Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number and Nanoparticle Concentration

A Thesis Submitted
in Partial Fulfillment of the Requirements for the Degree of

MASTER OF TECHNOLOGY

in
CAAD
by

SHANSI RAWAT
(2K23/CAD/09)

Under the supervision of

Dr. MOHAMMAD ZUNAID

Associate Professor, ME, DTU

DEPARTMENT OF MECHANICAL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

MAY, 2025

DEPARTMENT OF MECHANICAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, SHANSI RAWAT, Roll No. 2K23/CAD/09 student of M.Tech-CAAD Engineering, hereby declare that the project Dissertation titled "Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number and Nanoparticle Concentration" which is submitted by me to the Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma, Associateship, Fellowship or other similar title or recognition.

Place: Delhi

Date: **SHANSI RAWAT**

DEPARTMENT OF MECHANICAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled "Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number and Nanoparticle Concentration" which is submitted by **SHANSI RAWAT**, Roll No. **2K23/CAD/09**, Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the students under my supervision. To the best of knowledge, this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi

Date:

Dr. MOHAMMAD ZUNAID
Associate Professor
SUPERVISOR

DEPARTMENT OF MECHANICAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to my guide, **Dr. M. Zunaid**, Associate Professor in the Department of Mechanical Engineering, for his invaluable guidance, support, and expertise throughout the course of this research project. His constant encouragement, insightful feedback, and dedication have been instrumental in shaping the direction and progress of this work. His deep knowledge and passion for the subject have inspired me to strive for excellence and explore new avenues in the field.

I would also like to extend my heartfelt appreciation to **Prof. B.B. Arora,** Head of the Department of Mechanical Engineering, for his support and encouragement. His vision and leadership have provided a conducive environment for academic and research pursuits. I am grateful for his valuable insights and guidance that have contributed to the overall success of this project.

Lastly, I would like to acknowledge my family and friends for their unwavering support, encouragement, and understanding throughout this research endeavour. Their love, belief in my abilities, and motivation have been the driving force behind my perseverance and determination.

Place: Delhi

Date:

SHANSI RAWAT

2K23/CAD/09

M. Tech

Delhi Technological University

ABSTRACT

This thesis undertakes a numerical study by employing a unit cell approach for three-dimensional fluid flow and heat transfer in micro-channel heat sinks. Two configurations were simulated: a traditional straight rectangular microchannel and a novel design featuring circular bumps spaced evenly along the channel walls. The numerical simulations were carried out using ANSYS Design Modeler and Fluent, and the computed results were validated with existing experimental data. Both configurations' thermal and hydrodynamic characteristics were assessed for changing Reynolds numbers (100–900) and imposed heat fluxes (100, 150, and 200 W/cm²), and water and Al_2O_3-water nanofluids of 1.5% and 2.5% volume concentrations as working fluids. Microchannels were represented by oxygen-free copper walls because of its high thermal conductivity.

Results show that raising the Reynolds number has a remarkable effect on convective heat transfer, resulting in lower temperature differences across the inlet and outlet ($\Delta T = T_{out} - T_{in}$). The bumped geometry showed consistently better thermal performance than the straight channel owing to added surface area and flow disturbances that promote mixing and heat transfer. For example, at Re = 500, the surface temperature increases of 25% and 34% and the outlet temperature increases of 50% and 33% were observed, respectively, over heat flux ranges of 100–150 W/cm² and 150–200 W/cm².

Enhancement was even more noted with the inclusion of Al_2O_3 nanoparticles in the base fluid. At 1.5% concentration in bumped geometry, surface temperatures increased by 35% and 25%, whereas outlet temperatures increased by 35% and 37% for the same flux ranges. The same trends were noted for 2.5% concentration. The bumped design had greