbf{u} \cdot \nabla T) = \nabla \cdot (k \nabla T), \quad (2.6)$$

where ρ , μ , C_p , and k represent the fluid density, dynamic viscosity, specific heat capacity, and thermal conductivity of the coolant, respectively, while $\mathbf{u}$ denotes the velocity vector field, P is the pressure, and T is the temperature.

To evaluate and compare the performance of the proposed heat sink configurations, key thermo-hydraulic performance metrics are considered, including pressure drop, thermal resistance, and pumping power. The thermal resistance, which quantifies the ability of the heat sink to dissipate heat, is defined as

$$R_t = \frac{T_{hs,max} - T_{in}}{q'' A_{hs}} = \frac{\Delta T}{\bar{q}}, \quad (2.7)$$

where $T_{hs,max}$ is the maximum temperature at the base of the heat sink, which in this study is constrained by the maximum allowable chip temperature, and T_{in} is the inlet temperature of

the coolant. Consequently, the temperature difference ΔT remains constant at 53.15 K for all configurations. In addition, q'' represents the applied heat flux, A_{hs} is the heat sink surface area, and $\bar{q}$ denotes the total heat removed by the cooling system.

The pumping power required to drive the coolant through the manifold is calculated as

$$P_p = \Delta p \cdot \dot{V} = (P_{in} - P_{out}) \cdot \dot{V}, \quad (2.8)$$

where Δp is the pressure drop across the heat sink, P_{in} and P_{out} denote the inlet and outlet pressures, respectively, and $\dot{V}$ is the volumetric flow rate through each manifold channel, expressed in m^3/s . As a result, the pumping power is obtained in units of watts.

Since there exists an inherent trade-off between thermal performance and hydraulic losses, a thermal performance index (TPI) is introduced to evaluate the overall efficiency of the heat sink design. The TPI provides a comparative measure of heat transfer enhancement relative to the associated pressure drop penalty. In the present study, the TPI is defined as

$$\text{TPI} = \frac{\bar{q}_C / \bar{q}_S}{\left(\frac{\Delta P_C}{\Delta P_S} \right)^{1/2}}, \quad (2.9)$$

where the subscripts C and S refer to the curved and straight manifold configurations, respectively. A TPI value greater than unity indicates that the curved manifold design provides superior thermal performance relative to its hydraulic penalty compared to the baseline straight manifold configuration.

2.4 Mesh Independence Study and Validation

The Multizone meshing technique was employed to generate a high-quality, structured mesh within the revolved geometry of the unit manifold. This approach allows for better control over mesh alignment along the flow direction and improves numerical stability, particularly in regions with strong gradients such as near walls and within microchannels. To ensure the

accuracy and convergence of the numerical simulations, strict residual criteria were imposed. Specifically, all governing equations were required to converge to residuals below 10^{-6} , while a more stringent convergence criterion of 10^{-8} was applied to the energy equation to accurately capture thermal gradients and heat transfer behavior.

A mesh independence study was conducted to verify that the simulation results are not sensitive to mesh resolution. This step is essential to ensure that the numerical predictions are reliable and not influenced by discretization errors. In this analysis, both pressure drop and thermal resistance were evaluated across five different mesh densities, ranging from approximately 400,000 to 7,000,000 elements. The study was performed using the baseline configuration corresponding to the intermediate manifold length, with the nozzle positioned at the center of the inlet channel to represent a balanced flow condition.

The results of the mesh independence study, presented in Figure 2.4, illustrate the variation of pressure drop and thermal resistance as functions of mesh density for two different total volumetric flow rates. These results provide insight into the degree of numerical sensitivity associated with mesh refinement. It is observed that thermal resistance converges relatively quickly, with minimal variation beyond a mesh size of approximately 946,000 elements. In contrast, pressure drop exhibits a stronger dependence on mesh resolution due to its sensitivity to near-wall gradients and viscous effects. For both flow conditions, the relative error in pressure drop decreases below 1% when the mesh density reaches approximately 3,146,000 elements, indicating that further refinement yields negligible improvement in accuracy.

Based on these observations, the mesh with 3,146,000 elements was selected as the optimal compromise between computational cost and solution accuracy. This mesh density ensures that both thermal and hydraulic predictions are sufficiently accurate for the operating conditions considered in this study, while avoiding unnecessary computational expense associated with finer meshes.

To further assess the validity of the numerical model, simulation results for the baseline TMM heat sink were compared with experimental data reported by Kong et al. [74]. In their study,

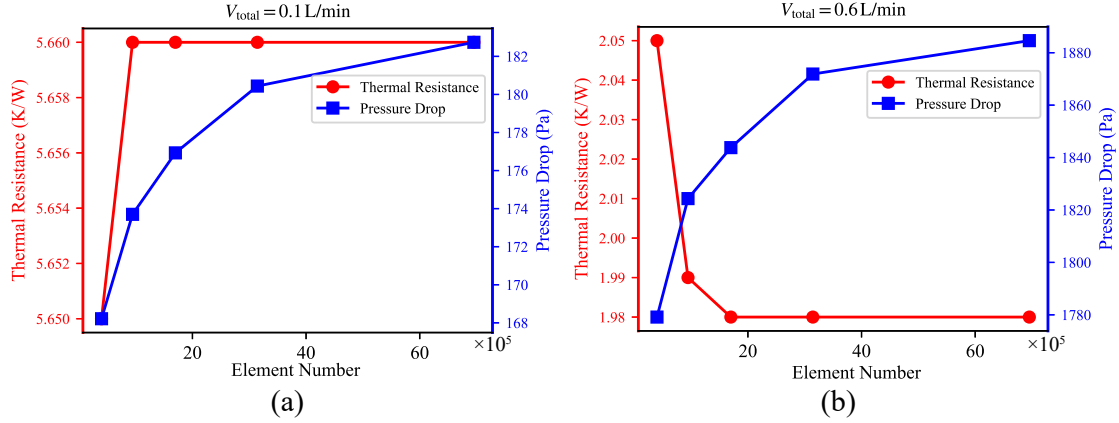


Figure 2.4: Mesh independence study of the middle manifold length with the middle nozzle for (a) the lowest total volumetric flow rate and (b) the highest total volumetric flow rate.

the authors investigated a TMM heat sink with a heated area of $10 \times 10 \text{ mm}^2$, subjected to a uniform heat flux of 100 W/cm^2 at the base. The dimensionless coefficient of performance (COP), defined as the ratio of heat dissipation to the pumping power required to drive the coolant, was used as the primary metric for comparison.

A comparison of COP as a function of total volumetric flow rate is presented in Figure 2.5 for both the current numerical predictions and the experimental data. The results show strong agreement between the two datasets across the investigated flow range, demonstrating that the numerical model accurately captures both the thermal and hydraulic behavior of the system. This validation confirms the reliability of the simulation framework and provides confidence in its application for evaluating and optimizing the proposed heat sink designs.

2.5 Results and Discussion

The effects of two key design parameters, namely the manifold channel length and the nozzle position along the inlet channel, are systematically investigated in this section. These parameters play a critical role in determining flow distribution, jet impingement behavior, and overall heat transfer performance within the heat sink. To gain deeper insight into the underlying fluid dynamics and thermal mechanisms, flow streamlines and velocity contours are analyzed in detail.

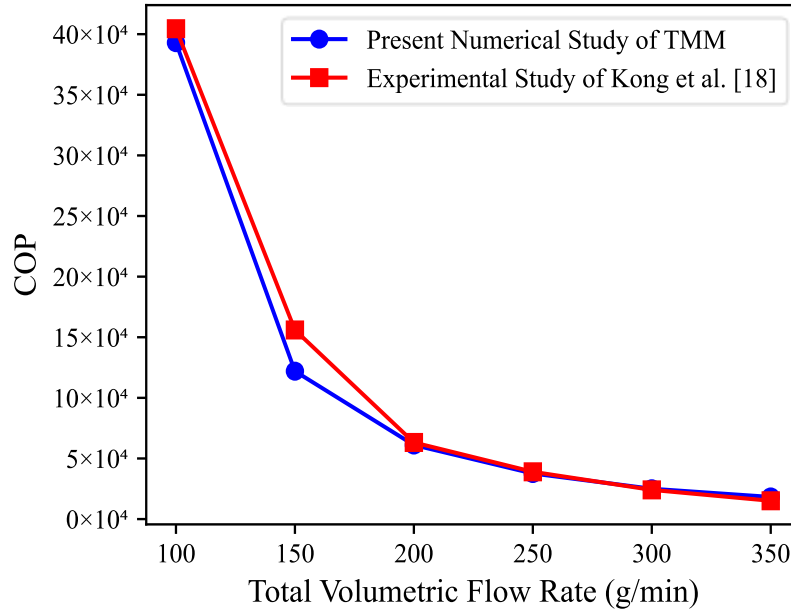


Figure 2.5: Validation of the simulation results against the experimental data of [74].

The study considers a range of operating conditions in which the total volumetric flow rate within the main feeder of the manifold varies from 0.1 to 0.6 L/min. This range enables the assessment of both low and relatively higher flow regimes, allowing for a comprehensive evaluation of how changes in flow rate influence velocity distribution, flow uniformity, and thermal performance. By examining these variations, the interaction between geometric design and operating conditions is clearly established, providing a better understanding of the performance enhancements associated with the proposed configuration.

2.5.1 Effects of Nozzle Position

The analysis begins by examining the influence of nozzle position on the overall heat transfer performance of the proposed RMM heat sink. As illustrated in Figure 2.6, the location of the inlet nozzle along the manifold channel is a critical design parameter that significantly affects flow distribution, jet impingement behavior, and consequently the thermal and hydraulic

characteristics of the system. To systematically evaluate this effect, numerical simulations were performed for three distinct nozzle positions, namely the left edge, the center, and the right edge, across all three manifold length configurations.

Figure 2.6 presents the variations in thermal resistance and pressure drop for the different nozzle positions. For all configurations, a consistent trend is observed: as the volumetric flow rate increases, thermal resistance decreases due to enhanced convective heat transfer, while pressure drop increases as a result of higher flow velocities and viscous effects. For comparison purposes, the performance of straight manifold channels with equivalent lengths is also included. The results indicate that curved manifolds consistently achieve superior thermal performance compared to their straight counterparts, owing to improved flow distribution and enhanced mixing. However, this thermal advantage comes at the expense of higher hydraulic losses, as evidenced by the increased pressure drop in curved configurations.

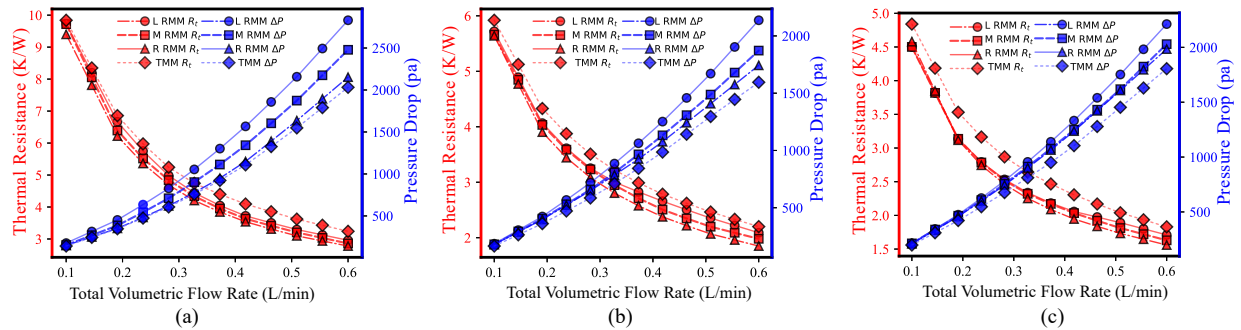


Figure 2.6: Variations of thermal resistance and pressure drop with volumetric flow rate for (a) the shortest manifold channel, (b) the middle manifold channel, and (c) the longest manifold channel.

From a hydraulic perspective, configurations with the nozzle positioned at the right edge exhibit lower pressure drop compared to other nozzle locations for all manifold lengths. In terms of thermal behavior, the shortest manifold channels show the highest thermal resistance across all flow rates. This is primarily attributed to their limited heat transfer surface area, which restricts heat dissipation. Nevertheless, the heat removal rate in these channels remains relatively high due to the dominance of entrance region effects, where thermal boundary layers are still

developing and heat transfer coefficients are elevated.

In contrast, the middle and longest manifold channels exhibit similar thermal resistance values. The longer channels benefit from increased surface area, which compensates for the gradual reduction in local heat transfer coefficients as the flow becomes thermally developed. As shown in Figure 2.7, the longest manifold configuration consistently yields the lowest thermal resistance across the investigated flow range. It is also observed that the reduction in thermal resistance tends to plateau at higher flow rates, indicating diminishing returns from further increases in flow rate regardless of manifold length.

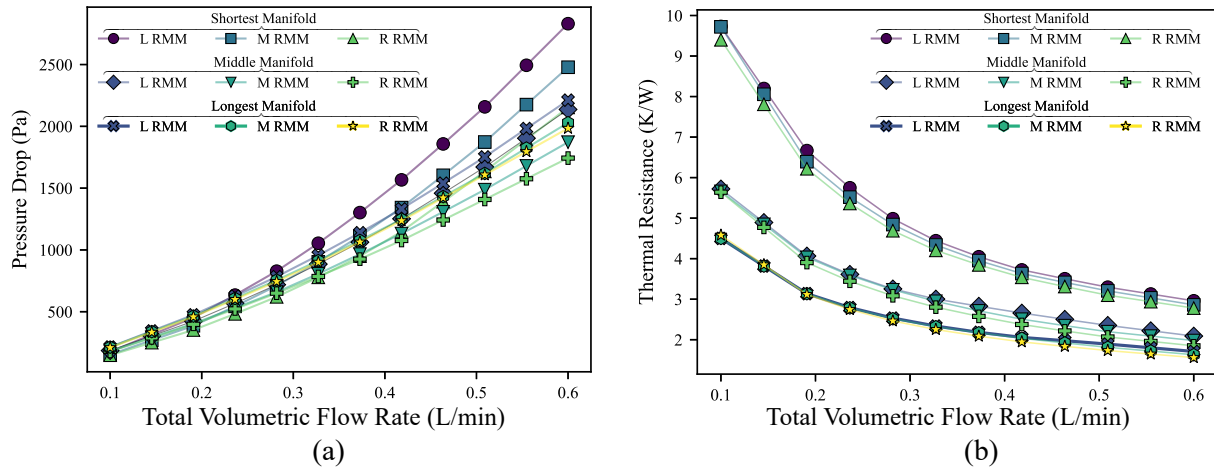


Figure 2.7: Variations of (a) pressure drop and (b) thermal resistance with volumetric flow rate for different manifold channel lengths.

The pressure drop behavior exhibits a more complex trend. Although the shortest manifold initially shows lower pressure drop at low flow rates, its pressure penalty increases more rapidly with increasing flow rate. This behavior is attributed to stronger entrance effects and the sharper curvature associated with a smaller radius, which induce higher local acceleration and pressure gradients. The fluid is forced to change direction more abruptly in shorter, highly curved channels, resulting in increased flow resistance compared to longer manifolds. Despite these differences, the right-edge nozzle configuration consistently demonstrates improved hydraulic performance across all manifold lengths.

The results highlight the strong interplay between manifold geometry and nozzle place-

ment, emphasizing the importance of optimizing both parameters to achieve a balance between thermal performance and hydraulic efficiency in advanced microchannel heat sink designs.

Figure 2.8(a–c) presents the flow streamlines within the middle manifold channel for all nozzle configurations at a total volumetric flow rate of 0.6 L/min. The results reveal distinct flow patterns that are strongly dependent on nozzle location. When the nozzle is positioned at the left edge of the inlet manifold, the coolant is forced to traverse a longer path before reaching the impingement surface, leading to a more distributed and less concentrated jet. In contrast, for the right-edge nozzle configuration, a significant portion of the coolant is directed toward the impingement region near the entrance of the channel, resulting in a more focused and effective jet interaction with the heated surface.

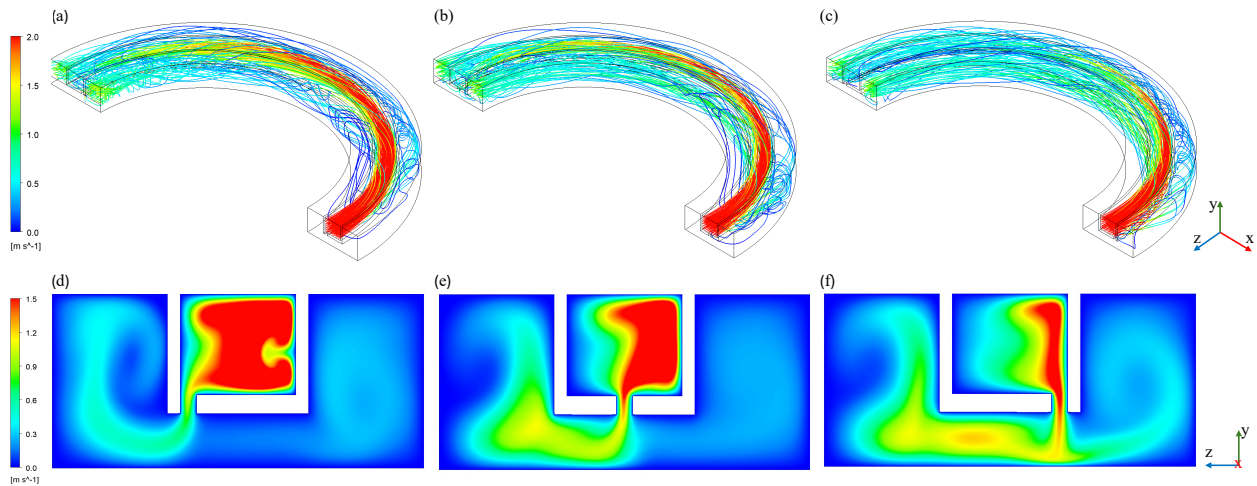


Figure 2.8: (a–c) Flow streamlines and (d–f) velocity contours of the middle-length manifold channel for different nozzle locations at a total volumetric flow rate of 0.6 L/min.

This behavior is further influenced by the curvature of the manifold channel, which induces centrifugal effects that preferentially drive fluid particles toward the outer (right) side of the channel. Positioning the nozzle beneath this region aligns the jet with the natural flow tendency, thereby enhancing the impingement strength and improving local heat transfer. In addition, the complex geometry of the manifold promotes the formation of localized vortical structures. These vortices increase fluid mixing and disrupt the thermal boundary layer, leading

to a more uniform temperature distribution and improved convective heat transfer performance.

The corresponding velocity contours, shown in Figure 2.8(d–f), are plotted on a mid-plane of the RMM heat sink to provide further insight into the flow dynamics for each nozzle position. The results indicate that the right-edge nozzle configuration produces a more organized and efficient flow field, characterized by a well-defined jet impingement region and relatively uniform velocity distribution. Compared to the other configurations, this case exhibits reduced flow recirculation and more effective utilization of the coolant momentum. In contrast, the left-edge and center nozzle configurations show more dispersed flow patterns, with less pronounced impingement behavior and weaker momentum concentration at the target surface.

These observations demonstrate that the interaction between manifold curvature and nozzle placement plays a critical role in governing flow structure, jet impingement characteristics, and ultimately the thermal performance of the RMM heat sink.

Figure 2.9(a–c) illustrates the temperature distribution on the mid-plane of the RMM heat sink for all three nozzle configurations, providing insight into the effect of nozzle placement on local heat transfer characteristics. As expected, the temperature of the working fluid increases most significantly near the bottom wall, where the imposed heat flux results in the highest thermal gradients and most intense heat exchange. Among the configurations, the right-edge nozzle produces the most uniform thermal boundary layer along the heated surface, indicating more effective coolant distribution and enhanced heat removal throughout the channel length.

In contrast, the center and left-edge nozzle configurations exhibit comparatively thicker thermal boundary layers, particularly in the mid-region of the channel. This behavior suggests reduced local convective heat transfer performance and the presence of less efficient coolant penetration in these regions. The non-uniformity in temperature distribution for these configurations can be attributed to weaker jet impingement and less favorable flow alignment with the channel curvature.

Further insight is provided by the two-dimensional streamlines shown in Figure 2.9(d–f), which highlight regions of flow stagnation and limited coolant access corresponding to elevated

temperature zones along the bottom wall. These stagnation regions impede effective heat transfer by reducing local fluid mixing and weakening thermal boundary layer disruption.

The results demonstrate that nozzle placement has a pronounced influence on both flow structure and thermal performance. The right-edge configuration, in particular, promotes improved coolant delivery, stronger boundary layer disruption, and more uniform temperature distribution, making it the most effective arrangement for thermal management in the proposed RMM heat sink design.

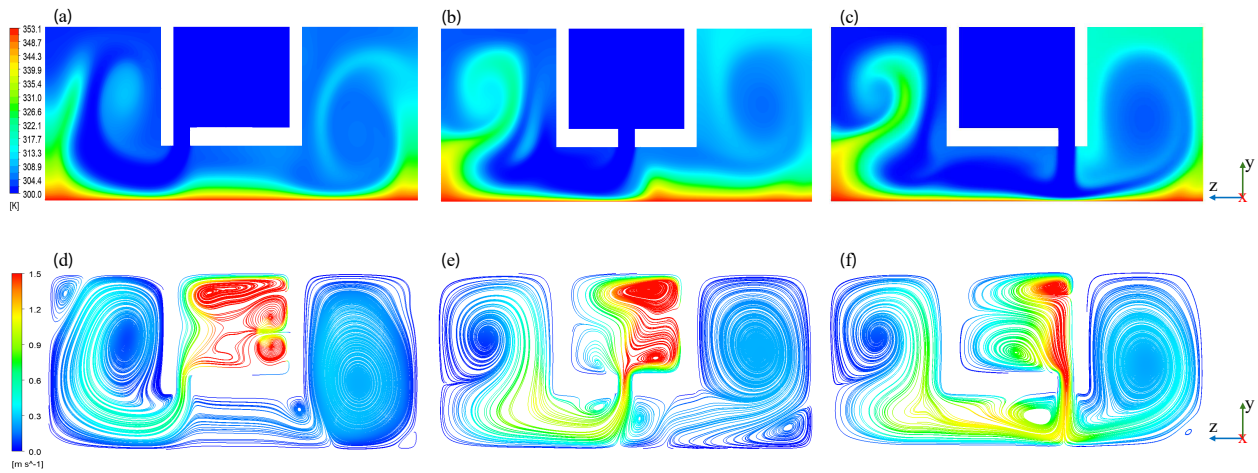


Figure 2.9: (a–c) Temperature contours and (d–f) two-dimensional streamlines of the middle-length manifold channel for different nozzle locations at a total volumetric flow rate of 0.6 L/min.

2.5.2 Effects of Channel Length

Figure 2.10 presents the heat flux contours for the three RMM heat sink configurations at a total volumetric flow rate of 0.6 L/min with the nozzle positioned at the right edge of the inlet channel. The results clearly demonstrate that manifold length has a pronounced effect on the spatial distribution of heat transfer, primarily due to variations in flow development and thermal boundary layer growth. In the shortest manifold configuration, which is largely dominated by entrance region effects, the heat flux distribution appears relatively high and uniform. This behavior can be attributed to the developing hydrodynamic and thermal boundary layers, which enhance local heat transfer coefficients and promote more effective heat removal.

In contrast, the middle and longest manifold configurations exhibit greater spatial variability in heat flux, characterized by alternating high- and low-flux regions along the channel length. These variations indicate partial flow maldistribution and reduced cooling effectiveness in certain regions, likely caused by flow deceleration, insufficient mixing, or the onset of localized stagnation zones as the flow progresses downstream.

Further insight into this behavior is provided by the corresponding flow streamlines shown in Figure 2.10(d–f). A direct comparison between the heat flux contours in Figure 2.10(a–c) and the streamline patterns reveals that regions with lower heat flux coincide with areas of reduced flow velocity and sparse streamline density. This observation confirms that the working fluid in these regions experiences limited mobility and mixing, and in extreme cases approaches near-stagnant conditions, thereby diminishing convective heat transfer performance.

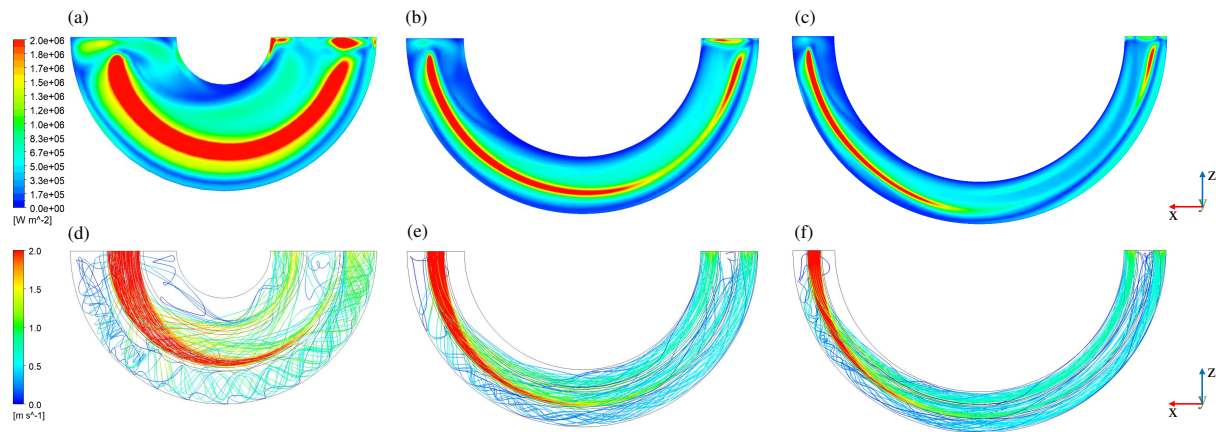


Figure 2.10: (a–c) Local wall heat flux values and (d–f) streamlines for all three manifold channel lengths at a total volumetric flow rate of 0.6 L/min.

Moreover, it is evident from Figure 2.10(d–f) that, for the right-edge nozzle configuration, increasing the manifold length results in a larger fraction of the coolant being discharged over a shorter portion of the channel. This behavior weakens the swirl-induced secondary flow structures in the downstream sections, leading to reduced flow uniformity and less effective heat removal in the outlet regions. Consequently, longer manifold channels tend to exhibit diminished

cooling uniformity compared to shorter configurations, highlighting the importance of balancing channel length with flow distribution characteristics in the design of high-performance RMM heat sinks.

Figure 2.11 presents the variation of pumping power for all curved manifold configurations alongside their corresponding straight manifold counterparts. The results highlight the inherent trade-off between thermal performance and hydraulic cost in the proposed designs. For a given pumping power, the longest manifold configuration consistently achieves the lowest thermal resistance across the entire range of operating conditions, indicating more effective heat removal for the same energy input. This improvement can be attributed to the increased surface area and enhanced flow development within longer channels, which promote more efficient convective heat transfer.

Among the different nozzle placements, the right-edge configuration demonstrates superior overall performance, achieving lower thermal resistance at comparable pumping power levels. This behavior is consistent with earlier observations, where improved flow alignment and stronger jet impingement enhanced coolant distribution and thermal boundary layer disruption. In contrast, the conventional straight manifold configurations exhibit the highest thermal resistance for a given pumping power, reflecting their relatively weaker flow mixing and less effective utilization of coolant momentum.

When evaluated from the perspective of maintaining a target thermal resistance, the longest manifold configuration again provides the most favorable outcome by requiring lower pumping power compared to shorter or straight channel designs. This indicates that, despite potential increases in pressure drop associated with extended channel length, the overall thermo-hydraulic efficiency of the system is improved. These findings underscore the importance of jointly optimizing manifold geometry and nozzle placement to achieve an optimal balance between heat transfer performance and energy consumption in advanced microchannel heat sink designs.

Figure 2.12 presents the thermal performance index for all investigated configurations in

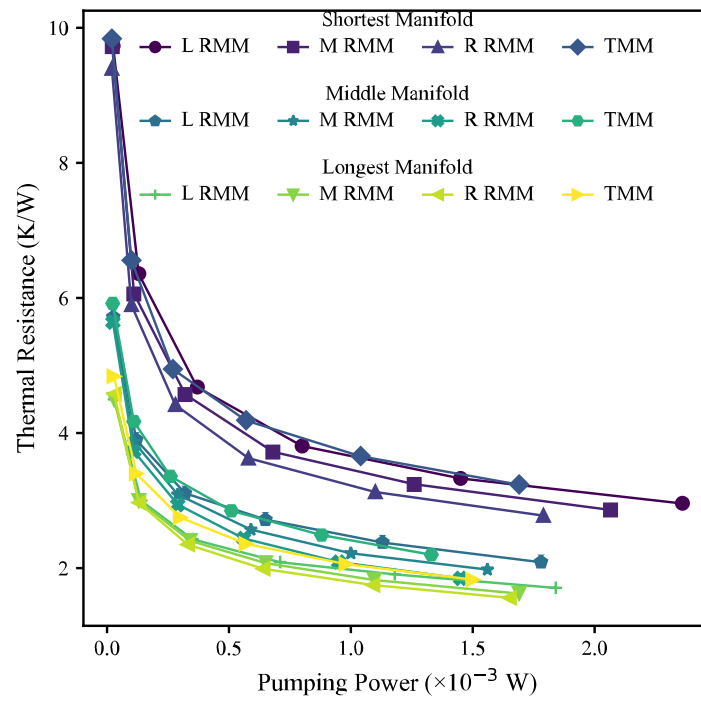


Figure 2.11: Thermal resistance as a function of pumping power for all studied curved manifold models along with their corresponding straight manifolds.

the form of a bar chart, allowing for a direct comparison of the thermo-hydraulic efficiency of the different manifold designs. The TPI serves as a comprehensive metric that accounts for both heat transfer enhancement and the associated hydraulic penalty. Values of TPI below unity indicate that a given configuration underperforms relative to the baseline straight manifold, rendering it an unfavorable design from an overall efficiency standpoint.

The results clearly demonstrate that configurations employing left-edge nozzle placement consistently yield TPI values below or close to unity, indicating limited or no improvement over the conventional design. This behavior can be attributed to poor flow alignment and reduced effectiveness of jet impingement, which ultimately diminish both thermal performance and flow utilization. In contrast, configurations with the nozzle positioned at the center or right edge of the inlet manifold channel exhibit significantly improved performance, with TPI values exceeding unity across a wide range of operating conditions.

At higher volumetric flow rates, these optimized configurations achieve TPI improvements of up to approximately 14%, highlighting the strong influence of nozzle placement on overall system efficiency. The enhanced performance is primarily driven by improved coolant distribution, stronger impingement effects, and more effective disruption of the thermal boundary layer. These findings underscore the critical role of nozzle positioning in the design of advanced manifold microchannel heat sinks and demonstrate that proper geometric optimization can lead to substantial gains in thermo-hydraulic performance.

2.6 Conclusions

- Curved manifold channels, through the introduction of swirl effects within both the inlet and outlet sections, improve the thermal performance of the heat sink compared to traditional straight manifold channels. This improvement amounts to approximately a 10% reduction in thermal resistance.
- Among the curved manifold designs evaluated, the configuration with the nozzle positioned at the right edge exhibited the best overall performance in both thermal and

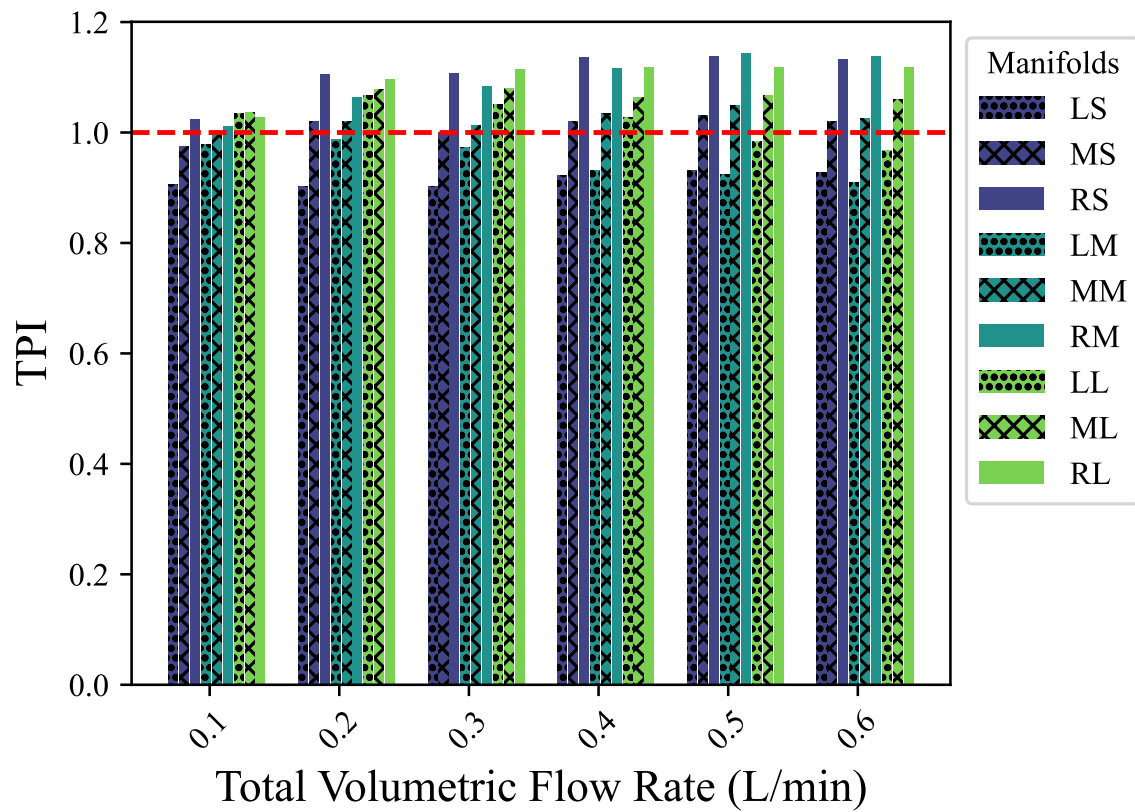


Figure 2.12: Values of the thermal performance index (TPI) for all manifold models with their nozzle placements over a range of volumetric flow rates.

hydraulic aspects. This setup leveraged centrifugal forces to improve coolant distribution and boost heat transfer, resulting in reductions in thermal resistance of 14.20%, 14.75%, and 15.9% for the shortest, middle, and longest manifold channels, respectively, at the highest volumetric flow rate considered in this study. Despite these thermal benefits, the pressure drop in the right-edge nozzle configuration remained higher than that of the straight manifold with comparable length and boundary conditions.

- Although straight manifold channels show lower pressure drops than curved ones, the thermal performance index indicates a notable advantage for curved manifold configurations. In particular, the right-edge nozzle design consistently achieves a TPI greater than unity across all channel lengths and flow rates, with the highest improvement—an increase of 14%—observed in the shortest manifold channel at a total volumetric flow rate of 0.6 L/min.
- The length of a curved manifold channel plays a crucial role in determining its thermo-hydraulic performance. Shorter manifold channels feature sharper curvatures, which can lead to increased pressure drop. Additionally, due to their limited length, these channels remain in the entrance region, where the thermal boundary layer has not yet fully developed. As a result, the average heat extraction rate is higher for shorter channels compared to longer ones at the same volumetric flow rate. On the other hand, the thermal resistances of the middle-length and longest manifold channels are relatively similar, specifically at higher flow rates. This is because as the channel length increases, it provides a larger surface area for heat transfer while simultaneously extending beyond the entrance region. This interplay creates a balance in the denominator of the thermal resistance equation, contributing to the observed similarities in thermal resistance between the middle and longest channels.

Chapter 3

Conjugate Heat Transfer Analysis and Geometric Optimization of a Radial Manifold Microchannel Heat Sink

3.1 Introduction

The results presented in Chapter 2 demonstrated that the right-edge nozzle configuration provides superior thermo-hydraulic performance compared with the other curved manifold designs. Consequently, this configuration was selected for further investigation under more realistic operating conditions. In this chapter, the right-edge radial manifold microchannel heat sink (RMMCHS) is numerically analyzed using a conjugate heat transfer model that considers separate fluid and solid domains with temperature-dependent thermophysical properties for both media. A comprehensive parametric study is conducted to evaluate the effects of the microchannel aspect ratio and the slot nozzle aspect ratio on the hydraulic and thermal performance of the proposed design.

3.2 Simulation Method and Data Reduction

3.2.1 Physical Model and Three-Dimensional Numerical Simulation Method

Conjugate heat transfer (CHT) refers to the simultaneous analysis of heat transfer within solid domains and fluid regions, accounting for the coupled interaction between conduction in solids and convection in fluids. In practical thermal systems, such as microchannel heat sinks, heat generated at the solid surface is conducted through the substrate and subsequently removed by the coolant via convective transport. The CHT approach captures this coupled behavior by solving the energy equation in both solid and fluid domains, while enforcing continuity of temperature and heat flux at the solid–fluid interface. This methodology provides a more physically accurate representation of thermal behavior compared to simplified models that consider only fluid flow or assume constant wall temperatures or heat fluxes.

In the context of manifold microchannel heat sinks, the use of a conjugate heat transfer framework is particularly important due to the strong interaction between the solid substrate and the cooling fluid, especially in regions with high heat flux and complex flow structures such as jet impingement and developing boundary layers. By incorporating both conduction and convection mechanisms within a unified computational domain, the CHT model enables a detailed evaluation of temperature distribution, thermal resistance, and local heat transfer characteristics. The conjugate heat transfer computational model was therefore employed in this study to investigate the thermo-hydraulic performance of the RMMCHS under the following assumptions:

- Fluid flow and heat transfer are considered to be in a steady-state condition.
- DI water is selected as the working fluid, operating within a laminar flow regime, and silicon is selected as the solid material.
- The thermophysical properties of both the fluid and solid regions are treated as temperature-dependent.
- Radiation heat transfer and viscous dissipation are negligible.

Based on these assumptions, the governing equations for the working fluid are defined as follows. The continuity equation is

$$\nabla \cdot \mathbf{u} = 0, \quad (3.1)$$

the momentum equation is

$$\rho_f C_{p,f} (\mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla P + \nabla \cdot (\mu_f \nabla \mathbf{u}), \quad (3.2)$$

and the energy equation in the fluid region is

$$\rho_f C_{p,f} (\mathbf{u} \cdot \nabla T_f) = \nabla \cdot (k_f \nabla T_f), \quad (3.3)$$

where ρ_f , μ_f , $C_{p,f}$, and k_f represent the density, viscosity, specific heat, and thermal conductivity of the working fluid, respectively. The only governing equation for the solid region is the energy equation, which can be expressed as

$$\nabla \cdot (k_s \nabla T_s) = 0, \quad (3.4)$$

where k_s is the thermal conductivity of silicon [68].

The temperature-dependent thermophysical properties of the materials utilized in the three-dimensional numerical simulation are summarized in Table 3.1. These properties are defined within the temperature range of 296 K to 373 K for DI water and 300 K to 500 K for silicon [86], [87].

Table 3.1: Thermophysical properties of the materials. In every correlation T denotes the local absolute temperature in kelvin.

Material property	DI water [86]	Silicon [87]
ρ [kg/m ³]	$1.025 \times 10^{-5} T^3 - 1.330 \times 10^{-2} T^2 + 4.935 T + 436.4$	2330
C_p [J/(kg · K)]	$-2.879 \times 10^{-5} T^3 + 3.778 \times 10^{-2} T^2 - 15.14 T + 6101$	$2.966 \times 10^{-6} T^3 - 5.091 \times 10^{-3} T^2 + 3.183 T + 140.3$
k [W/(m · K)]	$2.228 \times 10^{-8} T^3 - 3.059 \times 10^{-5} T^2 + 1.391 \times 10^{-2} T - 1.414$	$-7.342 \times 10^{-6} T^3 + 9.854 \times 10^{-3} T^2 - 4.652 T + 859.9$
μ [kg/(m · s)]	$-1.248 \times 10^{-9} T^3 + 1.353 \times 10^{-6} T^2 - 4.932 \times 10^{-4} T + 6.071 \times 10^{-2}$	—

The governing equations were discretized and solved using a finite volume framework based on control volumes within the *ANSYS Fluent 2023 R1* software package. This approach ensures strict conservation of mass, momentum, and energy over each computational cell, making it well suited for complex geometries such as manifold microchannel heat sinks. To accurately capture the coupling between pressure and velocity fields, the coupled pressure–

velocity algorithm was employed. This scheme solves the momentum and continuity equations simultaneously, providing improved convergence behavior and numerical stability, particularly for cases involving strong pressure gradients and complex flow structures.

Spatial discretization of gradients was carried out using the least-squares cell-based method, which offers enhanced accuracy on unstructured meshes and ensures reliable evaluation of spatial derivatives across irregular geometries. For the convective terms in both the momentum and energy equations, a second-order upwind scheme was adopted to minimize numerical diffusion and improve the resolution of velocity and temperature fields. Pressure discretization was performed using the standard scheme, which provides stable and consistent pressure interpolation across the computational domain.

3.2.2 Computational Domains and Boundary Conditions

Figure 3.1 presents the computational model employed in this study for the numerical investigation of the proposed RMMCHS configuration. To reduce computational cost while preserving the essential physics of the system, a representative periodic segment of the full geometry is considered. Specifically, a single inlet manifold section along with its two adjacent outlet manifold channels is modeled, which captures the key flow distribution and heat transfer mechanisms within the unit structure.

Figure 3.1(a) illustrates the computational domain corresponding to the fluid region. The coolant enters the domain through the inlet manifold with prescribed mass flow rates ranging from 13.85 to 96.95×10^{-5} kg/s and an inlet temperature of 23 °C. As the fluid progresses through the curved manifold channel, it is redirected toward the microchannel region via a slot jet nozzle positioned along the right edge of the inlet manifold. This configuration promotes jet impingement onto the heated substrate surface, enhancing local heat transfer. After interacting with the heated base and absorbing thermal energy, the fluid is discharged through two lateral outlet manifold channels, both defined with a pressure outlet boundary condition at zero gauge pressure, ensuring a physically consistent outflow.

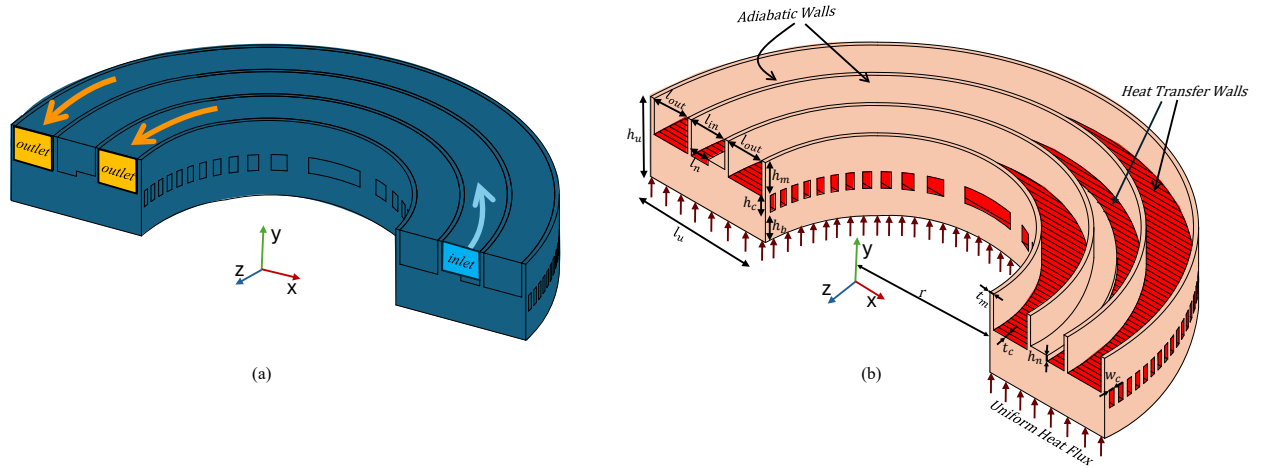


Figure 3.1: (a) Computational domain for the fluid region within the single manifold inlet channel and two lateral outlet channels. (b) Solid region of the model, including the substrate with its microchannels and the thermal boundary conditions for each surface.

The corresponding thermal boundary conditions are depicted in Figure 3.1(b). A uniform heat flux of 50 W/cm^2 is applied at the base of the substrate, representing the heat generation from the electronic component. All manifold walls, highlighted in orange, are treated as adiabatic to eliminate heat losses to the surroundings, thereby isolating the heat transfer within the cooling domain. In contrast, the microchannel walls and the substrate, highlighted in red, are assigned coupled boundary conditions, enabling the conjugate heat transfer mechanism between the solid and fluid regions. This setup ensures accurate representation of heat conduction within the substrate and convective heat removal by the coolant.

Table 3.2 summarizes the key geometric parameters illustrated in Figure 3.1(b), along with their corresponding values. To systematically evaluate the influence of geometric variations on performance, two dimensionless parameters are introduced. The parameter α represents the ratio of the slot nozzle length to the inlet manifold length,

$$\alpha = \frac{l_n}{l_{in}}, \quad (3.5)$$

while β denotes the ratio of the microchannel height to its width,

$$\beta = \frac{h_c}{t_c}. \quad (3.6)$$

A comprehensive parametric study was conducted by varying α from 0.1 to 0.5 and β from 2 to 4, allowing for a detailed assessment of the impact of nozzle geometry and microchannel aspect ratio on the thermo-hydraulic performance of the heat sink. Unlike conventional MMCHS designs, where all manifold channels have identical lengths, the RMMCHS configuration incorporates channels of varying lengths, as shown in Figure 3.1(b). This geometric variation introduces non-uniform flow paths, which can significantly influence local flow behavior, heat transfer characteristics, and overall system performance.

To account for these effects, simulations were performed for three distinct manifold channel lengths corresponding to radial distances of 2.35 mm, 4.65 mm, and 7.75 mm. These variations, combined with the selected ranges of α and β , resulted in a total of 45 unique design configurations. For each configuration, simulations were carried out at seven different mass flow rates within the laminar regime, enabling a comprehensive evaluation of the coupled effects of geometry and operating conditions on the thermal and hydraulic performance of the proposed heat sink design.

3.2.3 Grid Independence Test and Validation

Accurate prediction of flow behavior and conjugate heat transfer at the fluid–solid interface requires careful discretization of the computational domain, with particular attention to mesh quality in the near-wall and boundary layer regions [72]. In microchannel heat sink applications, where steep velocity and temperature gradients exist close to solid surfaces, inadequate mesh resolution can lead to significant errors in both thermal and hydraulic predictions. Therefore, a refined meshing strategy is essential to ensure reliable simulation outcomes.

Figure 3.2 presents the optimized unstructured polyhedral mesh employed in this study for the representative short manifold configuration with a microchannel aspect ratio of 4. The

Table 3.2: Key dimensions of the curved manifold microchannel heat sink.

Parameter	Description	Value (mm)
l_u	Unit model length	2.4
l_{in}	Manifold inlet length	0.7
l_{out}	Manifold outlet length	0.7
l_n	Slot nozzle length	0.07–0.35
t_m	Manifold wall thickness	0.075
t_c	Microchannel wall thickness	0.075–0.15
w_c	Wall spacing length	0.1
h_u	Unit model height	1.45
h_m	Manifold height	0.65
h_c	Microchannel height	0.3
h_b	Substrate height	0.5
h_n	Slot nozzle height	0.1
r	Radial distance	2.35–7.75

use of polyhedral elements enhances numerical stability and accuracy by providing improved cell connectivity and reduced skewness compared to conventional tetrahedral meshes. In the figure, the manifold channels and microchannel regions are highlighted with red mesh elements, while the surrounding fluid domain is illustrated in blue, clearly distinguishing the key regions of interest.

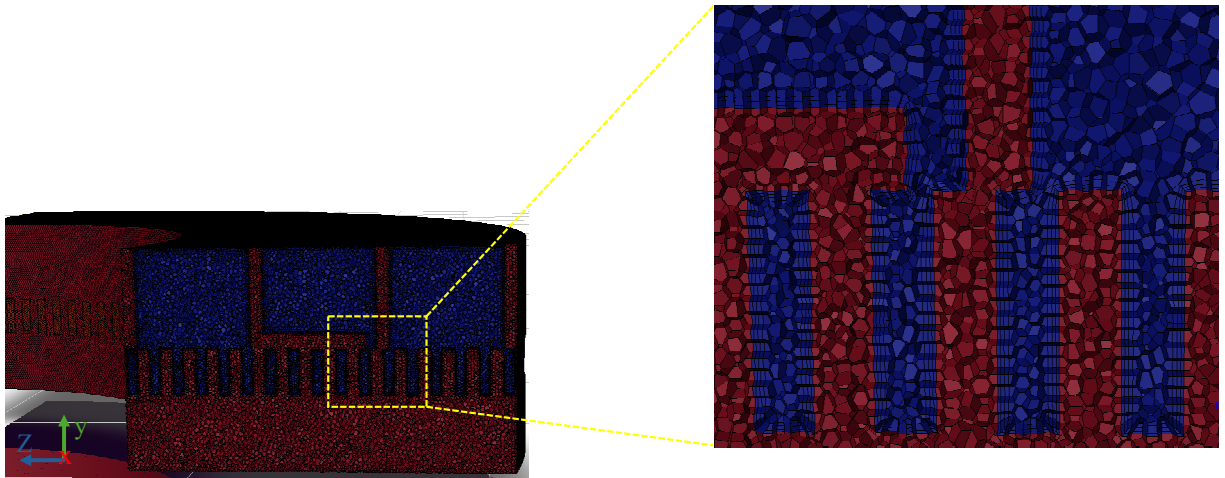


Figure 3.2: Local mesh of the computational domain on an arbitrary plane.

To accurately resolve the near-wall gradients, a series of inflation layers was generated

along all solid–fluid interfaces. These layers extend across the entire boundary layer thickness, ensuring that both velocity and temperature profiles are captured with high fidelity. This approach is particularly important for resolving viscous sublayer effects and thermal boundary layer development, which directly influence pressure drop and heat transfer performance.

The numerical solution was considered converged when the residuals of all velocity components and the continuity equation decreased below 1×10^{-6} , while a more stringent convergence criterion of 1×10^{-9} was applied to the energy equation. These criteria ensure that both the flow field and thermal field solutions are sufficiently stable and accurate for subsequent analysis.

To ensure that the numerical predictions are independent of grid resolution, a comprehensive mesh independence study was conducted for three representative configurations, as summarized in Table 3.3. These configurations were selected to span the design space considered in this work, including a short manifold model with $\alpha = 0.1$ and $\beta = 4$, a medium-length model with $\alpha = 0.5$ and $\beta = 3$, and a long manifold model with $\alpha = 0.3$ and $\beta = 2$. This selection enables evaluation of mesh sensitivity across different geometric scales and flow development characteristics.

For each configuration, simulations were performed at a representative mass flow rate of 69.25×10^{-5} kg/s using five progressively refined mesh densities. The key performance indicators, namely the pressure drop and the maximum base temperature of the substrate, were monitored to assess convergence with mesh refinement. These parameters were chosen due to their sensitivity to both near-wall resolution and overall flow distribution, making them suitable metrics for evaluating grid independence.

The results indicate that, for all three configurations, the deviations between the selected mesh (referred to as the model mesh in Table 3.3) and the finest mesh are negligible. Specifically, the relative difference in maximum base temperature remains below 0.07%, while the deviation in pressure drop is within 1.2%. The slightly higher sensitivity of pressure drop to mesh refinement is expected, as it is strongly influenced by near-wall velocity gradients and viscous effects.

Based on these findings, the selected mesh resolution is deemed sufficiently fine to accurately capture both the thermal and hydraulic characteristics of the system without incurring excessive computational cost. Therefore, all subsequent simulations were conducted using this mesh configuration, ensuring a reliable balance between numerical accuracy and computational efficiency.

Table 3.3: Mesh independence study. Each configuration was solved on five progressively refined meshes at a mass flow rate of 69.25×10^{-5} kg/s; the row labelled Model is the mesh used for all subsequent simulations.

Configuration	Mesh level	Elements ($\times 10^6$)	$T_{\text{base,max}}$ (K)	ΔP (kPa)
Short ($\alpha = 0.1, \beta = 4$)	Coarse mesh A	0.49	311.74	3.78
Short ($\alpha = 0.1, \beta = 4$)	Coarse mesh B	0.89	311.30	4.36
Short ($\alpha = 0.1, \beta = 4$)	Coarse mesh C	2.37	311.01	4.74
Short ($\alpha = 0.1, \beta = 4$)	Model	5.74	310.87	4.94
Short ($\alpha = 0.1, \beta = 4$)	Very fine mesh	11.41	310.76	5.03
Middle ($\alpha = 0.5, \beta = 3$)	Coarse mesh A	0.59	322.90	1.54
Middle ($\alpha = 0.5, \beta = 3$)	Coarse mesh B	0.764	322.73	1.56
Middle ($\alpha = 0.5, \beta = 3$)	Coarse mesh C	2.14	322.50	1.63
Middle ($\alpha = 0.5, \beta = 3$)	Model	8.74	322.25	1.68
Middle ($\alpha = 0.5, \beta = 3$)	Very fine mesh	17.02	322.03	1.70
Long ($\alpha = 0.3, \beta = 2$)	Coarse mesh A	0.634	333.15	2.01
Long ($\alpha = 0.3, \beta = 2$)	Coarse mesh B	1.22	332.73	2.07
Long ($\alpha = 0.3, \beta = 2$)	Coarse mesh C	2.57	332.87	2.13
Long ($\alpha = 0.3, \beta = 2$)	Model	12.16	332.73	2.21
Long ($\alpha = 0.3, \beta = 2$)	Very fine mesh	25.07	332.58	2.23

To validate the accuracy and reliability of the numerical methodology, the physical configuration reported in the experimental study of Drummond et al. [50] was reproduced and simulated under identical conditions. In particular, the geometry corresponding to sample C, with channel dimensions of $15 \mu\text{m} \times 300 \mu\text{m}$, was selected for validation. The simulations were carried out at a mass flux of $G = 2100 \text{ kg}/(\text{m}^2 \cdot \text{s})$, while maintaining the same boundary conditions and thermophysical properties as those reported in the experimental work.

The comparison between the numerical predictions and the experimental measurements of Drummond et al. [50] is presented in Figure 3.3. The results demonstrate excellent agreement, with a maximum deviation of less than 2 K across the investigated range. Such a small discrepancy indicates that the numerical model is capable of accurately capturing the coupled fluid flow and heat transfer phenomena within microchannel configurations.

This close agreement confirms the validity of the adopted modeling approach, including the governing equations, boundary conditions, and numerical schemes. Consequently, the developed computational framework can be considered sufficiently accurate and robust for predicting the thermo-hydraulic performance of the RMMCHS configurations investigated in this study.

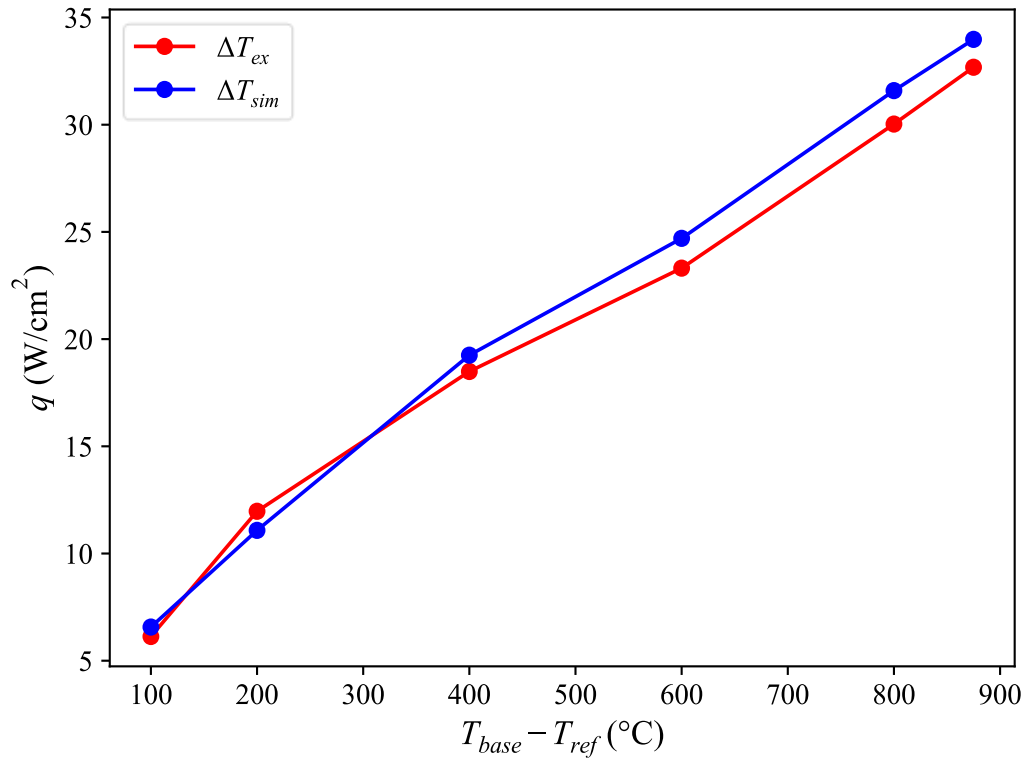


Figure 3.3: Comparison of the simulation results with the experimental data of [50].

3.2.4 Data Reduction

To comprehensively evaluate the flow and heat transfer performance of the RMMCHS under various geometric and operating conditions, a set of representative thermo-hydraulic performance parameters is employed. These parameters provide quantitative measures of heat dissipation capability, temperature uniformity, and flow resistance, enabling a systematic comparison of different design configurations.

The overall thermal resistance, which characterizes the ability of the heat sink to dissipate heat from the heated surface to the coolant, is defined as [40]

$$R_{\text{tot}} = \frac{T_{\text{base,max}} - T_{\text{in}}}{q'' A_{\text{hs}}}, \quad (3.7)$$

where $T_{\text{base,max}}$ is the maximum temperature at the base of the heat sink, and T_{in} is the inlet temperature of the coolant, which is fixed at 23 °C for all configurations considered in this study. The term A_{hs} represents the effective heat transfer area at the base of the heat sink, while q'' denotes the uniform heat flux applied to this surface. A lower value of R_{tot} indicates more effective heat removal and improved thermal performance.

To assess the uniformity of temperature distribution across the heat sink base, the mean absolute temperature deviation (MATD) is utilized [88], [89]. This parameter provides a measure of temperature non-uniformity and is expressed as

$$\text{MATD} = \frac{|T_{\text{base,max}} - T_{\text{base,avg}}| + |T_{\text{base,min}} - T_{\text{base,avg}}|}{2}, \quad (3.8)$$

where $T_{\text{base,max}}$, $T_{\text{base,avg}}$, and $T_{\text{base,min}}$ correspond to the maximum, average, and minimum temperatures at the heat sink base, respectively. Lower MATD values indicate more uniform temperature distributions, which are desirable for preventing thermal hotspots and enhancing device reliability.

The convective heat transfer coefficient, h , is used to quantify the effectiveness of heat

transfer between the solid surface and the coolant [67], and is defined as

$$h = \frac{q''}{T_{\text{base,ave}} - T_{\text{f,ave}}}, \quad (3.9)$$

where $T_{\text{f,ave}}$ is the average fluid temperature, calculated as the mean of the inlet and outlet temperatures of the microchannel region. This parameter reflects the ability of the coolant to absorb and transport heat away from the heated surface.

The hydraulic performance of the system is evaluated using the Darcy friction factor, which represents the flow resistance within the microchannels [90]. It is defined as

$$f = \frac{2 \Delta p_c D_h}{\rho_f u_c^2 L_{\text{ave}}}, \quad (3.10)$$

where Δp_c is the pressure drop across the microchannels, defined as the difference between inlet and outlet static pressures. The parameter D_h denotes the hydraulic diameter of the channel, u_c is the average fluid velocity at the microchannel inlet, and L_{ave} represents the average channel length. The fluid density ρ_f is evaluated at the inlet temperature. A higher friction factor indicates greater flow resistance and increased pumping requirements.

The hydraulic diameter of an individual microchannel, accounting for its rectangular geometry, is calculated as

$$D_h = \frac{2 t_c h_c}{t_c + h_c}, \quad (3.11)$$

where t_c and h_c denote the width and height of the microchannel, respectively. This definition enables accurate characterization of flow behavior in non-circular channels and is essential for evaluating both the Reynolds number and frictional losses.

3.3 Results and Discussion

This section presents a comprehensive analysis of the influence of key design and operating parameters on the thermo-hydraulic performance of the RMMCHS. The parameters investi-

gated include mass flow rate, manifold channel length, and two critical geometric ratios: the slot nozzle length to inlet manifold length ratio (α) and the microchannel height-to-width ratio (β). The effects of varying these parameters both individually and in combination are systematically evaluated. Detailed insights into the underlying flow physics and heat transfer mechanisms are obtained through the analysis of flow streamlines, velocity and temperature contour distributions, as well as variations in key performance indicators such as pressure drop and thermal resistance.

To enable a direct comparison of the trade-offs between hydraulic losses and thermal performance across different configurations, Figure 3.4 summarizes the results from a total of 315 CFD simulations. These simulations span a wide design space, with α varied from 0.1 to 0.5, β from 2 to 4, and three distinct radial distances (r), corresponding to short, middle, and long manifold channel configurations. The left column, Figure 3.4(a1–a3), illustrates the variation in pressure drop ($\Delta P = P_{\text{in}} - P_{\text{out}}$) across the computational domain, while the right column, Figure 3.4(b1–b3), presents the corresponding thermal resistance (R_{tot}) for each manifold length.

An increase in mass flow rate produces opposing effects on hydraulic and thermal performance. From a hydraulic perspective, higher flow rates reduce the thickness of the hydrodynamic boundary layer, leading to increased wall shear stress and, consequently, higher pressure drop. From a thermal standpoint, the reduction in thermal boundary layer thickness enhances convective heat transfer, resulting in lower thermal resistance. This opposing behavior highlights the inherent trade-off between energy consumption and cooling effectiveness in microchannel heat sink design.

Across all manifold configurations, the pressure drop increases with more extreme values of α and β . Smaller α values or larger β values tend to intensify flow acceleration and contraction effects at the manifold–microchannel interface, while also reducing the effective hydraulic diameter, both of which contribute to increased flow resistance. The sensitivity to α is more pronounced in shorter manifolds, where entrance effects and flow maldistribution play a dominant role. In contrast, longer manifolds exhibit reduced sensitivity to local geometric variations due

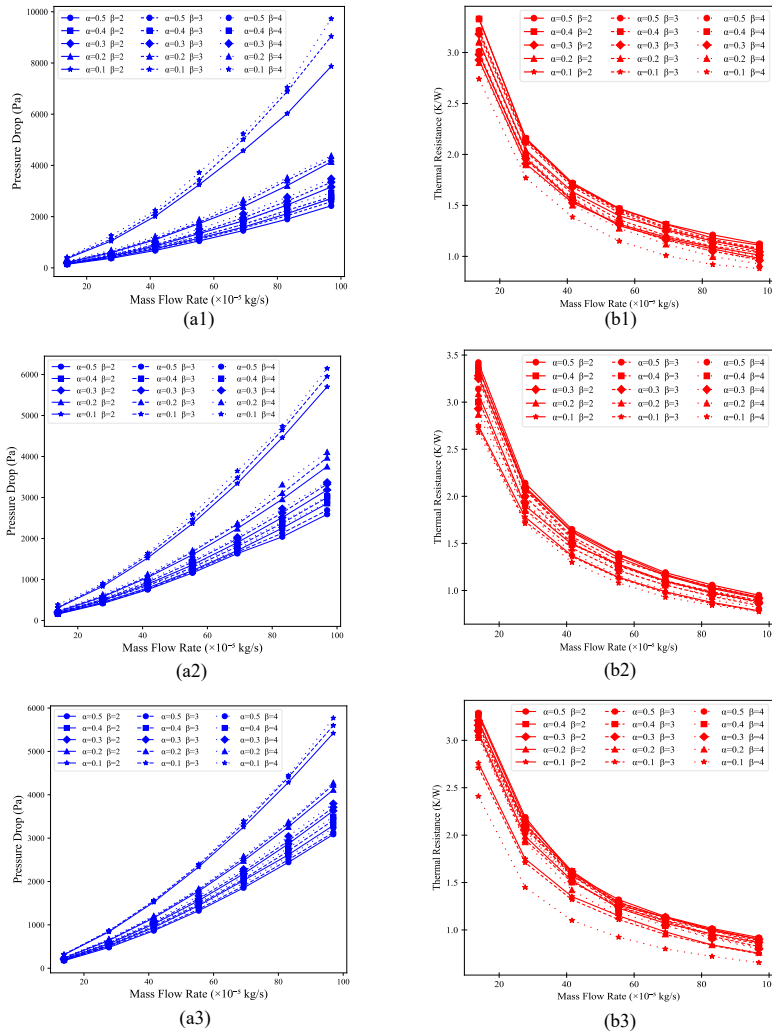


Figure 3.4: Variations of (a1–a3) pressure drop and (b1–b3) thermal resistance with mass flow rate for all models of the study.

to improved flow development, although they inherently experience higher baseline pressure drops due to increased frictional losses along the extended flow path.

Thermal resistance exhibits an inverse relationship with pressure drop. Configurations that promote higher local velocities, achieved through smaller α or larger β values, enhance convective heat transfer and reduce R_{tot} , albeit at the expense of increased hydraulic losses. Conversely, geometries that reduce pressure drop by increasing flow area or easing flow restrictions tend to exhibit weaker convective heat transfer, resulting in higher thermal resistance. These results emphasize the importance of optimizing geometric parameters to achieve a balanced thermo-hydraulic performance, rather than focusing solely on either thermal or hydraulic efficiency.

The results delineate a clear thermo-hydraulic trade-off, demonstrating that reductions in thermal resistance are consistently accompanied by increases in pressure drop, and vice versa. This inverse relationship highlights the competing nature of thermal and hydraulic performance in microchannel heat sink design. Configurations characterized by intermediate combinations of α and β exhibit more balanced behavior, avoiding the sharp increases in Δp observed at extreme geometric conditions. At the same time, these intermediate designs maintain significantly improved thermal performance compared to the least efficient configurations.

This behavior suggests that optimal performance is achieved not at the extremes of the design space, but rather within a moderate range of geometric parameters where both convective heat transfer enhancement and hydraulic penalties are carefully balanced. Consequently, identifying these intermediate configurations is essential for achieving efficient and practical heat sink designs that deliver strong thermal performance without incurring excessive pumping power requirements.

3.3.1 Effects of the Nozzle-to-Inlet Manifold Length Ratio

To comprehensively evaluate the thermo-hydraulic performance of the novel RMMCHS, three distinct cross-sectional planes denoted as AA' , BB' , and CC' were defined within the heat

sink. These planes are positioned at 45° from the domain inlet, at the geometric center of the heat sink, and at 45° from the domain outlet, respectively. Figure 3.5 depicts the locations of these planes in (a) the top view of the heat sink and (b) an illustrative three-dimensional view of the geometry.

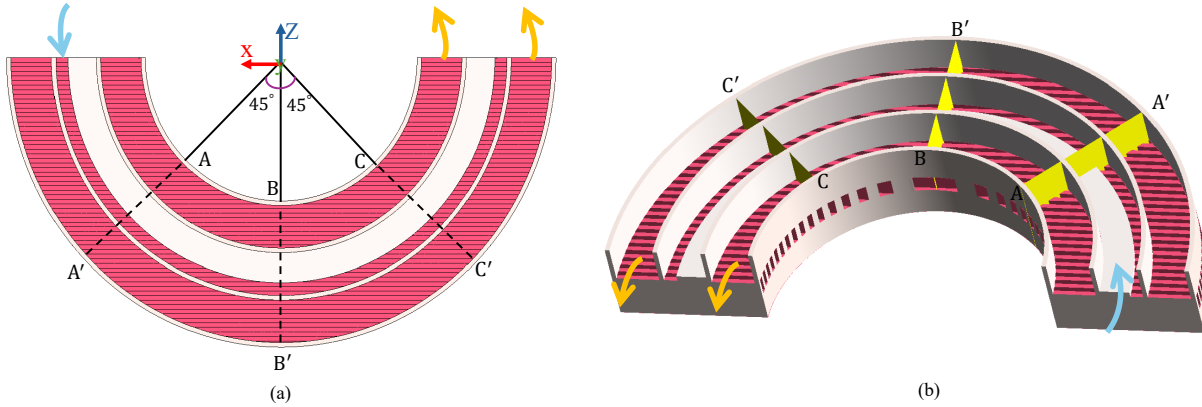


Figure 3.5: Location of the defined cross-sectional planes within the heat sink in (a) top view and (b) an arbitrary three-dimensional view.

Figure 3.6 presents the steady-state temperature distributions for the RMMCHS with a short manifold channel configuration at a mass flow rate of 69.25×10^{-5} kg/s under varying values of α . The top row, Figure 3.6(a1–e1), illustrates temperature contours on the heated wall (solid domain), while the bottom row, Figure 3.6(a2–e2), shows the temperature distribution on the BB' plane, capturing both solid and fluid regions. The investigated α values correspond to (a1, a2) $\alpha = 0.5$, (b1, b2) $\alpha = 0.4$, (c1, c2) $\alpha = 0.3$, (d1, d2) $\alpha = 0.2$, and (e1, e2) $\alpha = 0.1$.

A clear trend is observed along the heated surface, where decreasing α from 0.5 to 0.1 leads to a progressive reduction in peak temperature and a more uniform temperature distribution. For the case of $\alpha = 0.5$, a pronounced hotspot region is evident near the outer arc of the heat sink, indicating non-uniform coolant distribution and localized thermal accumulation. As α decreases, this hotspot diminishes in intensity, while the isotherms become more evenly spaced, reflecting improved lateral heat spreading and enhanced cooling uniformity across the surface.

The temperature contours on the BB' plane exhibit a consistent pattern within the coupled fluid–solid domain. At higher α values, the coolant penetration into the microchannel region is

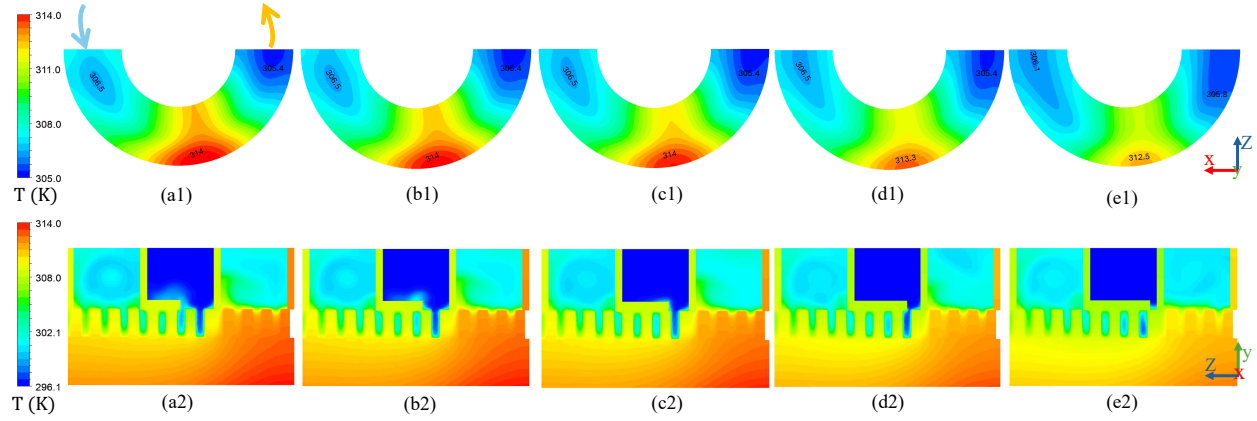


Figure 3.6: Contours of static temperature (a1–e1) on the heat sink base and (a2–e2) on the BB' plane for the short-manifold model with $\beta = 3$, an inlet mass flow rate of 69.25×10^{-5} kg/s, and varying values of α .

limited, resulting in thicker thermal boundary layers and steep temperature gradients along the flow direction. In contrast, lower α values promote deeper penetration of the coolant jet and stronger mixing, which reduces thermal boundary layer thickness and produces a more uniform temperature field. The case with $\alpha = 0.1$ demonstrates the most effective cooling behavior, characterized by the lowest maximum temperature and the most extensive cooling footprint.

This behavior can be physically attributed to the increase in jet momentum relative to channel width as α decreases. A smaller α effectively concentrates the inlet flow, enhancing impingement strength and improving coolant distribution throughout the microchannel array. As a result, convective heat transfer coefficients are increased, and a larger portion of the heat sink area actively participates in heat removal. However, this improvement in thermal performance is typically accompanied by an increase in pressure drop, reflecting the inherent trade-off between thermal and hydraulic performance.

The angular location of the hotspot remains largely unchanged across all α values, consistently appearing near the 90° sector of the curved channel. While its magnitude decreases with decreasing α , its position is governed by the interaction between channel curvature and outlet flow extraction. This region corresponds to the alignment between the heated wall hotspot and the warmer fluid zone associated with the right outlet manifold channel. The persistence of

this hotspot is attributed to curvature-induced secondary flows, commonly referred to as Dean vortices [91], [92]. These vortices arise from the imbalance between centrifugal and pressure forces in curved channels, generating pairs of counter-rotating flow structures perpendicular to the primary flow direction. The resulting flow maldistribution reduces coolant renewal in specific regions, thereby sustaining localized temperature elevations despite improvements in global cooling performance [93].

3.3.2 Effects of the Microchannel Aspect Ratio

Figure 3.7 presents the numerically obtained pressure contours and surface streamlines for the RMMCHS with the middle manifold length at three different microchannel aspect ratios, namely $\beta = 4$, $\beta = 3$, and $\beta = 2$. The upper row, Figure 3.7(a1–c1), illustrates the pressure distribution on the mid-plane of the manifold channel ($h_m/2$), while the subsequent rows, Figure 3.7(a2–c4), display the two-dimensional surface streamlines on the AA' , BB' , and CC' planes, respectively, providing detailed insight into the evolution of the flow field along the channel.

In the manifold mid-plane, the curvature of the channel induces a cross-stream pressure gradient that shifts the high-momentum core of the flow toward the outer arc of the bend. As β increases, corresponding to narrower microchannels, the mismatch between the header flow area and the microchannel inlet area becomes more pronounced. This geometric contraction intensifies entrance losses and leads to stronger local pressure drops downstream of the junction. For higher β values, distinct low-pressure regions develop near the channel entrances, while lower β values, corresponding to wider channels, facilitate a smoother pressure recovery along the flow path. These observations highlight the combined effects of contraction-induced losses and curvature-driven acceleration in shaping the pressure field.

Specifically, for $\beta = 4$, shown in Figure 3.7(a1), the pressure distribution is highly non-uniform, with strong gradients along the manifold and significant low-pressure pockets near the inlet of the microchannels, indicating elevated contraction losses. For $\beta = 3$, shown in Figure 3.7(b1), the pressure field becomes more uniform, with smoother gradients and reduced

severity of low-pressure zones, although some asymmetry between the inner and outer arcs of the bend remains evident. In contrast, for $\beta = 2$, shown in Figure 3.7(c1), the pressure contours exhibit the most uniform distribution, characterized by weaker gradients and more gradual pressure recovery, suggesting reduced maldistribution and improved hydraulic behavior.

The two-dimensional surface streamlines further elucidate how channel curvature, observation plane, and aspect ratio jointly influence the flow structure. As the fluid transitions from the manifold into narrower microchannels, it undergoes acceleration and forms a vena contracta near the entrance region [55]. This results in streamline contraction, localized pressure reduction, and elevated shear production before the flow reattaches downstream. At the AA' plane, which captures the early development of the flow, the $\beta = 4$ case, shown in Figure 3.7(a2), presents well-aligned and nearly parallel streamlines, indicating uniform flow distribution. For $\beta = 3$, shown in Figure 3.7(b2), small recirculation zones begin to appear near the channel walls, suggesting the onset of flow separation. In the $\beta = 2$ configuration, shown in Figure 3.7(c2), curvature-induced acceleration gives rise to more pronounced secondary vortical structures, which enhance mixing but also introduce additional form drag.

At the BB' plane, similar trends persist. The $\beta = 4$ case, shown in Figure 3.7(a3), maintains coherent and parallel streamlines, whereas $\beta = 3$, shown in Figure 3.7(b3), exhibits mild cross-flow between adjacent channels. For $\beta = 2$, shown in Figure 3.7(c3), the streamlines display significant bending and interaction, indicative of stronger secondary flow effects and enhanced mixing intensity. At the downstream CC' plane, the $\beta = 4$ configuration, shown in Figure 3.7(a4), achieves a relatively smooth exit profile, while $\beta = 3$, shown in Figure 3.7(b4), shows localized recirculation zones. In the $\beta = 2$ case, shown in Figure 3.7(c4), more prominent vortex structures are observed, reflecting increased exit losses associated with stronger flow disturbances.

It should be noted that the velocity scales in the streamline plots vary between different β configurations, as indicated by the legend bars. These differences reflect variations in absolute velocity magnitudes and must be considered when comparing flow intensity across cases. Collectively, these results demonstrate that while lower β values promote improved mixing

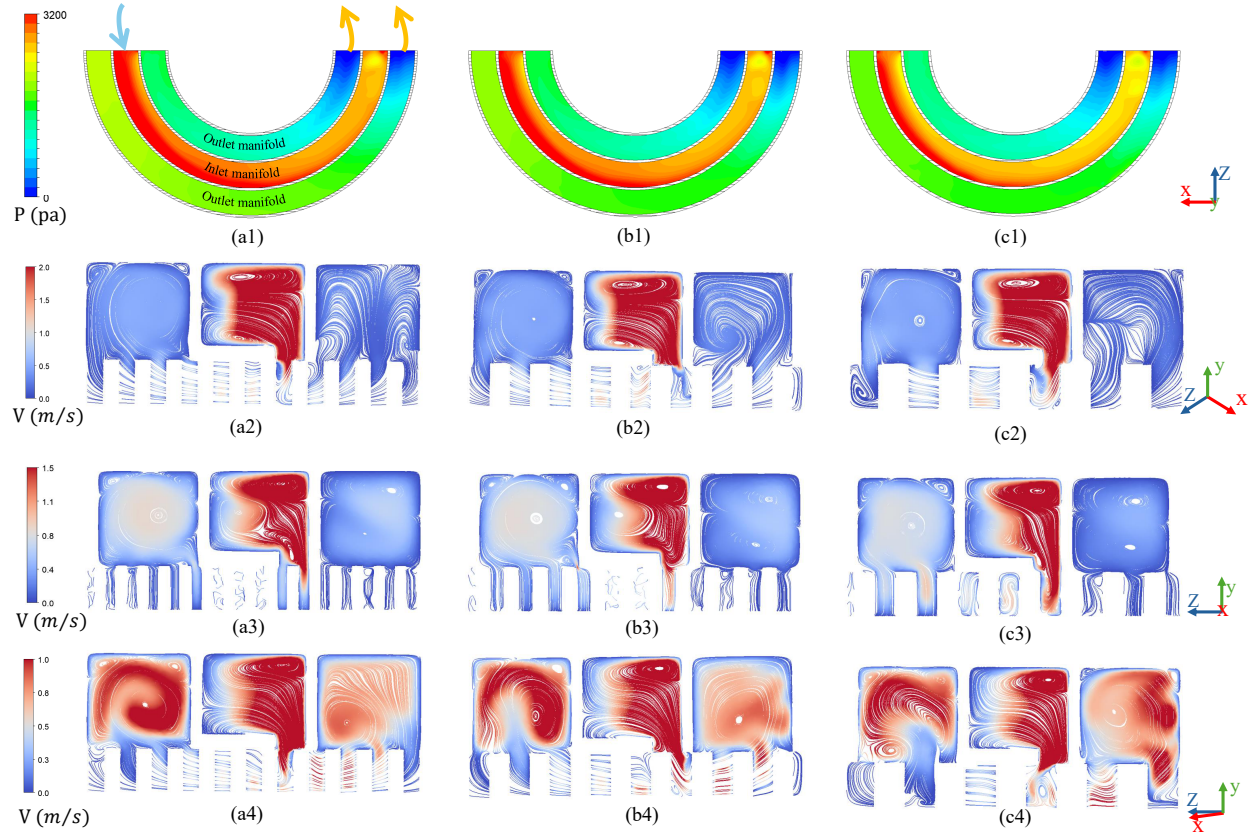


Figure 3.7: (a1–c1) Static pressure contours, (a2–c2) AA' plane surface streamlines, (a3–c3) BB' plane surface streamlines, and (a4–c4) CC' plane surface streamlines for the middle manifold model with $\alpha = 0.3$, a mass flow rate of 69.25×10^{-5} kg/s, and varying values of β .

and thermal transport through enhanced secondary flows, they also introduce higher hydraulic penalties due to increased pressure losses and flow disturbances.

Figure 3.8 presents the variation of the Darcy friction factor, f , and the convective heat transfer coefficient, h , for the same set of geometries as a function of β across the investigated range of mass flow rates. These integral performance metrics are consistent with the previously discussed local flow and thermal field characteristics.

As β increases, corresponding to a higher microchannel aspect ratio and therefore narrower channels, the hydraulic diameter decreases and the contraction ratio at the manifold–microchannel interface becomes more pronounced. These geometric changes intensify viscous effects and entrance losses, thereby requiring a higher pressure gradient to sustain the same flow rate. Although the absolute pressure drop increases under these conditions, the evaluated Darcy friction factor exhibits a decreasing trend with increasing β . This behavior arises because the denominator of the friction factor formulation, comprising fluid density, velocity, and hydraulic diameter terms, grows more rapidly due to increased flow velocities and temperature-dependent fluid properties, which collectively reduce the normalized friction factor value.

From a thermal perspective, increasing β enhances the convective heat transfer coefficient. This improvement is primarily attributed to higher core flow velocities and the corresponding reduction in thermal boundary layer thickness, which strengthens heat transfer at the fluid–solid interface. However, the rate of enhancement in h gradually diminishes at higher β values. This is due to the combined effects of increased entrance losses and curvature-induced flow maldistribution, which limit the effective delivery of coolant momentum to all microchannels.

These results highlight the coupled influence of geometric scaling and flow development on both hydraulic and thermal performance, reinforcing the importance of carefully balancing the channel aspect ratio to achieve optimal thermo-hydraulic efficiency.

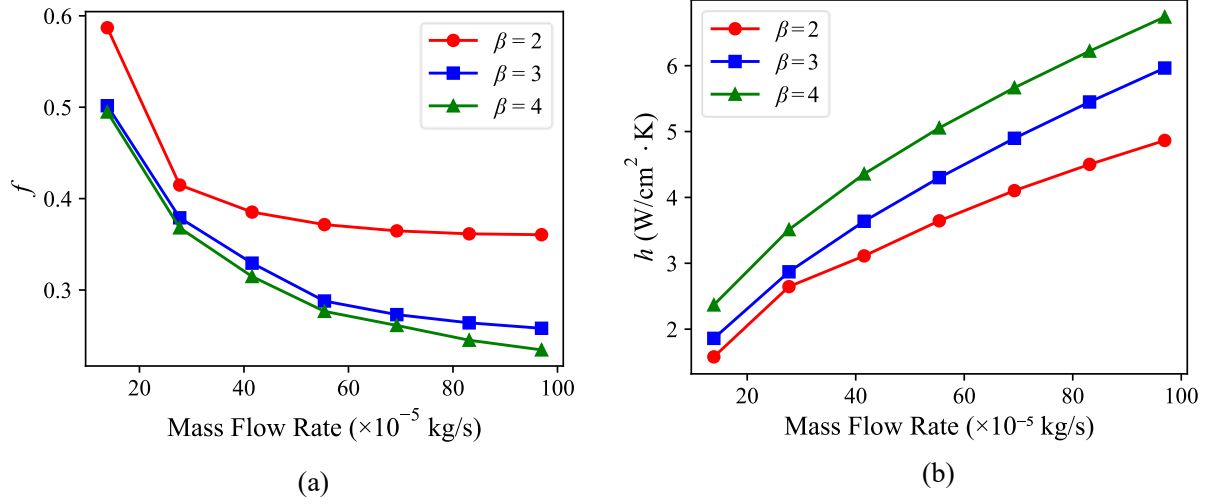


Figure 3.8: Trends of (a) the Darcy friction factor f and (b) the heat transfer coefficient h versus mass flow rate for the middle manifold length with $\alpha = 0.3$ and three different values of β .

3.3.3 Effects of Manifold Length

Figure 3.9 compares the steady-state wall temperature distributions at an inlet mass flow rate of 69.25×10^{-5} kg/s for three manifold channel lengths: (a) long, (b) middle, and (c) short. In all configurations, the hotspot consistently appears in the outer arc region of the curved manifold, aligned with the outlet-biased sector dictated by the combined effects of channel curvature and flow extraction. While the angular location of the hotspot remains essentially unchanged, its spatial extent and peak temperature exhibit a strong dependence on manifold length. The long manifold configuration, Figure 3.9(a), displays the broadest thermal footprint and the highest peak temperature, whereas the middle configuration, Figure 3.9(b), shows intermediate behavior. The short manifold, Figure 3.9(c), exhibits the most confined hotspot and the lowest peak temperature, indicating superior local cooling performance.

This trend can be interpreted through the field synergy principle [94], [95], which relates convective heat transfer enhancement to the alignment between velocity vectors and temperature gradients, as well as through the streamline topology in the outer arc region. As the manifold length increases, the cumulative pressure drop along the channel rises, leading to attenuation of flow momentum before reaching the downstream regions. This reduction in available mo-

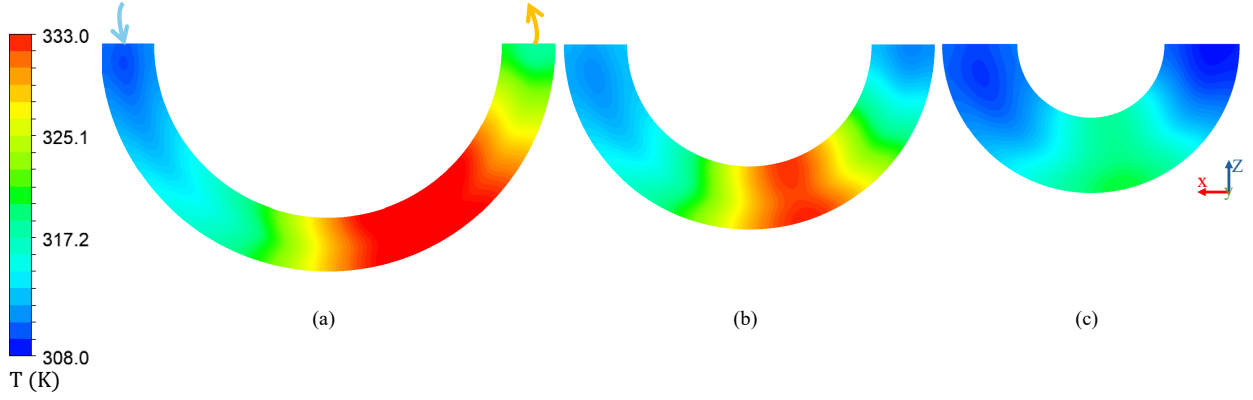


Figure 3.9: Heat sink base temperature contours for three manifold channel lengths: (a) long, (b) middle, and (c) short, at identical inlet mass flow rate, α , and β values.

momentum weakens the wall-normal component of velocity near the outer arc, resulting in thicker thermal boundary layers and elevated local temperatures over a broader azimuthal extent. In contrast, shorter manifolds preserve a greater portion of the inlet momentum and pressure, promoting stronger fluid renewal at the heated surface and more effective disruption of the thermal boundary layer. Consequently, the peak temperature is reduced and the hotspot region becomes more localized.

The velocity streamline patterns for the middle-length configuration, shown in Figure 3.10, provide further insight into the underlying mechanisms governing these temperature distributions. Within the hotspot region, the flow is predominantly tangential to the wall, with extended reattachment zones at the entrances of the microchannels. Under these conditions, the wall-normal velocity component is weak, leading to a larger local synergy angle and reduced convective heat transfer efficiency. For shorter manifold lengths, the retained jet momentum enhances the wall-normal velocity component, thereby improving the alignment between velocity and temperature gradient vectors, thinning the thermal boundary layer, and reducing peak temperatures. Conversely, longer manifolds experience increased distributed losses and stronger coupling between curvature and outlet flow extraction, which sustain larger regions of unfavorable flow alignment. This behavior contributes to both the expansion of the hotspot region and an increase in its peak temperature.

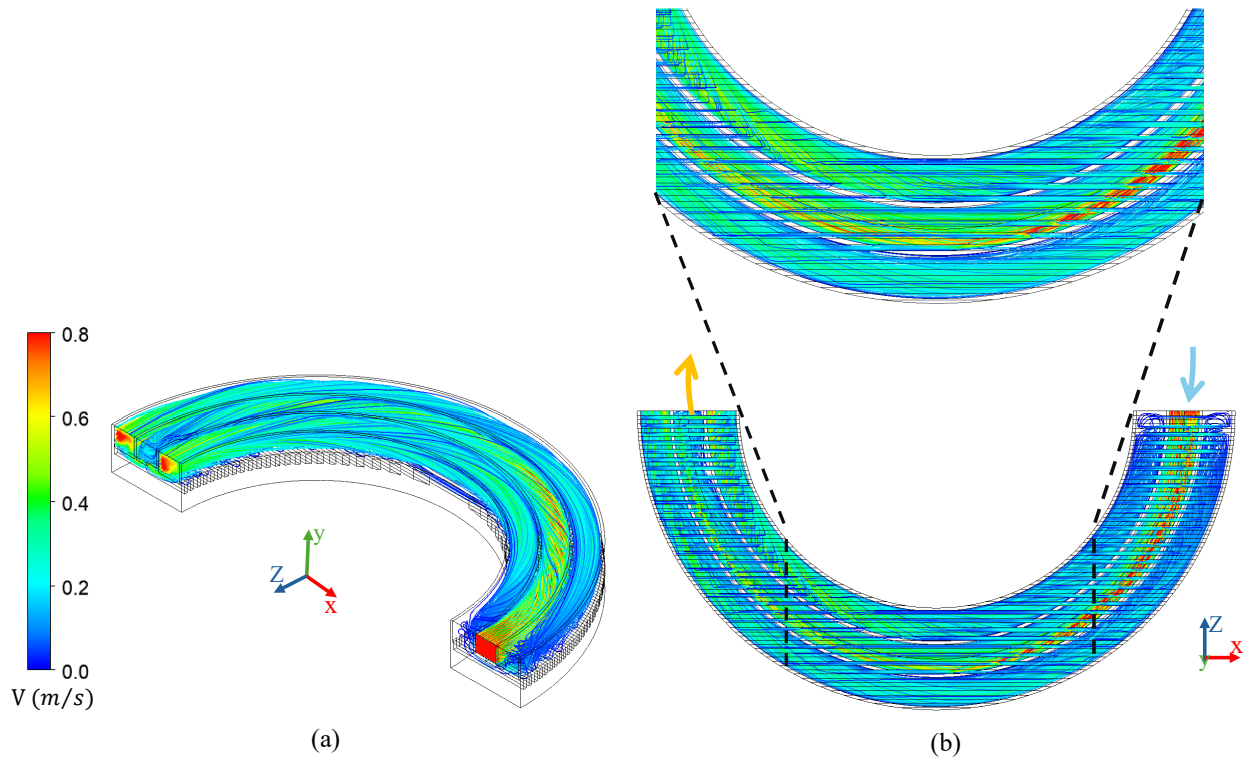


Figure 3.10: Flow streamlines in the middle manifold channel with $\alpha = 0.5$, $\beta = 3$, and an inlet mass flow rate of 41.55×10^{-5} kg/s: (a) isometric view and (b) bottom view of the heat sink.

Figure 3.11 presents the variation of the convective heat transfer coefficient, h , as a function of total mass flow rate for three microchannel aspect ratios. Each panel includes families of curves corresponding to two extreme values of α and three manifold channel lengths. The discussion that follows isolates the effect of manifold length by comparing configurations at fixed total mass flow rate, α , and β .

Across all β values, the heat transfer coefficient increases with increasing mass flow rate due to enhanced convective transport and thinning of the thermal boundary layer. However, when the mass flow rate and geometric ratios are held constant, manifold length exhibits a clear and systematic influence on h . In general, the shortest manifold configuration yields the highest heat transfer coefficient, followed by the middle length, while the longest manifold consistently produces the lowest values of h . This ordering is most pronounced at low to moderate mass flow rates and becomes less distinct as the flow rate increases.

The underlying mechanism can be attributed to the increase in hydraulic resistance along longer manifolds. For a fixed total mass flow rate, longer channels require larger axial pressure gradients to sustain the flow, which reduces the available pressure at downstream microchannel inlets. This results in weaker wall-normal momentum injection into several channels, leading to thicker thermal boundary layers and reduced local convective heat transfer. Consequently, the average heat transfer coefficient decreases with increasing manifold length.

Additionally, the curved geometry of the manifold amplifies these effects. As the manifold length increases, curvature-induced secondary flows, commonly referred to as Dean vortices, and outlet-induced flow extraction act over a longer distance, promoting azimuthal flow bias toward the outer arc. This behavior enhances flow maldistribution, particularly in downstream regions, and reduces the effectiveness of coolant renewal across the microchannel array. The combined effect is a reduction in h and a broader region of diminished thermal performance for longer manifolds.

The sensitivity of h to manifold length is strongly influenced by the microchannel aspect ratio β . For $\beta = 4$, where the channels are relatively narrow, the contraction ratio at the

microchannel entrance is high, resulting in pronounced vena-contracta effects. In this case, shorter manifolds better preserve jet momentum and maintain higher inlet velocities into the microchannels, leading to significantly higher local heat transfer coefficients. As the manifold length increases, these contraction effects become more severe downstream, intensifying flow maldistribution and further reducing h . Consequently, the difference in h between short and long manifolds is most significant for higher β values.

For lower aspect ratios ($\beta = 3$ and $\beta = 2$), similar trends are observed, although the magnitude of the effect is reduced. The vena-contracta phenomenon is still present, but its intensity and downstream persistence are weaker, resulting in less pronounced sensitivity to manifold length.

The influence of the nozzle parameter α also plays a critical role. Smaller values of α correspond to higher inlet jet momentum, which enhances flow penetration into the microchannel array and improves convective heat transfer. This effect increases h for all manifold lengths and partially mitigates the adverse impact of increased channel length. As a result, the influence of manifold length on h is more evident for larger α values, where jet momentum is weaker. Furthermore, at higher mass flow rates, inertial effects dominate and reduce the relative impact of header pressure losses, leading to convergence of h values across different manifold lengths.

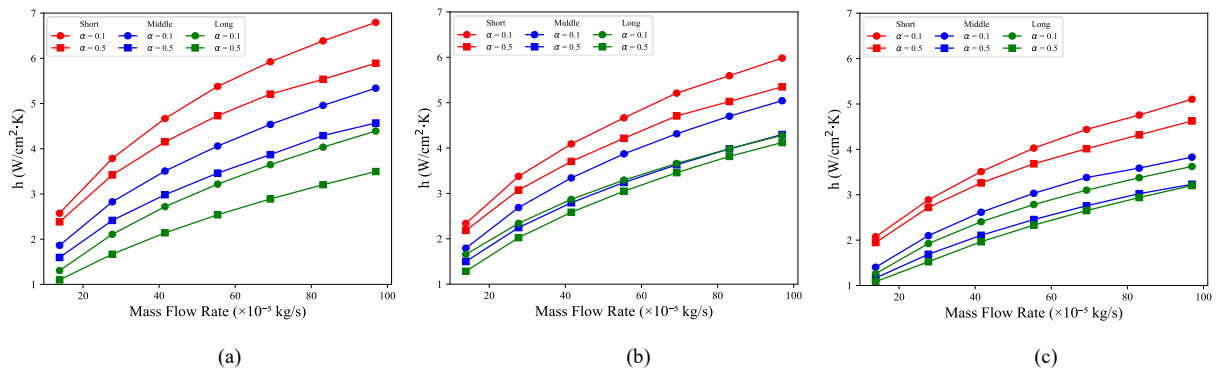


Figure 3.11: Heat transfer coefficient h versus mass flow rate for three manifold lengths and two extreme values of α , arranged by microchannel aspect ratio: (a) $\beta = 4$, (b) $\beta = 3$, and (c) $\beta = 2$.

Figure 3.12 presents the variation of the mean absolute temperature deviation as a func-

tion of mass flow rate for the same set of geometries considered in the heat transfer coefficient analysis, organized by microchannel aspect ratio: (a) $\beta = 4$, (b) $\beta = 3$, and (c) $\beta = 2$. Across all configurations and operating conditions, MATD decreases with increasing mass flow rate, indicating enhanced thermal uniformity as convective transport becomes stronger. This reduction is more pronounced at low flow rates, where increases in flow significantly improve coolant renewal, and gradually diminishes at higher flow rates as the system transitions toward an inertia-dominated regime.

For a fixed mass flow rate, α , and β , a consistent ordering is observed with respect to manifold length. The shortest manifold configuration yields the lowest MATD, indicating superior temperature uniformity, while the middle length exhibits intermediate behavior, and the longest manifold produces the highest MATD values. This trend aligns with the previously discussed temperature field distributions and hotspot characteristics. Shorter manifolds preserve a greater portion of wall-normal momentum, particularly in the outer arc region, promoting effective disruption of the thermal boundary layer and reducing temperature gradients. In contrast, longer manifolds accumulate distributed hydraulic losses and amplify curvature–outlet coupling effects, leading to increased flow maldistribution, broader warm regions, and higher deviation from the mean temperature.

The influence of the geometric parameter α follows a similar trend. Smaller values of α , corresponding to stronger inlet jet momentum, enhance coolant penetration into the microchannel array and improve flow alignment with local temperature gradients. This reduces the synergy angle in regions prone to thermal accumulation and consequently lowers MATD. Conversely, larger α values weaken the inlet jet, increase the prevalence of tangential flow relative to isotherms, and result in higher MATD values.

The sensitivity of MATD to manifold length is also strongly dependent on the aspect ratio β . For $\beta = 4$, where the channels are relatively narrow, strong contraction effects and pronounced vena-contracta phenomena amplify flow maldistribution along the curved manifold, making MATD more sensitive to changes in channel length. As β decreases toward 2, the contraction

effects become less severe, reducing curvature-induced non-uniformities and diminishing the influence of manifold length on MATD.

It is noteworthy that although higher β values generally enhance the convective heat transfer coefficient due to increased flow velocities and shear, they do not necessarily guarantee improved thermal uniformity. The same mechanisms that increase h , such as strong jetting and elevated shear rates, can also intensify spatial temperature gradients if flow distribution is not properly controlled. Consequently, achieving both high heat transfer performance and low MATD requires careful optimization of manifold length and the nozzle parameter α , ensuring a balance between enhanced convection and uniform coolant distribution.

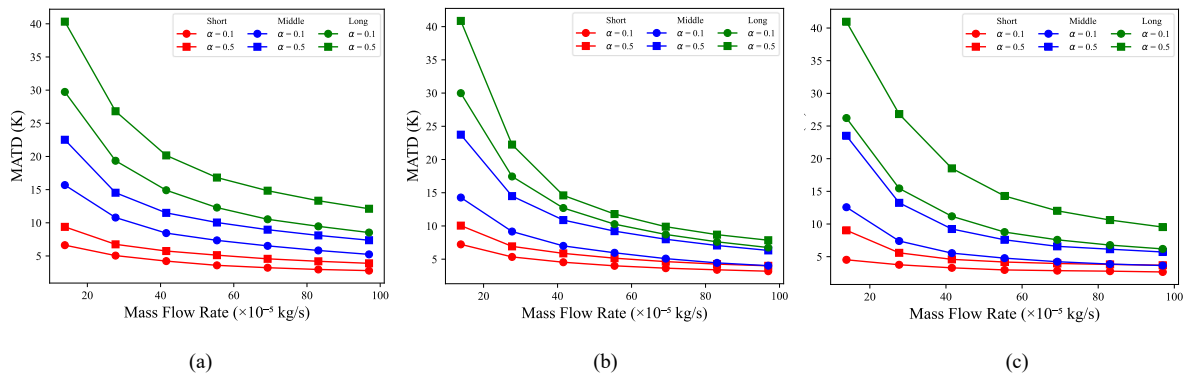


Figure 3.12: Mean absolute temperature deviation (MATD) versus mass flow rate for three manifold lengths and two extreme values of α , arranged by microchannel aspect ratio: (a) $\beta = 4$, (b) $\beta = 3$, and (c) $\beta = 2$.

3.4 Conclusions

This chapter proposes a novel manifold microchannel heat sink design incorporating curved manifold channels to improve both thermal and hydraulic performance. Detailed three-dimensional numerical simulations employing a conjugate heat transfer model were performed on representative unit cells to analyze heat transfer and fluid flow behavior under single-phase conditions. The study systematically investigates the influence of two dimensionless parameters: α , defined as the ratio of slot nozzle length to manifold inlet length, and β , defined as the ratio of

microchannel height to width, on the performance of the RMMCHS across three distinct curved manifold lengths.

The findings from this investigation can be summarized as follows:

- Higher flow rates enhance cooling effectiveness but incur greater pumping penalties. The ranking of optimal α - β configurations varies with manifold length. In the short manifold, the pressure drop penalty between the best and worst thermal performance cases ranges from 195% to 301% across different mass flow rates, whereas in the long manifold the range narrows to 81%–87%. This highlights the importance of tailoring aspect ratios to the geometric constraints and desired performance objectives.
- The severity of the hotspot is strongly dependent on α . At higher α , the peak wall temperature is higher and temperature gradients across the surface are steeper, indicating poor uniformity. As α decreases, jet momentum and penetration improve, enhancing channel engagement around the bend, thinning the thermal boundary layer in the affected sector, and extending the cold fluid plume farther downstream. For the short manifold configuration with $\beta = 4$, the thermally optimal case, the heat transfer coefficient increases by approximately 7.5% to 13.2% as α is reduced from 0.5 to 0.1. The $\alpha = 0.1$ case exhibits the most uniform distribution and the lowest peak wall temperature, though such improvement is typically accompanied by an increase in pressure drop.
- Reducing β enhances pressure uniformity by easing entrance contraction, lowering hydraulic resistance, and mitigating curvature-induced pressure imbalances across the array, although Darcy friction values increase because of the steep decrease of velocity in the denominator. Increasing β boosts h yet can leave residual uniformity penalties unless α and the header length are co-optimized. For the short manifold configuration with the most pronounced nozzle arrangement, the performance improves by 24% to 34% across the investigated mass flow rate range when β is increased from 2 to 4.
- Manifold channel length is a first-order geometric lever for convective heat transfer per-

formance. Shorter headers preserve inlet momentum and increase h , whereas longer headers increase distributed header losses and curvature-driven maldistribution, reducing h and widening hotspots. The magnitude of this effect scales with β and α , and it vanishes progressively as mass flow becomes strongly inertia-dominated. The overall thermal performance enhancement of the RMMCHS ranges from 54% to 97% for $\alpha = 0.1$ and $\beta = 4$, depending on whether the short or long channel configuration is considered.

Chapter 4

Numerical Investigation of Tapered Curved Radial Manifold Microchannel Heat Sinks Using Conjugate Heat Transfer

4.1 Introduction

This final chapter investigates the thermo-hydraulic performance of tapered curved manifold heat sinks using the same CHT approach employed in Chapter 3. Separate fluid and solid domains are considered, with temperature-dependent thermophysical properties assigned to both media to accurately represent realistic operating conditions.

The objective is to evaluate the influence of tapered curved manifolds on coolant flow distribution, pressure drop, heat transfer, and overall thermal performance. By gradually varying the manifold cross-sectional area along the flow path, the proposed design aims to improve coolant distribution, reduce flow maldistribution, and enhance the thermo-hydraulic performance of the radial manifold microchannel heat sink.

4.2 Simulation Method and Data Reduction

4.2.1 Physical Model and Three-Dimensional Numerical Simulation Method

Figure 4.1 presents both the conventional manifold design and the proposed tapered curved manifold configuration. This study aims to quantify the impact of manifold tapering on both hydraulic and thermal performance metrics. Particular attention is given to variations in pressure drop, thermal resistance, temperature uniformity, flow streamlines, and local heat transfer characteristics. The findings of this chapter provide further insight into the effectiveness of combining manifold curvature with tapering as a design strategy for enhancing the performance of manifold microchannel heat sinks used in advanced electronic cooling applications.

To systematically evaluate the influence of manifold tapering on the thermo-hydraulic performance of the proposed heat sink, two distinct design approaches, referred to as Group A

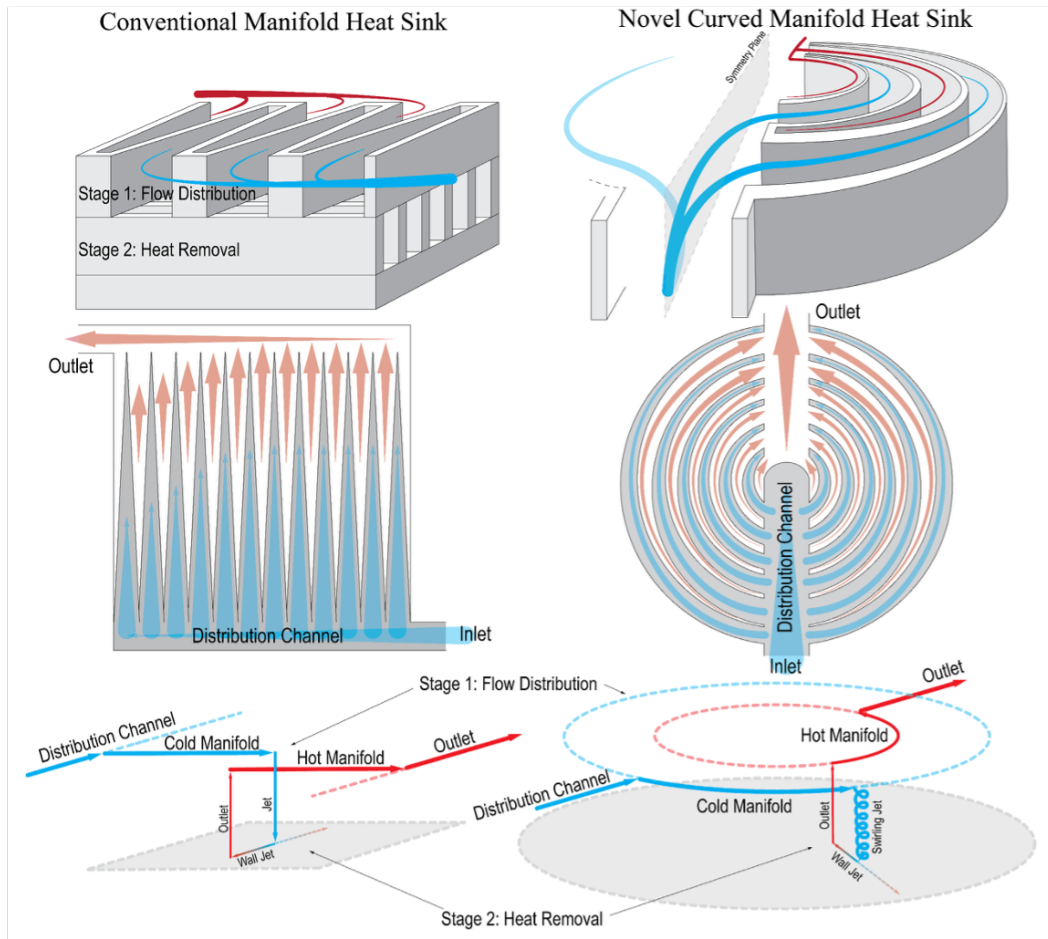


Figure 4.1: Conventional MMCHS along with the proposed tapered RMMCHS designs.

and Group B, were considered in this study. Each group consists of five geometrically unique unit-cell configurations, resulting in a total of ten tapered manifold designs. For every configuration in Group A, a geometrically corresponding counterpart exists in Group B, enabling a direct comparison of the effects of different tapering strategies under identical operating conditions.

In Group A, the cross-sectional dimensions at both the inlet and outlet of the manifolds remain constant for all configurations. The converging or diverging characteristics are introduced by modifying the dimensions at the opposite end of the manifold channel while maintaining fixed inlet and outlet hydraulic diameters. Consequently, any changes in flow behavior and heat transfer performance can be directly attributed to the variation in tapering geometry rather than to alterations in the entrance or exit conditions. This approach provides valuable insight into how gradual expansion or contraction within the manifold influences coolant acceleration, pressure distribution, and flow uniformity throughout the microchannel network.

In contrast, Group B follows the opposite design philosophy. For these configurations, the dimensions at the far ends of the manifold channels are maintained constant, while the inlet and outlet cross-sectional areas are systematically varied among the different cases. As a result, each Group B configuration possesses a different hydraulic diameter at the inlet and outlet of the computational domain. This design strategy enables the investigation of how modifying the entrance and exit flow conditions affects the development of the flow field, pressure losses, coolant distribution, and thermal performance within the heat sink. By comparing the results obtained from Groups A and B, a comprehensive understanding can be developed regarding the relative importance of manifold tapering location and its impact on the overall thermo-hydraulic characteristics of the proposed tapered radial manifold microchannel heat sink.

To further broaden the scope of the investigation and provide a more comprehensive assessment of the proposed tapered manifold concept, three additional reference configurations were examined for each design in Groups A and B. These supplementary models were introduced to isolate the effects of curvature, tapering, and manifold dimensions on the thermo-hydraulic performance of the heat sink.

The first additional configuration corresponds to a straight manifold design possessing the same geometric dimensions and operating conditions as its curved counterpart. By replacing the curved manifold with a straight channel while preserving all other geometric parameters, the direct influence of manifold curvature on flow development, pressure distribution, and heat transfer characteristics can be quantified. This comparison enables the evaluation of the benefits associated with curvature-induced secondary flows and enhanced mixing within the manifold channels.

The second configuration maintains the same tapering characteristics and geometric features as the original group members but employs a reduced overall height for the manifold section. Since manifold height directly affects the available flow area and hydraulic diameter, this modification allows the investigation of how a more compact manifold geometry influences pressure losses, coolant distribution, and thermal performance. The results provide valuable insight into the sensitivity of the proposed design to variations in manifold dimensions.

The third configuration serves as a baseline curved manifold model without tapering. In this case, both the inlet and outlet manifold cross-sectional areas remain constant throughout the entire computational domain, eliminating any converging or diverging effects. This model represents a conventional curved manifold geometry and provides a direct benchmark against which the performance of the tapered designs can be assessed. Comparing the tapered and non-tapered curved manifolds makes it possible to isolate the contribution of tapering to the overall thermo-hydraulic behavior of the system.

By incorporating these additional configurations into the analysis, a more complete understanding of the individual and combined effects of curvature, tapering, and manifold dimensions is obtained. This expanded study enables a rigorous evaluation of the proposed designs and facilitates the identification of the geometric features that contribute most significantly to enhanced cooling performance.

Figure 4.2 illustrates the geometric concept underlying the tapered curved manifold designs investigated in this chapter. The figure demonstrates the formation of the converging and

diverging manifold sections and defines the principal geometric parameters used to construct the two tapering strategies, Groups A and B. A comprehensive summary of the geometric characteristics of all investigated models, including their manifold configurations, tapering strategies, and corresponding geometric parameters, is presented in Table 4.1.

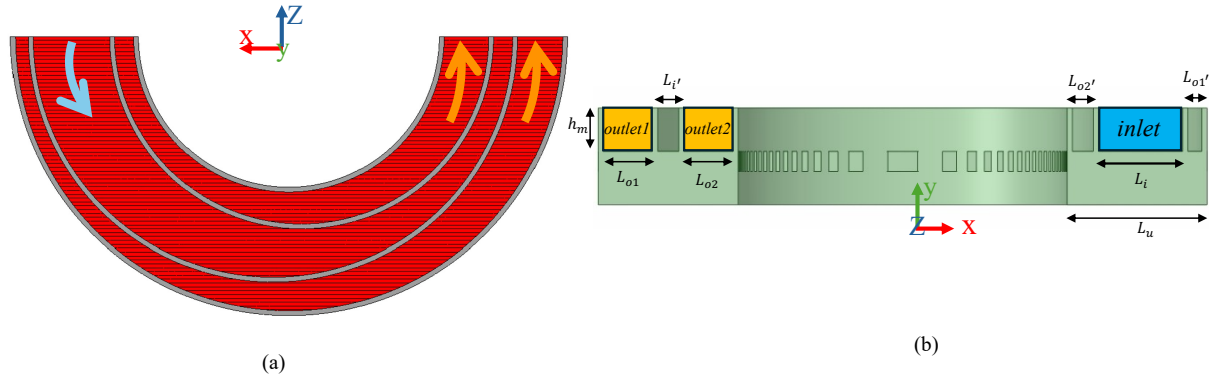


Figure 4.2: (a) Top view of the curved tapered manifold and (b) the unit cell.

Table 4.1: Geometric features of the original Group A and Group B tapered manifold models. All lengths are given in millimeters; a primed symbol denotes the dimension at the far end of the corresponding channel.

Group	Model	L_i	L'_i	L_{o1}	L_{o2}	L'_{o1}	L'_{o2}	L_u
A	A1	1.2	0.3	0.7	0.7	0.30	0.20	2.0
A	A2	1.2	0.4	0.7	0.7	0.35	0.25	2.1
A	A3	1.2	0.5	0.7	0.7	0.40	0.30	2.2
A	A4	1.2	0.6	0.7	0.7	0.45	0.35	2.3
A	A5	1.2	0.7	0.7	0.7	0.50	0.40	2.4
B	B1	0.3	0.15	0.80	0.75	0.70	0.70	2.0
B	B2	0.4	0.15	0.85	0.80	0.70	0.70	2.1
B	B3	0.5	0.15	0.90	0.85	0.70	0.70	2.2
B	B4	0.6	0.15	0.95	0.90	0.70	0.70	2.3
B	B5	0.7	0.15	1.00	0.95	0.70	0.70	2.4

Similar to Chapter 3, a conjugate heat transfer computational model was employed in the present study. The governing equations, modeling assumptions, numerical schemes, mesh

generation strategy, and convergence criteria remain identical to those described previously and are therefore not repeated here for brevity. Readers are referred to Chapter 3 for a detailed description of the mathematical formulation and numerical methodology. Furthermore, the temperature-dependent thermophysical properties of both the fluid and solid domains presented in Table 3.1 were adopted without modification and remain valid for all simulations performed in this chapter. Consequently, the present chapter focuses primarily on the geometric modifications associated with the tapered manifold configurations and their influence on the resulting flow and heat transfer characteristics.

4.2.2 Boundary Conditions

The computational domain is supplied with DI water as the coolant at a constant mass flux of $1000 \text{ kg}/(\text{m}^2 \cdot \text{s})$ and an inlet temperature of 23°C for all configurations investigated in this chapter. Consistent with the designs presented in the previous chapters, the coolant flows through the curved inlet manifold before being directed toward the microchannel region. This arrangement drives a swirling flow onto the heated substrate surface, thereby enhancing local convective heat transfer and improving the overall cooling effectiveness of the heat sink.

Unlike the designs investigated in the previous chapters, where a slot nozzle was incorporated into the inlet manifold to direct the coolant toward the microchannel region, the tapered manifold configurations do not employ a slot nozzle. Instead, the entire base of the converging inlet manifold serves as the flow delivery interface, allowing the working fluid to enter the microchannel section directly. This modification fundamentally alters the coolant distribution mechanism and enables the effects of manifold tapering on the thermo-hydraulic performance to be evaluated independently of nozzle-induced flow acceleration.

Following heat absorption from the substrate, the heated coolant is collected and discharged through two outlet manifold channels positioned on either side of the domain. Both outlets are prescribed with a pressure outlet boundary condition at zero gauge pressure to ensure physically realistic flow development and outflow behavior. A uniform heat flux of $100 \text{ W}/\text{cm}^2$ is

applied to the base of the substrate to represent the thermal load generated by the electronic device. To isolate the heat transfer processes occurring within the cooling system, all manifold walls are assumed to be adiabatic, preventing heat exchange with the surrounding environment. In contrast, the microchannel walls and substrate surfaces are defined using coupled boundary conditions, allowing for the simultaneous solution of heat conduction within the solid domain and convection within the fluid domain through the CHT framework. The dimensionless parameter β , defined in Equation 3.6, is maintained at a constant value of 5 throughout this study to maximize the available heat transfer surface area in the microchannel section.

4.2.3 Data Reduction

The data reduction methodology adopted in this chapter is identical to that presented in Chapter 3. Accordingly, the total thermal resistance (R_t), the mean absolute temperature deviation (MATD), and the average convective heat transfer coefficient (h) are calculated using Equation 3.7, Equation 3.8, and Equation 3.9, respectively. In addition, the pressure drop (ΔP) across the fluid domain is evaluated as the primary hydraulic performance parameter. These performance metrics are employed throughout this chapter to quantify and compare the thermo-hydraulic characteristics of the proposed tapered manifold configurations.

4.3 Results and Discussion

The results reveal two fundamentally different tapering mechanisms in Groups A and B. In Group A, the inlet and outlet boundary surfaces remain fixed for all five models; therefore, variations in pressure drop are primarily governed by the internal converging inlet and diverging outlet passages. As the design progresses from A1 to A5, the pressure drop decreases from 2491.29 Pa to 1915.36 Pa, corresponding to a reduction of approximately 23.1%, while the thermal resistance remains nearly unchanged at approximately 1.20 K/W. This indicates that the geometric modification in Group A effectively reduces hydraulic losses without sacrificing thermal performance.

In contrast, Group B follows an opposite tapering strategy, in which the far ends of the inlet and outlet channels remain fixed while the inlet and outlet boundary surfaces vary. As a result, the applied inlet and outlet areas change among B1–B5, directly affecting the effective flow entrance and discharge conditions. In this group, the thermal resistance decreases substantially from 2.75 K/W to 1.52 K/W, representing a 44.7% improvement, while the pressure drop increases from 601.19 Pa to 862.05 Pa, corresponding to a 43.4% increase. This behavior reflects a conventional thermo-hydraulic trade-off, where enhanced heat transfer is achieved at the expense of higher hydraulic resistance.

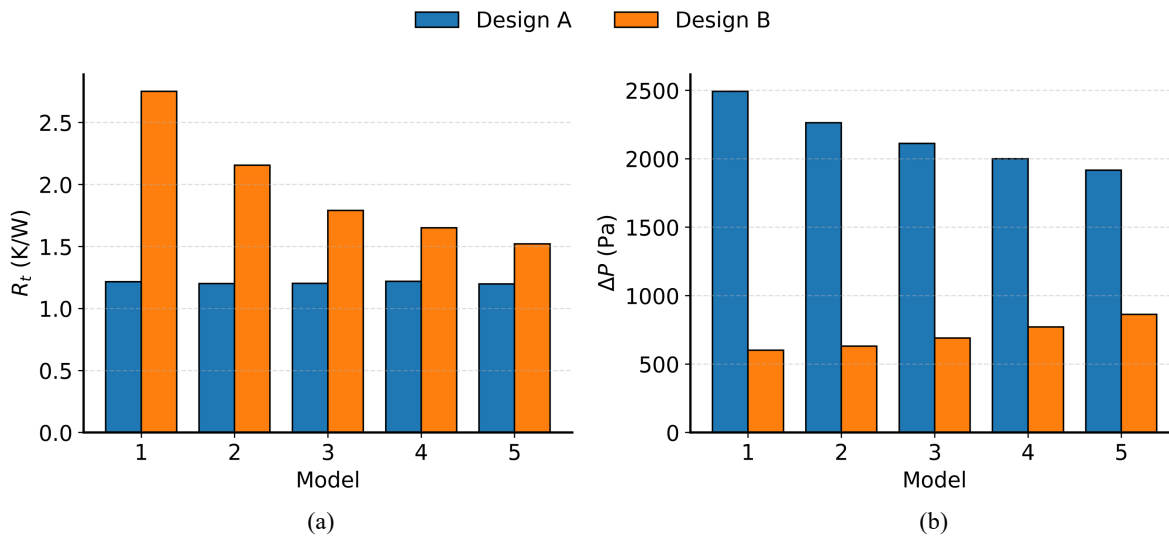


Figure 4.3: Direct comparison between Groups A and B: (a) thermal resistance and (b) pressure drop.

Figure 4.3 presents a direct comparison between corresponding cases. It shows that Group A consistently provides lower thermal resistance than Group B. For example, A1 reduces thermal resistance by approximately 55.8% compared with B1, while A5 still provides a 21.3% reduction relative to B5. However, this thermal advantage is accompanied by significantly higher pressure drops, with Group A requiring approximately 2.2 to 4.1 times greater pressure drop than Group B depending on the case. Therefore, Group A is more favorable when aggressive heat removal is prioritized, whereas Group B is more suitable when minimizing hydraulic losses and

pumping requirements is the dominant design objective.

The most important finding is that Group A achieves nearly constant thermal performance while progressively reducing pressure drop. This suggests that fixing the inlet and outlet boundary areas while modifying the internal tapering profile is an effective strategy for hydraulic optimization without degrading heat transfer. Conversely, Group B demonstrates that changing the inlet and outlet boundary areas strongly affects coolant delivery and thermal performance, but the improvement in heat removal is coupled with an increasing hydraulic penalty. Thus, the comparison confirms that the location at which tapering is introduced plays a critical role in determining the balance between pressure loss and thermal resistance in tapered curved manifold microchannel heat sinks.

The thermo-hydraulic performance of the Group B configurations and their corresponding reference models is presented in Figure 4.4 to evaluate the individual contributions of manifold tapering, curvature, and manifold height to the overall cooling effectiveness of the proposed heat sink.

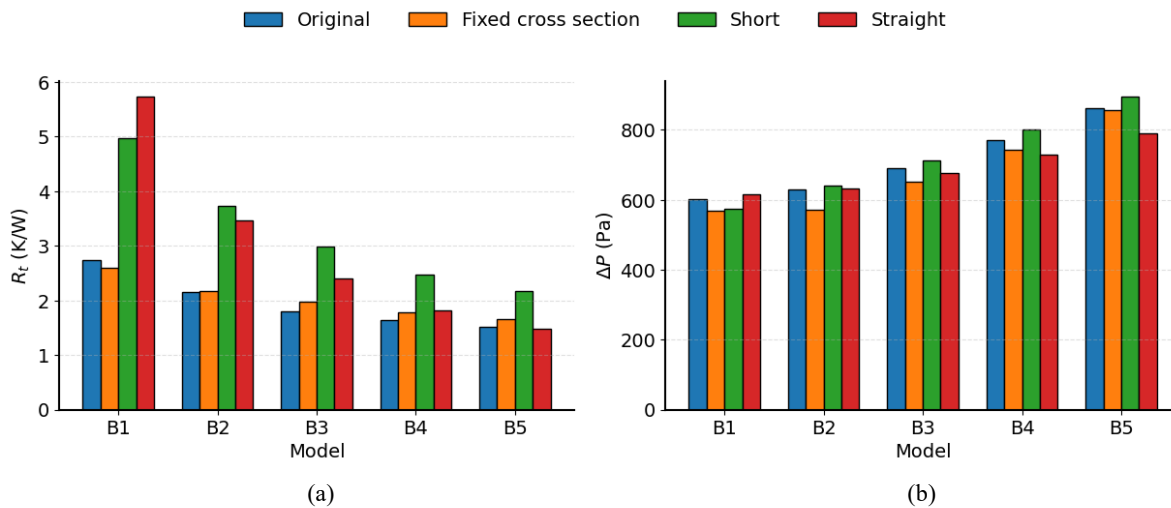


Figure 4.4: Thermo-hydraulic performance of the Group B configurations and their corresponding reference models: (a) thermal resistance and (b) pressure drop.

The original Group B behavior indicates that enlarging the inlet and outlet boundary dimensions progressively improves coolant access to the microchannel region, resulting in

stronger fluid penetration, more uniform coolant distribution, and enhanced convective heat transfer. However, these thermal improvements are accompanied by increased flow acceleration and redistribution within the manifold system, producing higher hydraulic losses. The results therefore demonstrate that Group B follows the conventional thermo-hydraulic relationship in which improvements in thermal performance require additional pumping power.

To isolate the effect of manifold curvature, the performance of the original Group B configurations was compared with geometrically equivalent straight manifold models. In general, the curved tapered manifolds exhibit lower thermal resistance than their straight counterparts for most investigated cases. This behavior can be attributed to the curvature-induced secondary flow structures generated within the manifold channels. As coolant travels through the curved passages, centrifugal forces create transverse pressure gradients that promote fluid mixing and redistribute momentum across the channel cross-section. These secondary motions enhance coolant renewal near the heated surfaces and reduce thermal boundary layer thickness within the microchannel region. Consequently, the curved manifold designs achieve superior heat transfer performance compared with straight channels possessing identical geometric dimensions. The differences become particularly noticeable for intermediate configurations where the balance between curvature-induced mixing and hydraulic losses is most favorable.

The influence of manifold height was investigated through a second set of reference models in which the tapered geometry was maintained while reducing the inlet and outlet manifold heights. These configurations consistently exhibit the highest thermal resistance among all studied cases. The deterioration in thermal performance can be explained by the reduction in available flow area within the manifold channels. Smaller manifold heights increase flow confinement and limit the ability of the coolant to redistribute uniformly before entering the microchannel section. As a result, regions of flow maldistribution become more pronounced, leading to localized reductions in coolant velocity and weaker heat transfer coefficients. Furthermore, the reduced hydraulic diameter intensifies viscous effects and promotes larger velocity gradients near the walls, which can further increase hydraulic losses without providing propor-

tional thermal benefits. The results therefore demonstrate that maintaining sufficient manifold height is essential for achieving effective coolant distribution and maximizing heat transfer performance in tapered manifold heat sinks.

A third comparison was conducted using curved manifold configurations without tapering. In these reference models, both the inlet and outlet manifold cross-sectional dimensions remain constant throughout the computational domain, thereby eliminating the converging and diverging characteristics present in the proposed design. Comparison between the tapered and non-tapered curved manifolds highlights the importance of tapering as a flow management mechanism. Although the performance differences are relatively small for some configurations, the tapered designs generally achieve lower thermal resistance, particularly for the larger geometric cases. The gradual expansion and contraction introduced by tapering modifies the local pressure distribution within the manifold and improves coolant delivery to the microchannel entrances. This enhanced flow distribution reduces thermal non-uniformities and promotes more effective utilization of the available coolant. Consequently, the tapered manifolds are able to achieve superior thermal performance without introducing excessive hydraulic penalties.

An important observation from the comparison of all Group B configurations is that no single geometric feature alone is responsible for the observed performance improvements. Instead, the thermal behavior results from the combined interaction of manifold curvature, tapering, and geometric dimensions. Curvature enhances mixing through secondary flow generation, tapering improves flow distribution and pressure management, and adequate manifold height ensures sufficient coolant transport capacity. When these features are properly integrated, substantial reductions in thermal resistance can be achieved while maintaining acceptable pressure losses. The original Group B designs successfully exploit this interaction, resulting in a progressive improvement in thermal performance from B1 to B5.

The results demonstrate that the proposed tapered curved manifold concept provides measurable thermo-hydraulic advantages compared with conventional straight, reduced-height, and non-tapered manifold configurations. The study confirms that careful control of manifold

geometry can significantly influence coolant distribution and heat transfer effectiveness. Furthermore, the findings emphasize that the location and implementation of tapering play a critical role in determining the balance between thermal enhancement and hydraulic resistance, providing valuable design guidelines for the development of next-generation manifold microchannel heat sinks for high-power electronic cooling applications.

The thermo-hydraulic performance of the Group A configurations and their corresponding reference models is examined in Figure 4.5. Unlike Group B, the observed performance variations originate primarily from modifications to the internal tapering geometry rather than from changes in the applied boundary conditions. This distinction makes Group A particularly valuable for isolating the influence of tapering on flow redistribution and heat transfer mechanisms within the manifold network.



Figure 4.5: Thermo-hydraulic performance of the Group A configurations and their corresponding reference models: (a) thermal resistance and (b) pressure drop.

The original Group A behavior is fundamentally different from that observed in Group B. Rather than following the conventional thermo-hydraulic trade-off, Group A achieves a significant reduction in hydraulic losses while preserving nearly identical thermal performance. Such behavior indicates that the tapering strategy successfully improves the internal pressure distribution and flow development without degrading coolant delivery to the microchannel region.

The reduced sensitivity of thermal resistance to geometric modifications suggests that the fixed inlet and outlet areas provide a stable coolant supply to the heat sink regardless of the internal tapering profile. Because the boundary conditions remain unchanged, the mass and momentum entering the manifold are nearly identical for all Group A cases. Therefore, the geometric modifications primarily influence how efficiently the coolant is transported through the manifold system rather than how much coolant reaches the cooling region. This explains why substantial reductions in pressure drop can be achieved while maintaining nearly constant heat transfer performance.

The influence of tapering becomes more apparent when the original Group A configurations are compared with the corresponding non-tapered curved reference models. For all investigated cases, the fixed cross-sectional manifolds exhibit noticeably higher thermal resistance than the tapered designs. For example, the thermal resistance increases from approximately 1.20 K/W in the tapered configurations to values ranging between 1.34 K/W and 1.66 K/W in the non-tapered cases. These results demonstrate that tapering promotes a more favorable pressure distribution within the manifold and improves coolant utilization throughout the microchannel array. The gradual expansion and contraction of the flow passages reduce local flow maldistribution and allow the coolant to interact more effectively with the heated surface.

The reduced-height manifold configurations exhibit the poorest thermal performance among all Group A reference models. Their thermal resistance ranges from approximately 1.56 K/W to 1.64 K/W, substantially exceeding the values obtained for the original tapered designs. The deterioration in performance can be attributed to the reduction in hydraulic diameter and available flow area within the manifold channels. Smaller manifold heights increase flow confinement and restrict coolant redistribution before entering the microchannel region. Consequently, local velocity non-uniformities become more severe, leading to reduced convective heat transfer coefficients and elevated wall temperatures. These findings highlight the importance of maintaining sufficient manifold volume to ensure effective coolant distribution.

A particularly important observation emerges from the comparison with the straight man-

ifold reference cases. In contrast to the behavior observed in Group B, the straight configurations consistently produce thermal resistance values that are equal to or lower than those of the original curved tapered designs. The straight manifold thermal resistance ranges from approximately 1.08 K/W to 1.21 K/W, representing the lowest values among all investigated Group A geometries. This result suggests that, under the fixed inlet and outlet boundary conditions employed in Group A, the additional pressure losses associated with curvature outweigh the benefits of curvature-induced secondary flow structures. Although curved channels can promote mixing and flow redistribution, they also introduce centrifugal effects and additional flow resistance. When the coolant supply conditions are already stabilized by fixed boundary areas, the simpler straight configuration becomes more effective at transporting coolant to the microchannels while minimizing hydraulic penalties.

A comparison between Groups A and B reveals fundamentally different thermo-hydraulic behaviors. Group B follows a traditional trade-off relationship, where thermal resistance decreases by approximately 45% while pressure drop increases by approximately 43%. In contrast, Group A achieves nearly constant thermal resistance while simultaneously reducing pressure drop by approximately 23%. Furthermore, the thermal resistance of the original Group A designs remains significantly lower than that of the original Group B configurations throughout the entire study. For example, the first configurations produce thermal resistances of approximately 1.22 K/W and 2.75 K/W for Groups A and B, respectively, while the final configurations yield approximately 1.20 K/W and 1.52 K/W. Although Group A incurs higher pressure drops, its superior thermal performance indicates that maintaining fixed inlet and outlet boundary areas is more effective for promoting heat transfer than modifying the boundary areas themselves.

These findings demonstrate that the location at which tapering is introduced plays a critical role in determining heat sink performance. Group A shows that internal tapering can be used to significantly reduce hydraulic losses without compromising thermal effectiveness when the inlet and outlet boundary conditions are preserved. Group B, however, indicates that altering the inlet and outlet areas strongly influences coolant delivery and results in a more conventional

thermo-hydraulic trade-off. Therefore, among the investigated approaches, the Group A design philosophy provides the most favorable balance between hydraulic optimization and thermal performance, while also offering greater robustness to geometric variations.

Figure 4.6 presents the mean absolute temperature deviation for the original Group A and Group B configurations. Since MATD directly measures the deviation of the maximum and minimum substrate temperatures from the average substrate temperature, lower values indicate more uniform temperature distributions and improved thermal management.

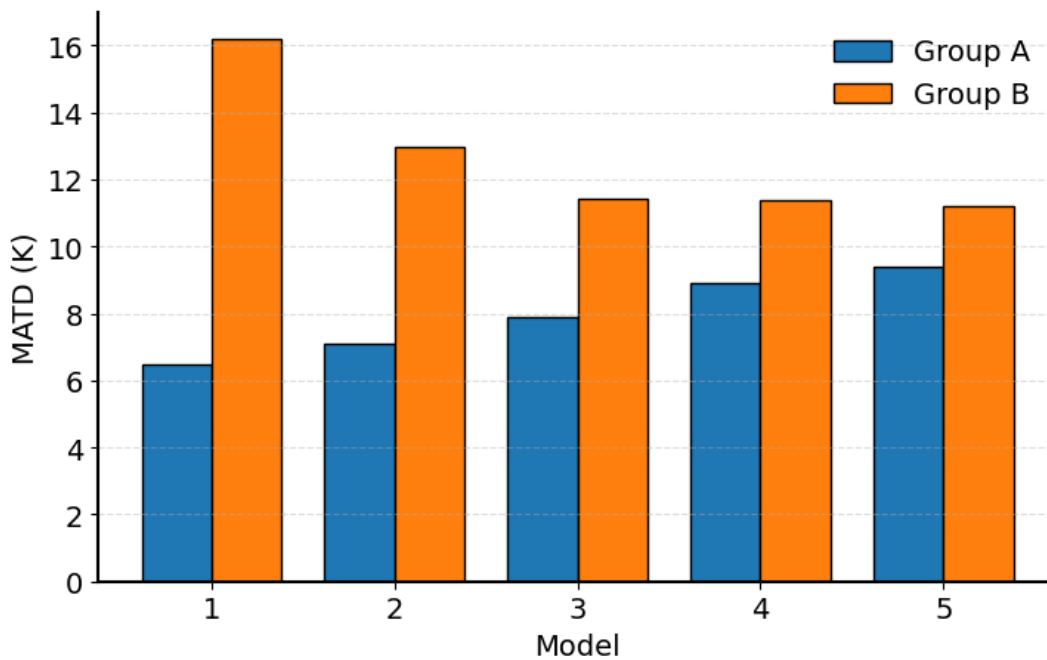


Figure 4.6: MATD study for both group designs.

The results reveal fundamentally different thermal uniformity characteristics between the two tapering strategies. For Group A, MATD increases monotonically from 6.47 K for A1 to 9.40 K for A5, corresponding to an increase of approximately 45.4%. This trend indicates that although the Group A configurations maintain nearly constant thermal resistance across all designs, the temperature distribution becomes progressively less uniform as the geometry evolves from A1 to A5. In other words, the ability of the heat sink to remove heat remains largely unchanged, but the spatial distribution of cooling becomes less effective. This observation demonstrates that

thermal resistance alone is insufficient for evaluating thermal performance, as two designs with nearly identical thermal resistance may exhibit significantly different hotspot characteristics.

In contrast, Group B exhibits the opposite behavior. The MATD decreases substantially from 16.18 K for B1 to 11.19 K for B5, representing a reduction of approximately 30.8%. This trend is consistent with the thermal resistance results previously obtained for Group B, where the thermal resistance decreased continuously from B1 to B5. As the inlet and outlet boundary areas increase, coolant distribution throughout the microchannel array becomes more effective, reducing localized temperature gradients and improving substrate temperature uniformity. Consequently, both thermal resistance and MATD improve simultaneously, indicating that the enhanced heat transfer performance is accompanied by more uniform cooling of the heated surface.

A direct comparison between the two groups reveals several important observations. First, all Group A configurations produce significantly lower MATD values than their corresponding Group B counterparts. For example, A1 exhibits a MATD of only 6.47 K, whereas B1 produces a MATD of 16.18 K. Even the least uniform Group A configuration (A5) achieves a MATD of 9.40 K, which remains approximately 16% lower than the best-performing Group B configuration (B5). These results indicate that preserving the inlet and outlet boundary dimensions, as implemented in Group A, promotes substantially more uniform coolant distribution throughout the heat sink than varying the inlet and outlet boundary areas as in Group B.

Second, the MATD results provide valuable insight into the physical mechanisms governing the two design philosophies. In Group A, the fixed inlet and outlet boundary conditions ensure relatively stable coolant delivery to the microchannel region. Consequently, the temperature field remains highly uniform despite the geometric modifications introduced through tapering. However, as the tapering becomes more pronounced from A1 to A5, local flow maldistribution gradually develops, causing a modest increase in MATD without significantly affecting the overall heat removal capability.

For Group B, the situation is markedly different. The smallest inlet and outlet bound-

ary areas in B1 severely restrict coolant access to portions of the microchannel array, creating large temperature gradients and substantial hotspot formation. As the inlet and outlet areas increase toward B5, coolant distribution improves considerably, reducing thermal non-uniformity and lowering MATD. Therefore, the decrease in MATD observed in Group B directly reflects the progressive improvement in coolant accessibility and flow distribution within the heat sink.

When the MATD and thermal resistance results are considered together, an important design trade-off emerges. Group A consistently provides superior temperature uniformity and lower thermal resistance than Group B, but at the expense of significantly higher pressure drops. Group B, on the other hand, achieves substantial reductions in pressure drop and gradually improves both thermal resistance and MATD, although it never reaches the temperature uniformity achieved by Group A. These findings suggest that Group A is more suitable for applications where hotspot suppression and thermal uniformity are critical design objectives, whereas Group B may be preferred when hydraulic performance and pumping power consumption are primary concerns.

The MATD results reinforce the conclusion that the location at which tapering is introduced plays a critical role in determining thermo-hydraulic performance. Maintaining fixed inlet and outlet boundary conditions, as implemented in Group A, produces substantially more uniform substrate temperatures and superior thermal management characteristics. Conversely, modifying the inlet and outlet boundary areas, as in Group B, leads to greater sensitivity of coolant distribution and thermal uniformity to geometric variations. The combined analysis of thermal resistance, pressure drop, and MATD therefore demonstrates that Group A offers the most favorable balance between heat removal capability and temperature uniformity among the investigated tapered manifold designs.

Figure 4.7 presents the temperature distribution on the heat sink base for all five configurations in Groups A and B. The left column, Figure 4.7(a1–a5), corresponds to Group A, while the right column, Figure 4.7(b1–b5), corresponds to Group B. These contours provide a direct visual explanation of the MATD trends discussed previously, since MATD quantifies the degree of

temperature non-uniformity across the heated base.

For Group A, the temperature field remains relatively uniform across all five configurations. Although MATD increases from A1 to A5, the contours show that the temperature gradients remain moderate and the high-temperature region does not expand severely. This indicates that fixing the inlet and outlet boundary areas in Group A helps preserve stable coolant delivery and suppresses strong localized hot spots. The gradual increase in MATD from A1 to A5 can be attributed to the progressive development of local flow maldistribution caused by the internal tapering variation; however, the thermal field remains considerably more uniform than that observed in Group B.

In contrast, Group B exhibits much stronger temperature non-uniformity, particularly in B1 and B2. The contours show a distinct high-temperature region concentrated near the lower-right sector of the heat sink base, indicating insufficient coolant renewal and weaker heat removal in that region. As the design progresses from B1 to B5, the hot spot becomes less severe and the temperature field becomes more evenly distributed, which is consistent with the reduction in MATD from 16.18 K to 11.19 K. This improvement suggests that increasing the inlet and outlet boundary areas in Group B enhances coolant accessibility and reduces local thermal accumulation.

A direct comparison between the two groups confirms that Group A provides superior temperature uniformity. Even though Group B improves significantly from B1 to B5, its temperature contours still show stronger spatial gradients than Group A. This agrees with the MATD data, where the highest MATD in Group A remains lower than the lowest MATD in Group B. Therefore, the base temperature distributions demonstrate that the Group A tapering strategy is more effective for hot-spot suppression and uniform thermal management, while Group B follows a more gradual improvement trend as the boundary-condition surfaces are enlarged.

These results further confirm that the location of tapering plays a critical role in the thermal behavior of tapered manifold heat sinks. When the inlet and outlet boundary surfaces are kept fixed, as in Group A, the coolant distribution remains more stable and the base temperature

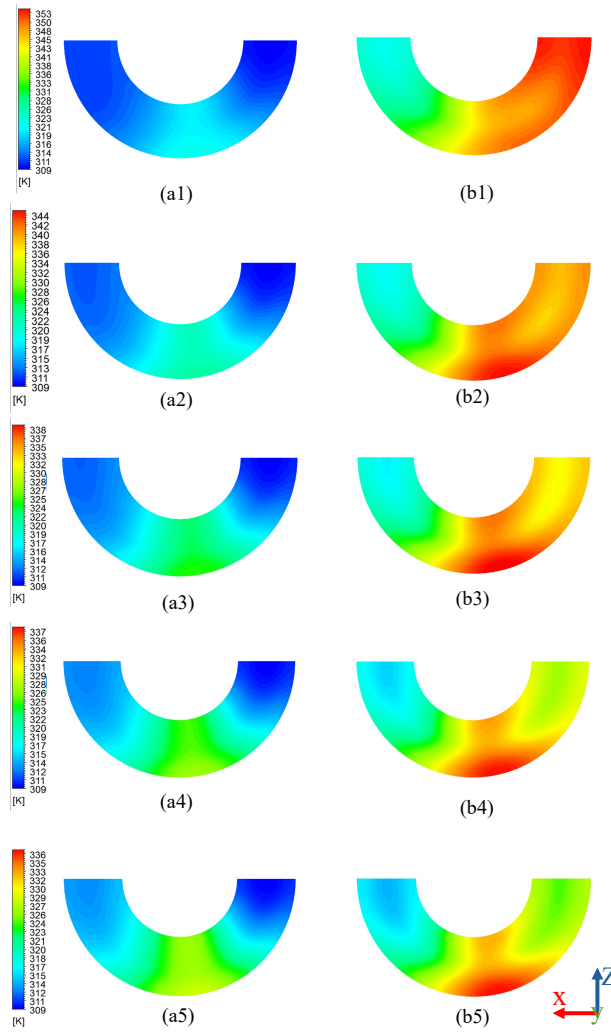


Figure 4.7: Temperature distribution on the heat sink base for all configurations: (a1–a5) Group A and (b1–b5) Group B.

field becomes more uniform. Conversely, when the boundary surfaces themselves are varied, as in Group B, thermal uniformity becomes more sensitive to geometric changes. Therefore, the combined MATD and temperature-contour analysis shows that Group A offers a more favorable thermal-uniformity performance, whereas Group B requires careful geometric optimization to reduce localized hot spots.

Figure 4.8 compares the average convective heat transfer coefficient for the original configurations of Groups A and B. Since the convective heat transfer coefficient directly represents the ability of the coolant to remove heat from the heated walls, higher values indicate more effective convective cooling and stronger fluid–solid thermal interaction.

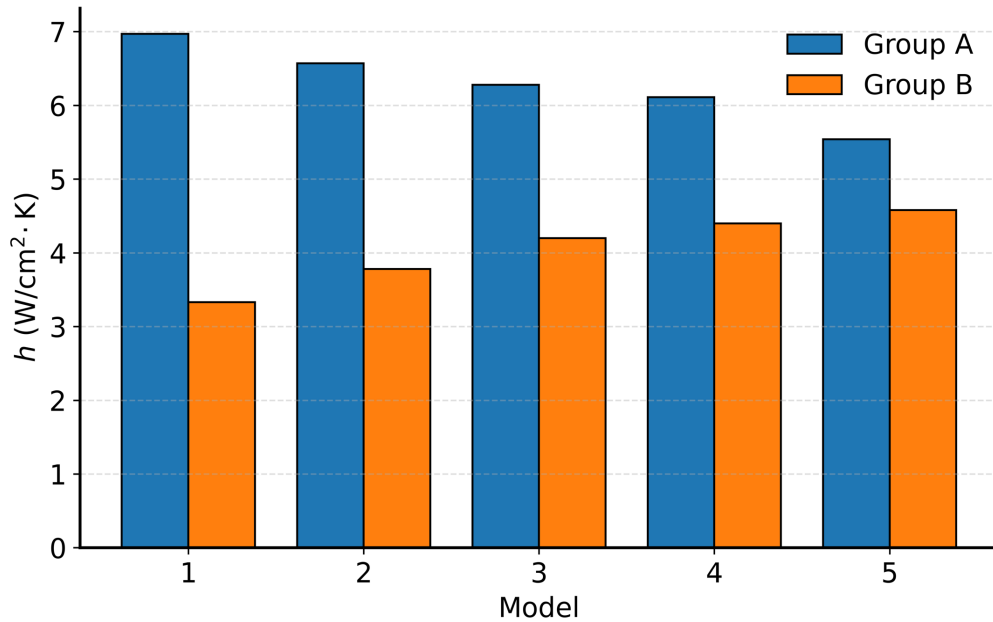


Figure 4.8: Heat transfer coefficient for both group designs.

The results reveal opposite trends for the two tapering strategies. In Group A, the average heat transfer coefficient gradually decreases from approximately 7.0×10^4 to 5.6×10^4 $W/(m^2 \cdot K)$ as the design progresses from A1 to A5, corresponding to an overall reduction of approximately 20%. This decrease indicates that increasing the degree of tapering while maintaining fixed inlet and outlet boundary dimensions weakens the local convective heat transfer. As the internal man-

ifold geometry changes, coolant velocity near portions of the microchannel entrance decreases and the thermal boundary layer becomes progressively thicker, resulting in a lower average heat transfer coefficient. Nevertheless, despite this reduction in h , the thermal resistance remains nearly unchanged because Group A maintains highly uniform coolant distribution throughout the microchannel array, as confirmed by the relatively low MATD values obtained for all five configurations.

Conversely, Group B exhibits a continuous increase in the average heat transfer coefficient, rising from approximately 3.3×10^4 to 4.6×10^4 W/(m² · K), representing an increase of nearly 37%. Unlike Group A, enlarging the inlet and outlet boundary areas progressively improves coolant accessibility to the manifold and microchannel region. The larger entrance area reduces flow restriction at the inlet, allowing a greater portion of the coolant to penetrate the microchannel array and enhancing fluid renewal along the heated surfaces. Consequently, stronger convective heat transfer is achieved, leading to the simultaneous reduction in thermal resistance and MATD observed previously.

A comparison between the two groups further highlights the influence of the tapering strategy on convective heat transfer. Every configuration in Group A exhibits a higher average heat transfer coefficient than its corresponding counterpart in Group B. For example, the heat transfer coefficient of A1 is more than twice that of B1, while even the lowest-performing configuration in Group A (A5) still exceeds the highest-performing Group B design (B5). These findings indicate that preserving the inlet and outlet boundary dimensions produces stronger local convection within the microchannels than varying the boundary areas themselves.

When the heat transfer coefficient is considered together with the thermal resistance, pressure drop, and MATD results, an important thermo-hydraulic relationship emerges. Group A maintains superior convective heat transfer and excellent temperature uniformity throughout all configurations; however, these advantages are accompanied by higher hydraulic losses because the fixed inlet and outlet dimensions sustain relatively high flow velocities and stronger momentum exchange within the manifold system. In contrast, Group B progressively improves

its convective heat transfer capability as the inlet and outlet areas increase, but the average heat transfer coefficient remains lower than that of Group A. Therefore, the improved thermal performance observed in Group B is primarily attributed to enhanced coolant distribution and increased wetted flow coverage rather than to exceptionally high local convection.

The heat transfer coefficient results reinforce the conclusions obtained from the previous performance metrics. Group A represents a design philosophy that emphasizes strong local convective heat transfer and excellent thermal uniformity through fixed boundary conditions, whereas Group B improves its thermal behavior by progressively enhancing coolant accessibility through enlarged inlet and outlet boundaries. The combined analysis demonstrates that the location at which tapering is introduced fundamentally governs the balance between convective heat transfer, hydraulic resistance, and temperature uniformity in tapered radial manifold microchannel heat sinks.

Figure 4.9 and Figure 4.10 present the velocity contours at the mid-plane of the heat sink for the original configurations of Groups A and B, respectively. These contours provide valuable insight into the influence of the two tapering strategies on coolant distribution within the manifold and microchannel sections and help explain the thermo-hydraulic behavior discussed in the previous sections.

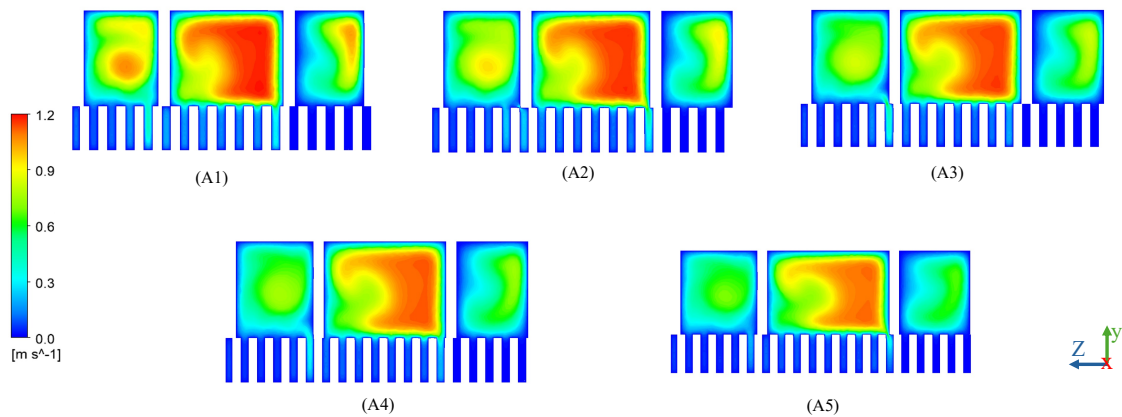


Figure 4.9: Velocity contours of the Group A designs.

For Group A (A1–A5), the coolant enters the converging inlet manifold through a constant

inlet cross-sectional area and is progressively accelerated as it approaches the microchannel region. Since the inlet and outlet boundary dimensions remain identical for all five configurations, the flow enters the heat sink under nearly the same inlet conditions, resulting in relatively consistent velocity distributions throughout the manifold. Although the internal tapering gradually changes from A1 to A5, the contours demonstrate that the coolant remains well distributed across the entire microchannel entrance, with no significant stagnant regions or severe flow maldistribution. The highest velocity region remains concentrated near the center of the converging manifold immediately upstream of the microchannels, while the velocity field inside the microchannel array remains relatively uniform. As the tapering increases toward A5, a slight reduction in the peak velocity can be observed, indicating a weaker local flow acceleration. This behavior explains the gradual decrease in the average convective heat transfer coefficient reported previously. Nevertheless, the overall coolant distribution remains highly uniform, which is consistent with the nearly constant thermal resistance and the relatively low MATD values observed for all Group A configurations.

In contrast, the velocity contours of Group B exhibit a significantly different flow evolution. For B1, the relatively small inlet boundary area produces a highly restricted entrance, generating strong local acceleration near the inlet while limiting coolant penetration into portions of the manifold. Consequently, the velocity distribution entering the microchannel region is highly non-uniform, with portions of the microchannel array receiving considerably less coolant than others. This flow maldistribution is directly responsible for the higher thermal resistance and larger MATD previously obtained for B1.

As the inlet and outlet boundary dimensions increase progressively from B1 to B5, the velocity field becomes increasingly uniform throughout the manifold and across the microchannel entrance. The enlarged inlet area reduces the entrance restriction and allows the coolant to spread more evenly before entering the microchannels. Consequently, the regions of relatively low velocity gradually diminish, while a more homogeneous velocity profile develops over the entire microchannel width. This improved coolant distribution explains the continuous reduction

in thermal resistance and MATD together with the increase in the average heat transfer coefficient observed from B1 to B5.

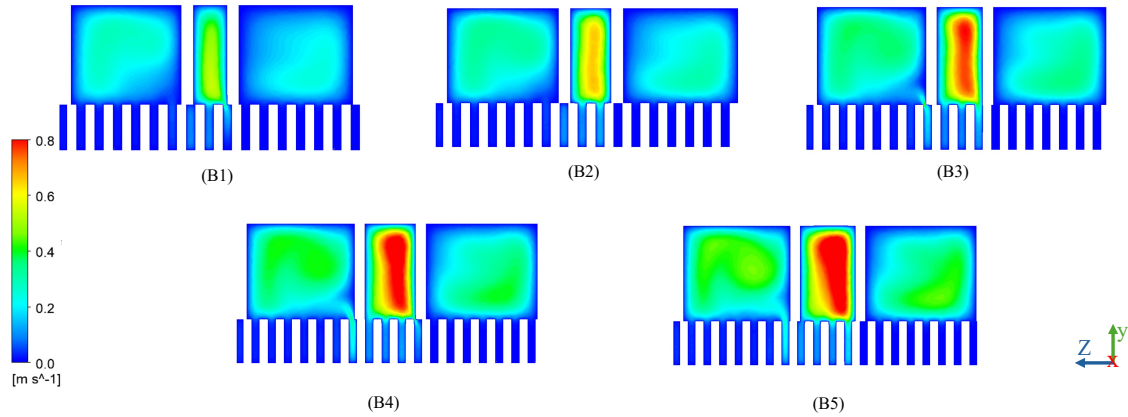


Figure 4.10: Velocity contours of the Group B designs.

A direct comparison between the two groups highlights the fundamentally different flow mechanisms governing the two tapering strategies. Group A maintains a relatively uniform velocity distribution for all five configurations because the inlet and outlet boundary conditions remain unchanged and only the internal manifold geometry is modified. As a result, the coolant is distributed uniformly throughout the microchannel array despite the gradual reduction in local flow acceleration. Conversely, Group B exhibits substantially greater sensitivity to geometric variation because the inlet and outlet boundary areas themselves are modified. The smallest boundary dimensions generate pronounced entrance restrictions and severe flow non-uniformity, whereas enlarging these boundaries progressively improves coolant accessibility and flow distribution within the manifold.

These velocity contours provide the physical explanation for the thermo-hydraulic trends obtained throughout this chapter. The superior temperature uniformity achieved by Group A is primarily attributed to its more homogeneous coolant distribution, whereas the higher pressure losses result from maintaining relatively high flow velocities through fixed inlet and outlet boundaries. On the other hand, the gradual improvement observed in Group B originates from progressively reducing entrance restrictions, allowing a more uniform velocity distribution to

develop before the coolant reaches the microchannel array. Therefore, the velocity field confirms that the location at which tapering is introduced fundamentally controls the coolant distribution mechanism and ultimately governs the pressure drop, thermal resistance, convective heat transfer coefficient, and temperature uniformity of the tapered radial manifold microchannel heat sink.

A subtle but important distinction deserves emphasis here. Groups A and B do not differ simply in velocity magnitude; they differ in how the flow develops before entering the microchannels. In Group A, the coolant has sufficient distance to redistribute within the converging manifold before entering the microchannel array, resulting in a relatively uniform entrance velocity profile. In Group B, and especially for B1 and B2, the restricted inlet boundary delays this redistribution, so the coolant enters the microchannels with a less uniform velocity profile.

Figure 4.11 presents the streamlines within the tapered curved manifold for the representative configurations A1, A5, B1, and B5. The streamlines provide additional insight into the coolant transport mechanism and complement the previously discussed velocity and temperature contour results. In all four configurations, the coolant follows a continuous flow path from the inlet manifold through the microchannel region toward the outlet manifolds without exhibiting large-scale flow separation or stagnant regions. This indicates that the proposed tapered manifold geometries maintain stable flow throughout the computational domain.

For Group A, both A1 and A5 exhibit relatively smooth streamline trajectories that gradually converge toward the microchannel section before being redistributed into the outlet manifolds. The flow remains well organized throughout the manifold, indicating that the constant inlet and outlet boundary dimensions provide stable coolant delivery regardless of the degree of tapering. Although A5 exhibits slightly weaker streamline convergence than A1, the overall flow pattern remains highly uniform, explaining why the thermal resistance changes only marginally despite the gradual reduction in the average convective heat transfer coefficient. The homogeneous streamline distribution is also consistent with the low MATD values observed for Group A, demonstrating effective coolant coverage over the heated substrate.

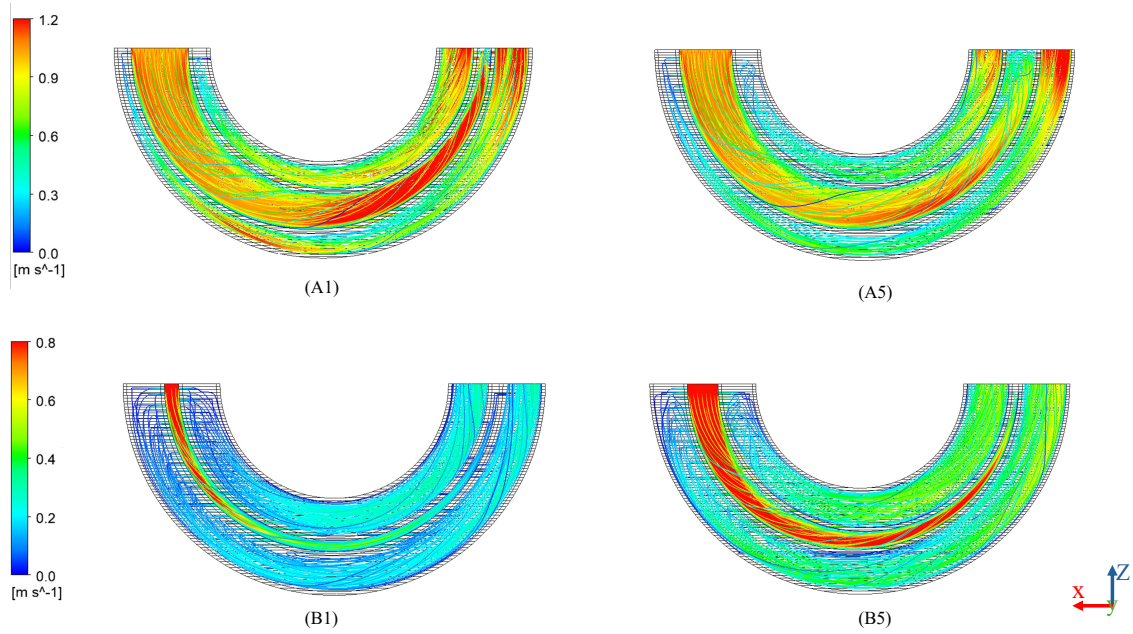


Figure 4.11: Velocity streamlines of both group designs.

The streamline patterns for Group B reveal a different flow evolution. In configuration B1, the coolant enters the converging manifold through the smallest inlet boundary area, producing a more concentrated flow path before reaching the microchannel region. The streamlines indicate that coolant redistribution within the manifold is less effective, resulting in a less uniform flow entering the microchannels. This behavior agrees with the previously reported high thermal resistance, elevated MATD, and relatively low heat transfer coefficient for B1.

As the geometry progresses to B5, the enlarged inlet boundary allows the coolant to spread more uniformly throughout the inlet manifold before entering the microchannel array. Consequently, the streamlines become more evenly distributed across the entire flow passage, indicating improved coolant accessibility to the heated region. This enhanced flow redistribution explains the simultaneous reduction in thermal resistance and MATD together with the increase in the average heat transfer coefficient previously observed for Group B.

A comparison between the two tapering strategies demonstrates that the principal difference is not the existence of secondary flow structures but rather the effectiveness of coolant redistribution within the inlet manifold prior to entering the microchannels. Group A maintains

a relatively uniform streamline distribution throughout the investigated configurations because the inlet and outlet boundary dimensions remain unchanged, whereas Group B exhibits progressively improved streamline uniformity as the inlet boundary area increases from B1 to B5. These observations provide a physical explanation for the thermo-hydraulic trends discussed throughout this chapter and further confirm that the location at which tapering is introduced governs the coolant distribution mechanism, ultimately influencing pressure drop, thermal resistance, heat transfer coefficient, and temperature uniformity.

4.4 Conclusions

- A conjugate heat transfer model incorporating separate fluid and solid domains with temperature-dependent thermophysical properties was employed to investigate the thermo-hydraulic performance of tapered curved radial manifold microchannel heat sinks. The numerical model enabled a comprehensive evaluation of thermal resistance, pressure drop, heat transfer coefficient, and temperature uniformity under realistic operating conditions.
- The tapering strategy significantly influenced the overall thermo-hydraulic behavior of the heat sink. In Group A, the total thermal resistance remained nearly constant at approximately 1.20 K/W despite a reduction of approximately 23% in pressure drop from A1 to A5. This demonstrates that appropriate internal tapering can effectively reduce hydraulic losses without degrading the overall cooling performance.
- Group B exhibited a conventional thermo-hydraulic trade-off. Increasing the inlet and outlet boundary dimensions reduced the thermal resistance from approximately 2.75 K/W to 1.52 K/W, corresponding to an improvement of nearly 45%, while the pressure drop increased from approximately 601 Pa to 862 Pa, representing an increase of about 43%. These results indicate that enlarging the flow entrance improves coolant accessibility to the microchannel region at the expense of higher hydraulic resistance.

- The temperature uniformity analysis further demonstrated the advantages of maintaining fixed inlet and outlet boundary conditions. Although the MATD of Group A increased moderately from 6.47 K to 9.40 K, all Group A configurations exhibited substantially lower MATD values than their Group B counterparts. In Group B, MATD decreased from 16.18 K to 11.19 K as coolant distribution improved, yet even the best-performing Group B configuration remained less thermally uniform than the least favorable Group A design.
- The average convective heat transfer coefficient further confirmed the different flow mechanisms governing the two tapering strategies. Group A maintained heat transfer coefficients between approximately 5.6×10^4 and 7.0×10^4 W/(m² · K), whereas Group B increased from approximately 3.3×10^4 to 4.6×10^4 W/(m² · K) as the inlet and outlet boundary areas increased. Combined with the velocity and temperature field analyses, these results demonstrate that the location of tapering governs coolant redistribution within the manifold and ultimately controls the balance between heat transfer enhancement, pressure drop, and temperature uniformity.

References

- [1] N. Jones et al., “How to stop data centres from gobbling up the world’s electricity,” *nature*, vol. 561, no. 7722, pp. 163–166, 2018.
- [2] A. H. Khalaj and S. K. Halgamuge, “A review on efficient thermal management of air-and liquid-cooled data centers: From chip to the cooling system,” *Applied energy*, vol. 205, pp. 1165–1188, 2017.
- [3] D. Huang, C. Yang, and B. Li, “Rack-level cooling and server-level cooling,” *Data center handbook: plan, design, build, and operations of a smart data center*, pp. 227–237, 2021.
- [4] R. Kong et al., “A comprehensive review of direct liquid cooling technologies,” *Energy*, 2024, ScienceDirect article (S0360544224026203).
- [5] V. E. Ahmadi et al., “Performance evaluation of two-phase direct-to-chip liquid cooling in retrofitted servers,” *Journal of Building Engineering*, 2026, ScienceDirect article (S2214157X26000122).
- [6] Uptime Institute, “Cooling systems survey 2024: Direct liquid cooling,” Uptime Institute Intelligence, 2024, Resource article and survey findings.
- [7] U.S. Department of Energy, “Best practices guide for energy-efficient data center design,” U.S. Department of Energy, 2024, Guidance covering air, cold plate, and immersion cooling.
- [8] R. Zhao, Y. Du, X. Yang, Z. Zhou, W. Wang, and X. Yang, “A critical review on the thermal management of data center for local hotspot elimination,” *Energy and Buildings*, vol. 297, p. 113 486, 2023.
- [9] ASHRAE, “Emergence and expansion of liquid cooling in mainstream data centers,” ASHRAE, 2021, White paper discussing drivers and implementation considerations.
- [10] J. Cho, “Numerical analysis of rack-based data center cooling with rear door heat exchanger (rdhx): Interrelationship between thermal performance and energy efficiency,” *Case Studies in Thermal Engineering*, vol. 63, p. 105 247, 2024.
- [11] M. Azarifar, M. Arik, and J.-Y. Chang, “Liquid cooling of data centers: A necessity facing challenges,” *Applied Thermal Engineering*, vol. 247, p. 123 112, 2024.
- [12] S. Rangarajan, S. N. Schiffres, and B. Sammakia, “A review of recent developments in “on-chip” embedded cooling technologies for heterogeneous integrated applications,” *Engineering*, vol. 26, pp. 185–197, 2023.
- [13] V. Shalom Simon, H. Modi, K. B. Sivaraju, P. Bansode, S. Saini, P. Shahi, S. Karajgikar, V. Mulay, and D. Agonafer, “Feasibility study of rear door heat exchanger for a high capacity

- data center,” in *International electronic packaging technical conference and exhibition*, American Society of Mechanical Engineers, vol. 86557, 2022, V001T01A018.
- [14] R. Kumar and S. Jain, “Performance analysis of an in-situ data centre,” *Energy and Buildings*, vol. 209, p. 109 679, 2020.
 - [15] T. Gao, *Experimental, Analytical and Numerical Investigation of the Transient Thermal Management of Hybrid and Liquid Cooled Data Centers*. State University of New York at Binghamton, 2015.
 - [16] D. W. Demetriou, “The ashrae thermal guidelines for data centers: Past, present and future,” ASHRAE, 2025, Presentation summarizing guideline evolution and resiliency considerations.
 - [17] Open Compute Project, “Oai system liquid cooling guidelines,” Open Compute Project, 2023, Guidelines on design, validation, and implementation in OCP environments.
 - [18] Schneider Electric, *Choosing the right liquid cooling architecture*, Accessed: 2026-05-27, 2026. [Online]. Available: https://download.schneider-electric.com/files?p_enDocType=Brochure&p_File_Name=Choosing+the+right+liquid+cooling+fv.pdf&p_Doc_Ref=Liquid-cooling-architecture.
 - [19] Open Compute Project, “Liquid cooling integration and logistics white paper,” Open Compute Project, 2021, Guidance on integration logistics and liquid heat capture fractions.
 - [20] W. Zhou, K. Dong, Q. Sun, W. Luo, B. Zhang, S. Guan, and G. Wang, “Research progress of the liquid cold plate cooling technology for server electronic chips: A review,” *International Journal of Energy Research*, vol. 46, no. 9, pp. 11 574–11 595, 2022.
 - [21] N. Gilmore, V. Timchenko, and C. Menictas, “Manifold microchannel heat sink topology optimisation,” *International Journal of Heat and Mass Transfer*, vol. 170, p. 121 025, 2021.
 - [22] Y. Pan, R. Zhao, Y. Nian, and W. Cheng, “Study on the flow and heat transfer characteristics of pin-fin manifold microchannel heat sink,” *International Journal of Heat and Mass Transfer*, vol. 183, p. 122 052, 2022.
 - [23] O. Osman, R. El-Zoheiry, M. Elsharnoby, and S. Nada, “Performance enhancement and comprehensive experimental comparative study of cold plate cooling of electronic servers using different configurations of mini-channels flow,” *Alexandria Engineering Journal*, vol. 60, no. 5, pp. 4451–4459, 2021.
 - [24] M. Stahlhut et al., “Data centers with direct liquid-cooled servers: Energy demand reduction and waste heat reuse opportunities,” *Energy Science & Engineering*, 2025.
 - [25] A. Bar-Cohen, M. Arik, and M. Ohadi, “Direct liquid cooling of high flux micro and nano electronic components,” *Proceedings of the IEEE*, vol. 94, no. 8, pp. 1549–1570, 2006.

- [26] Y. Wang, C. Wen, Q. Huang, X. Kang, M. Chen, and H. Wang, "Performance comparison between ethanol phase-change immersion and active water cooling for solar cells in high concentrating photovoltaic system," *Energy Conversion and Management*, vol. 149, pp. 505–513, 2017.
- [27] C. Roe, X. Feng, G. White, R. Li, H. Wang, X. Rui, C. Li, F. Zhang, V. Null, M. Parkes, et al., "Immersion cooling for lithium-ion batteries—a review," *Journal of Power Sources*, vol. 525, p. 231 094, 2022.
- [28] Y. Zhang, S. L. Ho, and W. Fu, "Applying response surface method to oil-immersed transformer cooling system for design optimization," *IEEE Transactions on Magnetics*, vol. 54, no. 11, pp. 1–5, 2018.
- [29] O. Sevgin, *Multi-liquid heat transfer*, US Patent 3,406,244, Oct. 1968.
- [30] R. C. Chu and J. H. Seely, *Immersion cooling system for modularly packaged components*, US Patent 3,512,582, May 1970.
- [31] N. A. Pambudi, A. Sarifudin, R. A. Firdaus, D. K. Ulfa, I. M. Gandidi, and R. Romadhon, "The immersion cooling technology: Current and future development in energy saving," *Alexandria Engineering Journal*, vol. 61, no. 12, pp. 9509–9527, 2022.
- [32] P. E. Tuma, "The merits of open bath immersion cooling of datacom equipment," in *2010 26th Annual IEEE Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM)*, IEEE, 2010, pp. 123–131.
- [33] P. E. Tuma, "Design considerations relating to non-thermal aspects of passive 2-phase immersion cooling," in *2011 27th annual IEEE semiconductor thermal measurement and management symposium*, IEEE, 2011, pp. 1–9.
- [34] GIGABYTE Technology, *Two-phase and single-phase liquid immersion cooling*, Published May 18, 2022. Accessed: May 31, 2026, May 2022. [Online]. Available: <https://www.gigabyte.com/Article/two-phase-and-single-phase-liquid-immersion-cooling>.
- [35] A. Bar-Cohen, M. Arik, and M. Ohadi, "Direct liquid cooling of high flux micro and nano electronic components," *Proceedings of the IEEE*, vol. 94, no. 8, pp. 1549–1570, 2006.
- [36] M. Asadi, G. Xie, and B. Sunden, "A review of heat transfer and pressure drop characteristics of single and two-phase microchannels," *International Journal of Heat and Mass Transfer*, vol. 79, pp. 34–53, 2014.
- [37] S. A. Nisar and D. Banerjee, "Immersion cooling in data centers: A comprehensive review of benefits, challenges and future directions," *Preprint / review*, 2025, Publicly available review (ResearchGate-hosted copy).

- [38] Uptime Institute Intelligence, *Immersion cooling evolves in response to IT power density*, Industry analysis discussing immersion and direct liquid cooling adoption, 2024.
- [39] D. B. Tuckerman and R. F. W. Pease, "High-performance heat sinking for vlsi," *IEEE Electron device letters*, vol. 2, no. 5, pp. 126–129, 2005.
- [40] K. Tang, G. Lin, Y. Guo, J. Huang, H. Zhang, and J. Miao, "Simulation and optimization of thermal performance in diverging/converging manifold microchannel heat sink," *International Journal of Heat and Mass Transfer*, vol. 200, p. 123 495, 2023.
- [41] M. Dehghan, M. Daneshipour, M. S. Valipour, R. Rafee, and S. Saedodin, "Enhancing heat transfer in microchannel heat sinks using converging flow passages," *Energy Conversion and Management*, vol. 92, pp. 244–250, 2015.
- [42] O. A. Akbari, E. Khodabandeh, F. Kahbandeh, D. Toghraie, and M. Khalili, "Numerical investigation of heat transfer of nanofluid flow through a microchannel with heat sinks and sinusoidal cavities by using novel nozzle structure," *Journal of Thermal Analysis and Calorimetry*, vol. 138, no. 1, pp. 737–752, 2019.
- [43] M. Eisapour, A. H. Eisapour, M. Hosseini, and P. Talebizadehsardari, "Exergy and energy analysis of wavy tubes photovoltaic-thermal systems using microencapsulated pcm nano-slurry coolant fluid," *Applied Energy*, vol. 266, p. 114 849, 2020.
- [44] A. Radwan, S. Ookawara, and M. Ahmed, "Thermal management of concentrator photovoltaic systems using two-phase flow boiling in double-layer microchannel heat sinks," *Applied Energy*, vol. 241, pp. 404–419, 2019.
- [45] Y. Zhang, Q. Lv, H. Wang, S. Zhao, Q. Xiong, R. Lv, and X. Zhang, "Simultaneous electrical and thermal rectification in a monolayer lateral heterojunction," *Science*, vol. 378, no. 6616, pp. 169–175, 2022.
- [46] H. M. Ali and W. Arshad, "Thermal performance investigation of staggered and inline pin fin heat sinks using water based rutile and anatase tio2 nanofluids," *Energy conversion and management*, vol. 106, pp. 793–803, 2015.
- [47] Y.-H. Pan, R. Zhao, Y.-L. Nian, and W.-L. Cheng, "Numerical study on heat transfer characteristics of a pin–fin staggered manifold microchannel heat sink," *Applied Thermal Engineering*, vol. 219, p. 119 436, 2023.
- [48] M. A. S. Al-Baghdadi, Z. M. Noor, A. Zeiny, A. Burns, and D. Wen, "Cfd analysis of a nanofluid-based microchannel heat sink," *Thermal Science and Engineering Progress*, vol. 20, p. 100 685, 2020.
- [49] S. Kumar, A. Kumar, A. D. Kothiyal, and M. S. Bisht, "A review of flow and heat transfer behaviour of nanofluids in micro channel heat sinks," *Thermal Science and Engineering Progress*, vol. 8, pp. 477–493, 2018.

- [50] K. P. Drummond, D. Back, M. D. Sinanis, D. B. Janes, D. Peroulis, J. A. Weibel, and S. V. Garimella, "A hierarchical manifold microchannel heat sink array for high-heat-flux two-phase cooling of electronics," *International Journal of Heat and Mass Transfer*, vol. 117, pp. 319–330, 2018.
- [51] K. W. Jung, C. R. Kharangate, H. Lee, J. Palko, F. Zhou, M. Asheghi, E. M. Dede, and K. E. Goodson, "Embedded cooling with 3d manifold for vehicle power electronics application: Single-phase thermal-fluid performance," *International Journal of Heat and Mass Transfer*, vol. 130, pp. 1108–1119, 2019.
- [52] G. M. Harpole and J. E. Eninger, "Micro-channel heat exchanger optimization," in *Proceeding of the 7th IEEE SEMI-THERM Symposium*, 1991, pp. 59–63.
- [53] N. Gilmore, V. Timchenko, and C. Menictas, "Manifold microchannel heat sink topology optimisation," *International Journal of Heat and Mass Transfer*, vol. 170, p. 121 025, 2021.
- [54] E. Kermani, S. Dessiatoun, A. Shooshtari, and M. M. Ohadi, "Experimental investigation of heat transfer performance of a manifold microchannel heat sink for cooling of concentrated solar cells," in *2009 59th Electronic Components and Technology Conference*, IEEE, 2009, pp. 453–459.
- [55] Y. Kim, D. Kong, R. D. Selvakumar, M. Kang, N. Kang, J. Kwon, and H. Lee, "Thermal-hydraulic characterization in manifold-microchannel heat sinks for energy-efficient cooling of hev/ev power modules," *Applied Thermal Engineering*, vol. 265, p. 125 611, 2025.
- [56] F. Zhou, Y. Liu, Y. Liu, S. N. Joshi, and E. M. Dede, "Modular design for a single-phase manifold mini/microchannel cold plate," *Journal of Thermal Science and Engineering Applications*, vol. 8, no. 2, p. 021 010, 2016.
- [57] H. Yang, Z. Wang, M. Li, F. Ren, and Y. Feng, "A manifold channel liquid cooling system with low-cost and high temperature uniformity for lithium-ion battery pack thermal management," *Thermal Science and Engineering Progress*, vol. 41, p. 101 857, 2023.
- [58] D. Boyea, A. Shooshtari, S. V. Dessiatoun, and M. M. Ohadi, "Heat transfer and pressure drop characteristics of a liquid cooled manifold-microgroove condenser," in *Heat Transfer Summer Conference*, American Society of Mechanical Engineers, vol. 55492, 2013, V003T23A003.
- [59] Y. Qiu, H. Wang, J. Carter, J. McGuffin-Cawley, and C. R. Kharangate, "Experimental investigation of heat transfer and pressure drop in copper manifold microchannel heat sinks," *Applied Thermal Engineering*, vol. 255, p. 124 024, 2024.
- [60] Z. Tang, R. Sun, K. Lu, J. Cheng, and P. Zhou, "Study on the flow and heat transfer performance of nanofluid in a novel manifold microchannel heat sink combining trapezoidal cross-section and curved corners," *Journal of Thermal Analysis and Calorimetry*, vol. 149, no. 15, pp. 8597–8615, 2024.

- [61] C. Xiao, M. Nourbakhsh, A. Alizadeh, D. Toghraie, P. Barnoon, and A. Khan, "Investigation of thermal behavior and performance of different microchannels: A case study for traditional and manifold microchannels," *Case Studies in Thermal Engineering*, vol. 39, p. 102 393, 2022.
- [62] J. Cheng, H. Xu, Z. Tang, and P. Zhou, "Multi-objective optimization of manifold microchannel heat sink with corrugated bottom impacted by nanofluid jet," *International Journal of Heat and Mass Transfer*, vol. 201, p. 123 634, 2023.
- [63] Y. Yue, S. K. Mohammadian, and Y. Zhang, "Analysis of performances of a manifold microchannel heat sink with nanofluids," *International Journal of Thermal Sciences*, vol. 89, pp. 305–313, 2015.
- [64] W. Escher, B. Michel, and D. Poulikakos, "A novel high performance, ultra thin heat sink for electronics," *International Journal of Heat and Fluid Flow*, vol. 31, no. 4, pp. 586–598, 2010.
- [65] W. Escher, T. Brunschweiler, B. Michel, and D. Poulikakos, "Experimental investigation of an ultrathin manifold microchannel heat sink for liquid-cooled chips," 2010.
- [66] L. Boteler, N. Jankowski, P. McCluskey, and B. Morgan, "Numerical investigation and sensitivity analysis of manifold microchannel coolers," *International Journal of Heat and Mass Transfer*, vol. 55, no. 25-26, pp. 7698–7708, 2012.
- [67] J.-B. Li, T.-Y. Zhang, Z.-D. Li, L. Chen, and W.-Q. Tao, "Multi-objective parameter optimization design of tapered-type manifold/variable cross-section microchannel heat sink," *Applied Thermal Engineering*, vol. 251, p. 123 587, 2024.
- [68] K. Tang, G. Lin, Y. Guo, J. Huang, H. Zhang, J. Miao, and H. Wang, "Thermal-hydraulic characterization of manifold microchannel heat sink with diverging channels and uniform heating," *Thermal Science and Engineering Progress*, vol. 46, p. 102 235, 2023.
- [69] X. Pu, Z. Zhao, M. Sun, and Y. Huang, "Numerical study on temperature distribution uniformity and cooling performance of manifold microchannel heat sink," *Applied Thermal Engineering*, vol. 237, p. 121 779, 2024.
- [70] Y. Luo, J. Li, K. Zhou, J. Zhang, and W. Li, "A numerical study of subcooled flow boiling in a manifold microchannel heat sink with varying inlet-to-outlet width ratio," *International Journal of Heat and Mass Transfer*, vol. 139, pp. 554–563, 2019.
- [71] K. W. Jung, C. R. Kharangate, H. Lee, J. Palko, F. Zhou, M. Asheghi, E. M. Dede, and K. E. Goodson, "Embedded cooling with 3d manifold for vehicle power electronics application: Single-phase thermal-fluid performance," *International Journal of Heat and Mass Transfer*, vol. 130, pp. 1108–1119, 2019.
- [72] T. Wei, S. Hazra, Y. Lin, M. P. Gupta, M. Degner, M. Asheghi, and K. E. Goodson, "Numerical study of large footprint (24× 24 mm²) silicon-based embedded microchannel three-

- dimensional manifold coolers,” *Journal of Electronic Packaging*, vol. 145, no. 2, p. 021 008, 2023.
- [73] N. Gilmore, A. Hassanzadeh-Barforoushi, V. Timchenko, and C. Menictas, “Manifold configurations for uniform flow via topology optimisation and flow visualisation,” *Applied Thermal Engineering*, vol. 183, p. 116 227, 2021.
 - [74] D. Kong, E. Jung, Y. Kim, V. V. Manepalli, K. J. Rah, H. S. Kim, Y. Hong, H. G. Choi, D. Agonafer, and H. Lee, “An additively manufactured manifold-microchannel heat sink for high-heat flux cooling,” *International Journal of Mechanical Sciences*, vol. 248, p. 108 228, 2023.
 - [75] X. Wei, Y. Feng, T. Yan, W. Li, Q. Lv, and J. Qin, “Optimization of manifold microchannel heat sink based on design of experiment method,” *ASME Journal of Heat and Mass Transfer*, vol. 146, no. 8, 2024.
 - [76] C. Chen, Y. Liu, J. Li, D. Shi, and G. Xin, “Flow and heat transfer characteristics of manifold microchannels with different microchannel arrangements,” *Thermal Science and Engineering Progress*, p. 103 354, 2025.
 - [77] H. Kang, X. Mei, K. Xu, and J. Cui, “Investigation of heat transfer performance of the manifold microchannel heat sink with different interface configurations,” *International Communications in Heat and Mass Transfer*, vol. 159, p. 107 807, 2024.
 - [78] Y.-H. Pan, R. Zhao, Y.-L. Nian, and W.-L. Cheng, “Numerical study on heat transfer characteristics of a pin-fin staggered manifold microchannel heat sink,” *Applied Thermal Engineering*, vol. 219, p. 119 436, 2023.
 - [79] Y.-H. Pan, R. Zhao, X.-H. Fan, Y.-L. Nian, and W.-L. Cheng, “Study on the effect of varying channel aspect ratio on heat transfer performance of manifold microchannel heat sink,” *International Journal of Heat and Mass Transfer*, vol. 163, p. 120 461, 2020.
 - [80] Y. Pan, R. Zhao, Y. Nian, and W. Cheng, “Study on the flow and heat transfer characteristics of pin-fin manifold microchannel heat sink,” *International Journal of Heat and Mass Transfer*, vol. 183, p. 122 052, 2022.
 - [81] J. Yang, K. Cheng, K. Zhang, C. Huang, and X. Huai, “Numerical study on thermal and hydraulic performances of a hybrid manifold microchannel with bifurcations for electronics cooling,” *Applied Thermal Engineering*, vol. 232, p. 121 099, 2023.
 - [82] M. Yang and B.-Y. Cao, “Numerical study on flow and heat transfer of a hybrid microchannel cooling scheme using manifold arrangement and secondary channels,” *Applied Thermal Engineering*, vol. 159, p. 113 896, 2019.
 - [83] S. Chen, S. Li, and Z.-Z. Hu, “Fluid flow and heat transfer characteristics of manifold microchannel heat sinks with ribs of different shapes,” *International Journal of Heat and Mass Transfer*, vol. 251, p. 127 300, 2025.

- [84] F. Kahbandeh, M. Azarifar, M. Arik, and D. K. Harris, "Numerical simulation of thermal performance of radial manifold microchannel heat sink for electronics thermal management," *Thermal Science and Engineering Progress*, p. 103 930, 2025.
- [85] M. Li, W. Liu, L. Yang, P. Chen, and C. Chen, "Chip temperature optimization for dark silicon many-core systems," *IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems*, vol. 37, no. 5, pp. 941–953, 2017.
- [86] W. Wagner and H.-J. Kretzschmar, "Iapws industrial formulation 1997 for the thermodynamic properties of water and steam," *International steam tables: properties of water and steam based on the industrial formulation IAPWS-IF97*, pp. 7–150, 2008.
- [87] H. Shanks, P. Maycock, P. Sidles, and G. Danielson, "Thermal conductivity of silicon from 300 to 1400 k," *Physical Review*, vol. 130, no. 5, p. 1743, 1963.
- [88] D. Ansari and K.-Y. Kim, "Hotspot thermal management using a microchannel-pinfin hybrid heat sink," *International Journal of Thermal Sciences*, vol. 134, pp. 27–39, 2018.
- [89] D. Ansari and J. H. Jeong, "A silicon-diamond microchannel heat sink for die-level hotspot thermal management," *Applied Thermal Engineering*, vol. 194, p. 117 131, 2021.
- [90] Z. Wan and Y. Joshi, "Pressure drop and heat transfer characteristics of pin fin enhanced microgaps in single phase microfluidic cooling," *International Journal of Heat and Mass Transfer*, vol. 115, pp. 115–126, 2017.
- [91] X. Liu, Y. Fu, J. Wang, and J. Ge, "Dean instability and thermal characteristic in sinusoidal structure," *International Journal of Heat and Mass Transfer*, vol. 206, p. 123 938, 2023.
- [92] S. Hussen, R. K. Chanda, M. S. Alam, and R. N. Mondal, "Flow instability and development of dean vortices in a bending rectangular channel with bottom wall heating and cooling from the ceiling," *International Journal of Thermofluids*, vol. 26, p. 101 049, 2025.
- [93] C. Guigui, P. Manno, P. Moulin, M. J. Clifton, J.-C. Rouch, P. Aptel, and J. Laine, "The use of dean vortices in coiled hollow-fibre ultrafiltration membranes for water and wastewater treatment," *Desalination*, vol. 118, no. 1-3, pp. 73–79, 1998.
- [94] W.-Q. Tao, Z.-Y. Guo, and B.-X. Wang, "Field synergy principle for enhancing convective heat transfer—its extension and numerical verifications," *International Journal of Heat and Mass Transfer*, vol. 45, no. 18, pp. 3849–3856, 2002.
- [95] Z.-Y. Guo, W.-Q. Tao, and R. Shah, "The field synergy (coordination) principle and its applications in enhancing single phase convective heat transfer," *International Journal of Heat and Mass Transfer*, vol. 48, no. 9, pp. 1797–1807, 2005.

Appendix 1

Full-Scale Comparison of the Traditional and Radial Manifold Heat Sinks

Figure A1.1 illustrates the arrangement of the inlet and outlet manifold channels for the conventional TMM and the proposed R-RMM heat sink configurations. In the TMM design, the inlet and outlet manifold channels are positioned adjacent to one another in an alternating sequence along the heat sink, requiring the coolant to repeatedly change its flow direction before entering and exiting the microchannel array. In contrast, the R-RMM configuration separates the inlet and outlet manifolds into two distinct halves located on opposite sides of the heat sink and connected through curved manifold channels. Furthermore, the conventional lateral inlet and outlet employed in the TMM design are replaced by a centralized feeder and collector in the proposed R-RMM heat sink. This novel arrangement is intended to improve coolant distribution throughout the manifold system while reducing unnecessary flow redirection before the coolant reaches the microchannels.

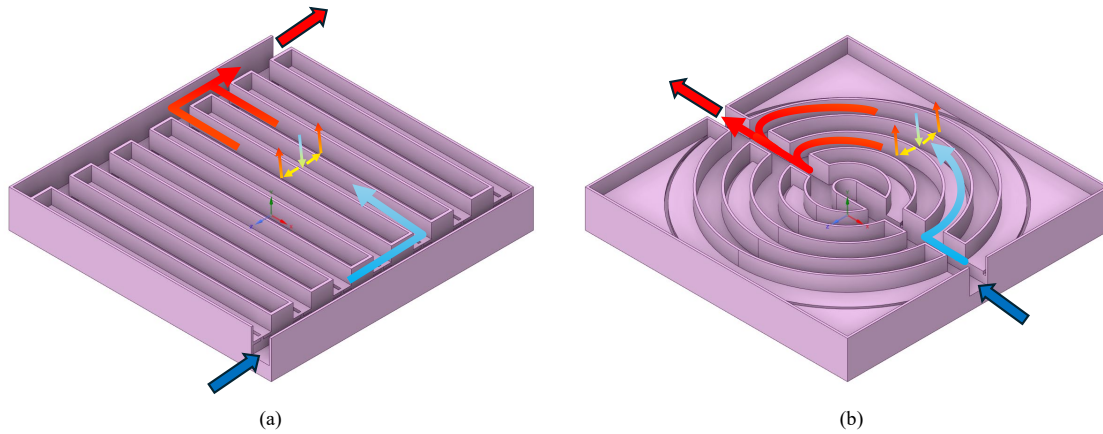


Figure A1.1: Flow pattern and structure diagram of (a) the TMM and (b) the R-RMM heat sink.

A three-dimensional conjugate heat transfer model is employed to evaluate the thermo-hydraulic performance of both heat sink configurations. Silicon is used as the solid substrate, while DI water serves as the working fluid. The performance of the two designs is investigated over a total volumetric flow rate ranging from 3.6 to 12.6 L/h. As illustrated in Figure A1.2, the

proposed R-RMM heat sink simultaneously enhances heat transfer performance and reduces pressure drop compared with the conventional TMM design. Although the unit-cell investigations presented in Chapter 2 indicated that the R-RMM slices produced higher hydraulic resistance than their corresponding TMM counterparts, the complete heat sink exhibits the opposite trend. This improvement is primarily attributed to the replacement of the lateral inlet and outlet channels with a centralized feeder and collector, which substantially shortens the coolant flow path and eliminates the multiple sharp flow redirections required in the conventional TMM configuration. Consequently, the coolant is guided more directly toward the center of the heat sink, reducing hydraulic losses while improving overall flow distribution.

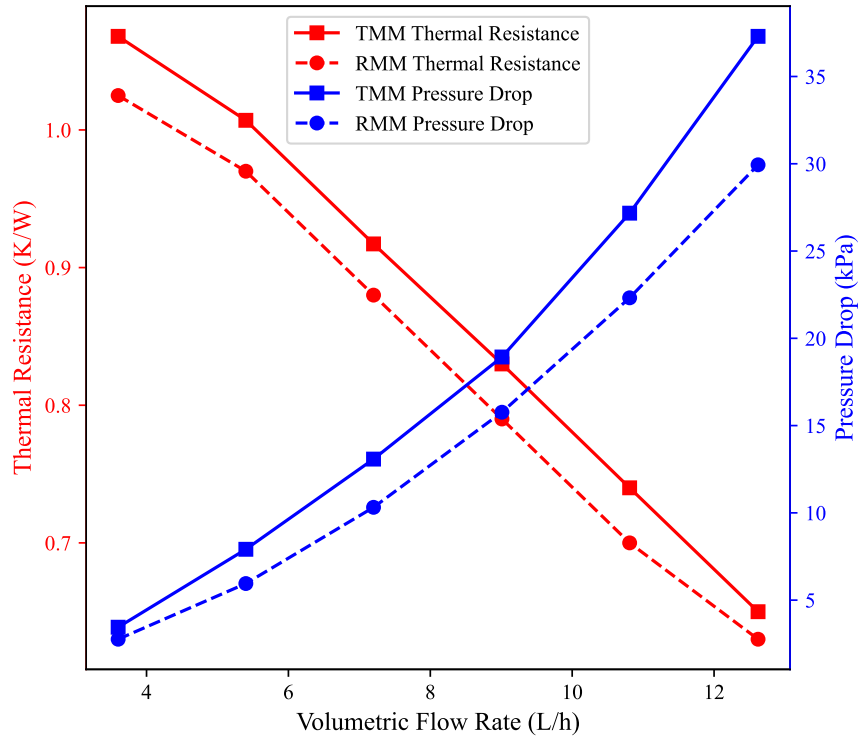


Figure A1.2: Thermo-hydraulic performance of the RMM heat sink and the TMM heat sink.

Figure A1.3 compares the velocity contours at the center cross-section of both heat sink configurations for a total volumetric flow rate of 7.2 L/h. The proposed R-RMM heat sink produces higher flow velocities in the vicinity of the microchannel base, particularly near the central region of the heat sink where the spreading thermal resistance is most significant. Concentrating

the coolant in this critical region enhances local convective heat transfer while utilizing the available pumping power more effectively. In addition, the centralized feeder promotes a more uniform flow distribution among the manifold channels, allowing coolant to be delivered more evenly throughout the microchannel array. In contrast, the conventional TMM heat sink exhibits noticeable flow maldistribution caused by its lateral supply and return manifold arrangement, resulting in non-uniform coolant distribution and less effective utilization of the available coolant for heat removal.

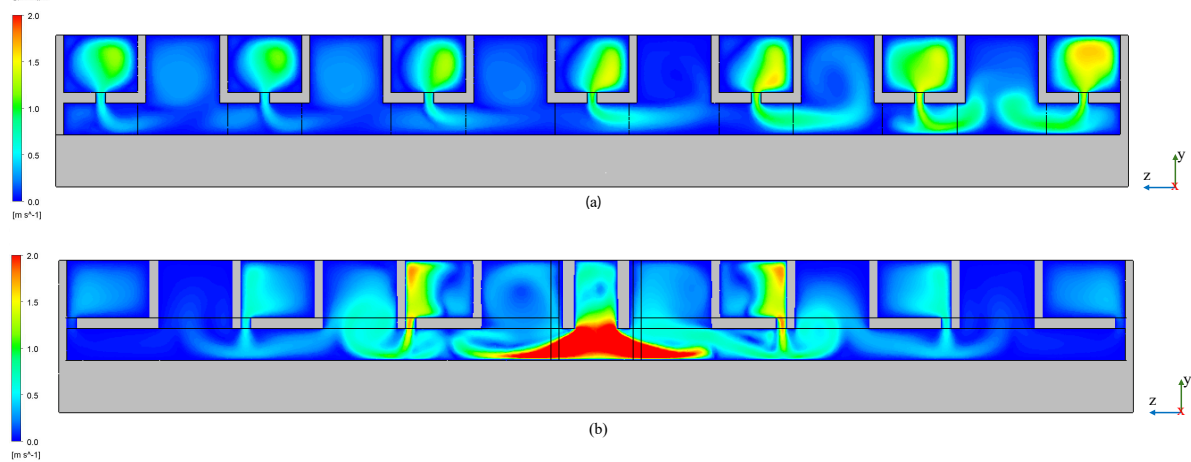


Figure A1.3: Velocity contours at the center cross-section of the heat sink for a total volumetric flow rate of 7.2 L/h for (a) the TMM and (b) the R-RMM.

Appendix 2

Tapered Configurations Geometry and Flow Patterns in the Manifold Section

This appendix collects the geometry of every tapered configuration studied in Chapter 4. For each model, one figure compares the original tapered design with its fixed cross-section and straight reference counterparts, and a second figure compares the original manifold height with its reduced-height counterpart.

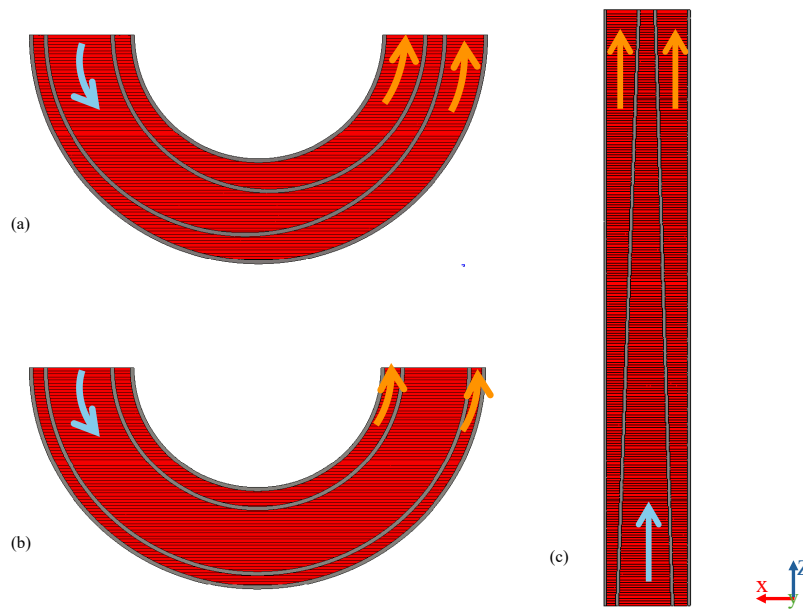


Figure A2.1: Flow pattern and structure diagram of the A1 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

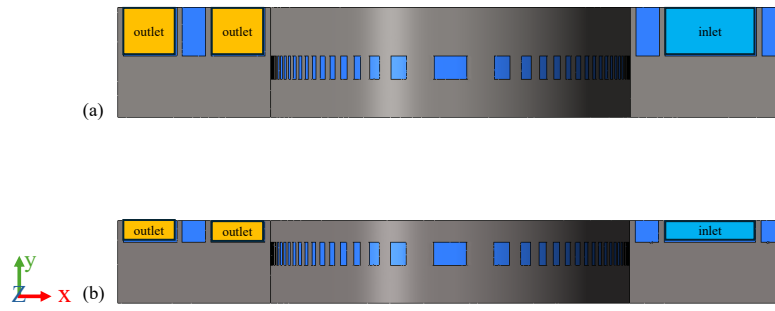


Figure A2.2: Manifold height comparison of (a) the original A1 design and (b) its short counterpart.

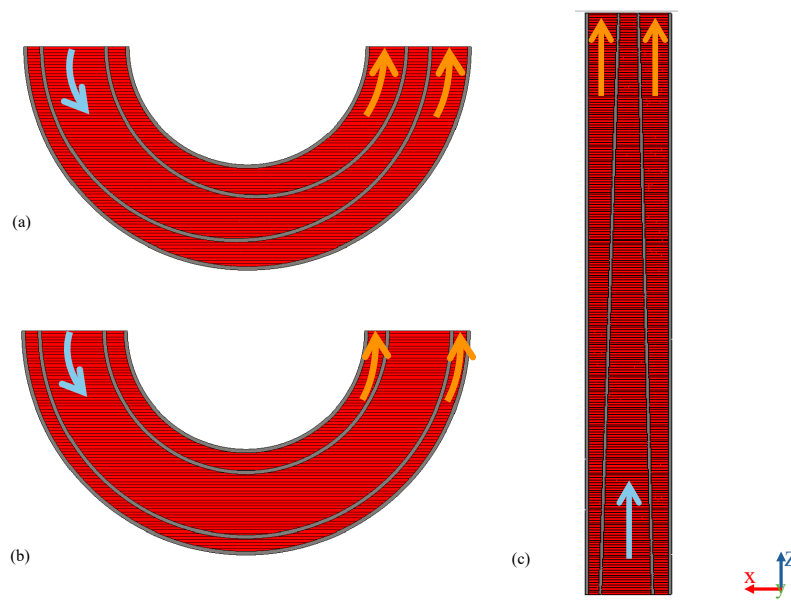


Figure A2.3: Flow pattern and structure diagram of the A2 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

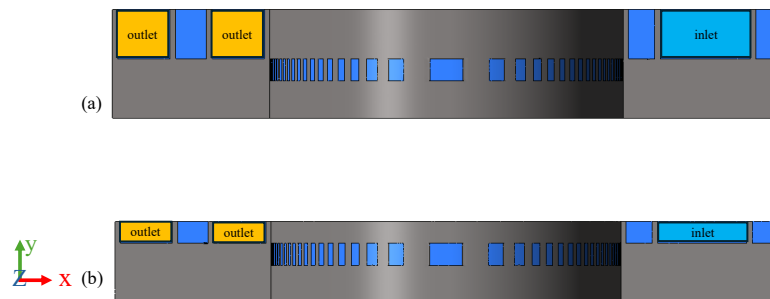


Figure A2.4: Manifold height comparison of (a) the original A2 design and (b) its short counterpart.

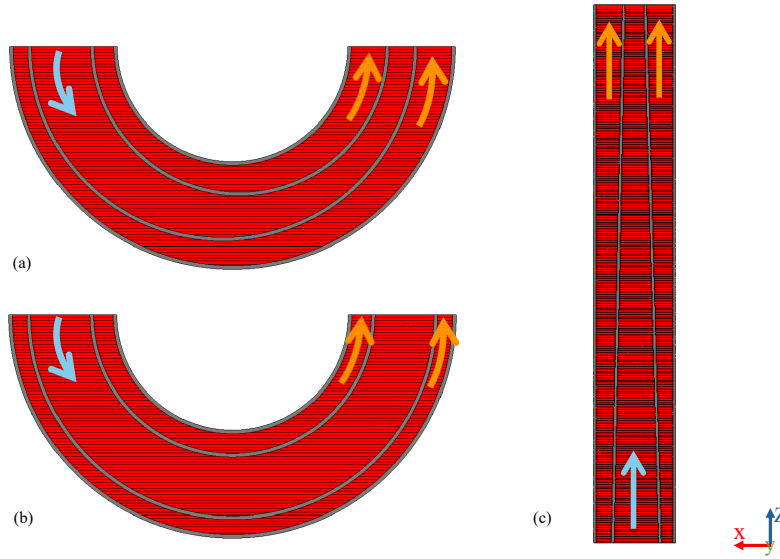


Figure A2.5: Flow pattern and structure diagram of the A3 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

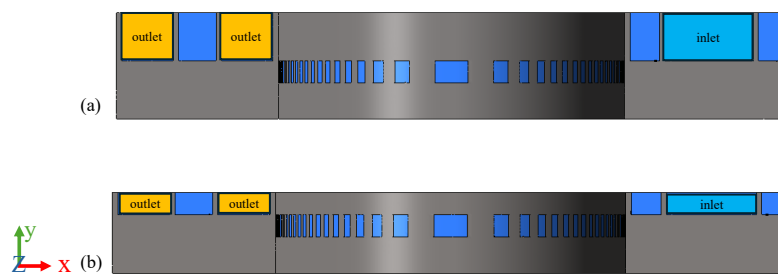


Figure A2.6: Manifold height comparison of (a) the original A3 design and (b) its short counterpart.

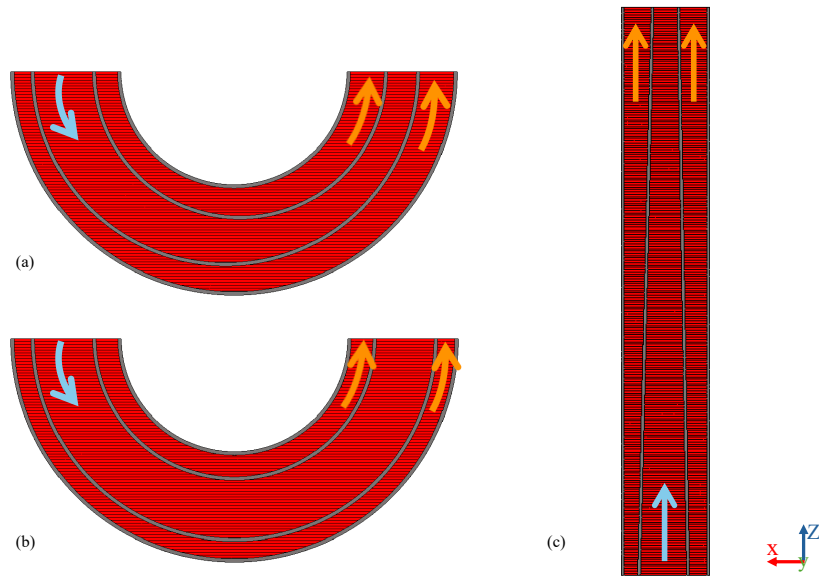


Figure A2.7: Flow pattern and structure diagram of the A4 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

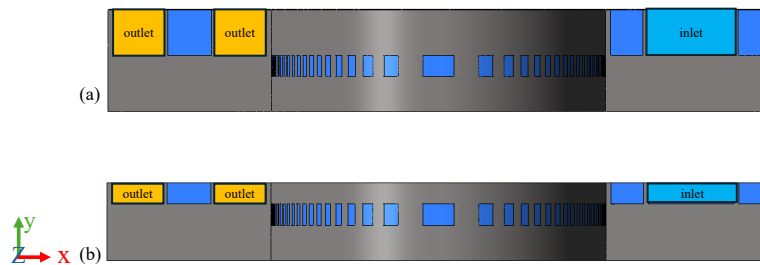


Figure A2.8: Manifold height comparison of (a) the original A4 design and (b) its short counterpart.

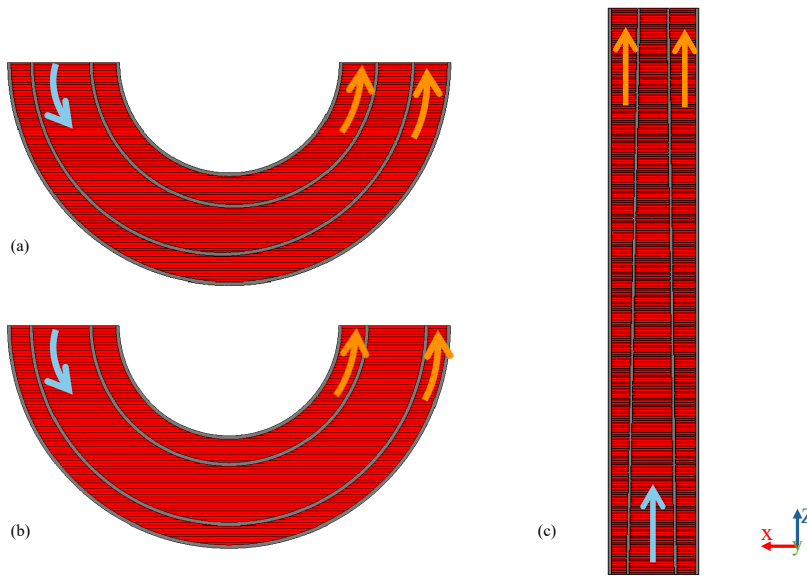


Figure A2.9: Flow pattern and structure diagram of the A5 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

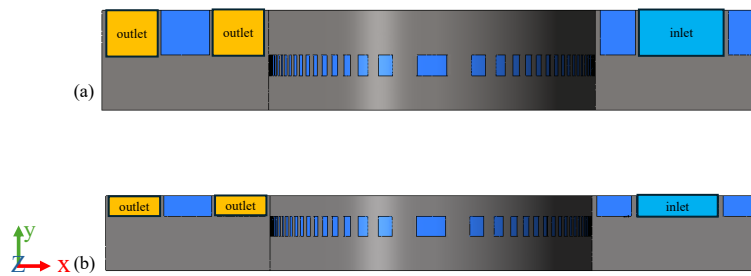


Figure A2.10: Manifold height comparison of (a) the original A5 design and (b) its short counterpart.

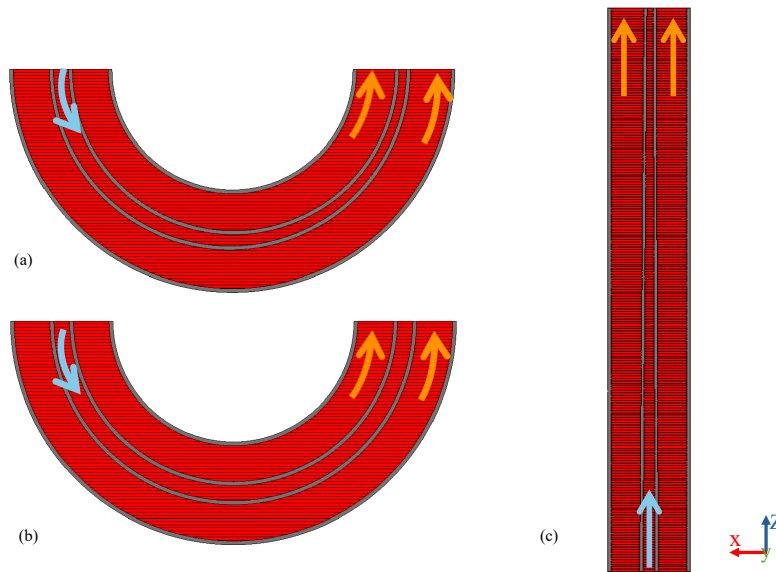


Figure A2.11: Flow pattern and structure diagram of the B1 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

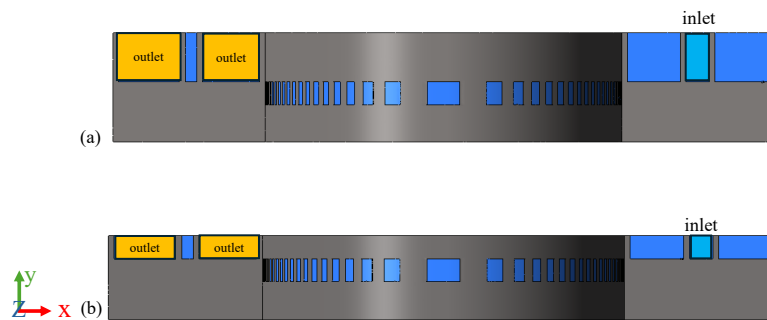


Figure A2.12: Manifold height comparison of (a) the original B1 design and (b) its short counterpart.

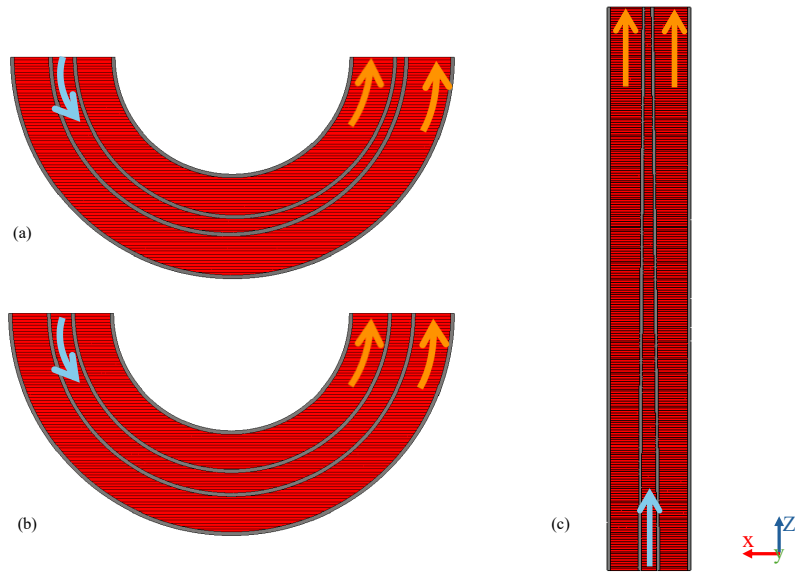


Figure A2.13: Flow pattern and structure diagram of the B2 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

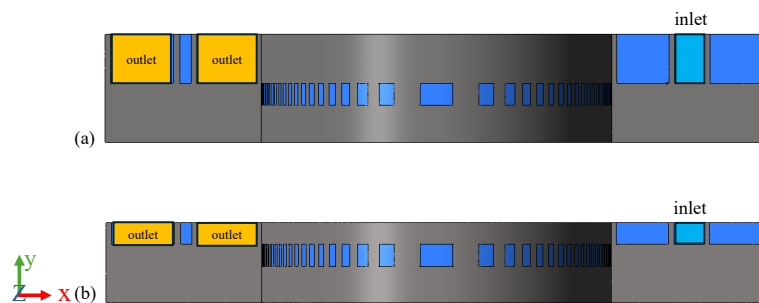


Figure A2.14: Manifold height comparison of (a) the original B2 design and (b) its short counterpart.

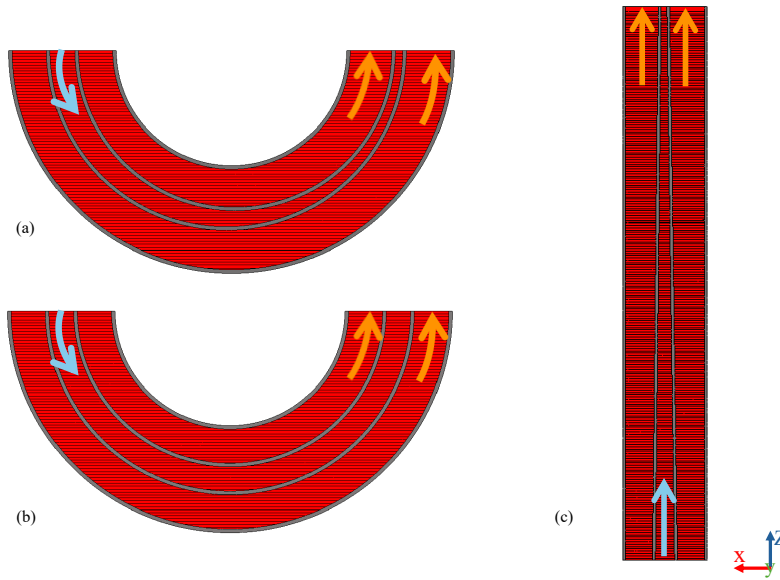


Figure A2.15: Flow pattern and structure diagram of the B3 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

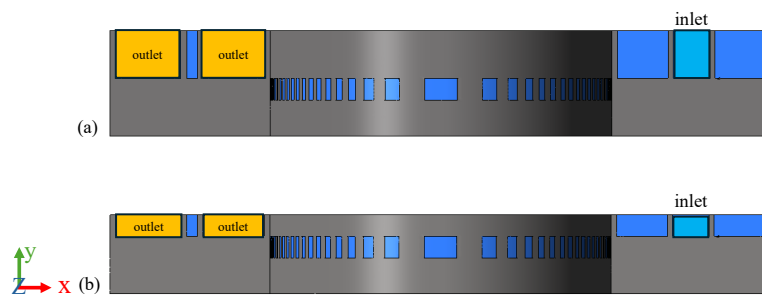


Figure A2.16: Manifold height comparison of (a) the original B3 design and (b) its short counterpart.

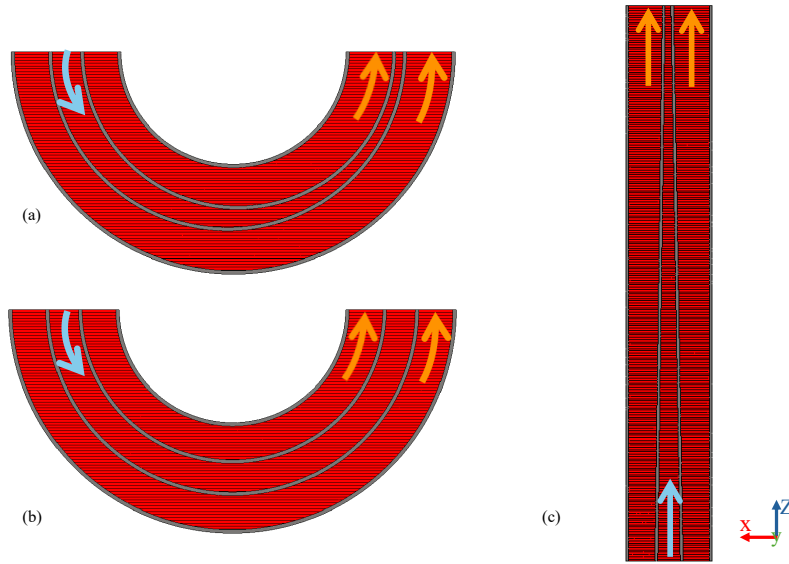


Figure A2.17: Flow pattern and structure diagram of the B4 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

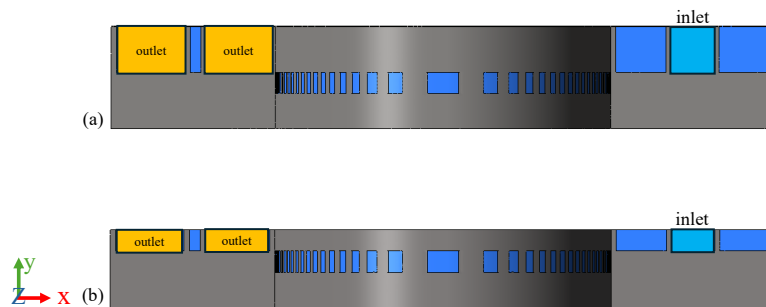


Figure A2.18: Manifold height comparison of (a) the original B4 design and (b) its short counterpart.

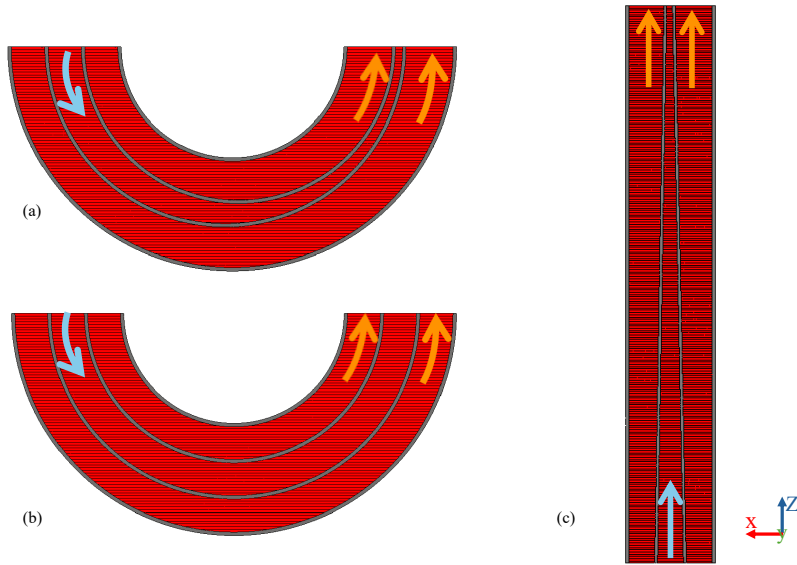


Figure A2.19: Flow pattern and structure diagram of the B5 configuration: (a) original design, (b) fixed cross-section design, and (c) straight counterpart.

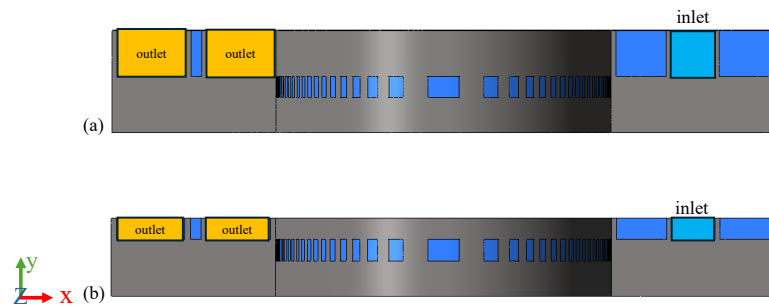


Figure A2.20: Manifold height comparison of (a) the original B5 design and (b) its short counterpart.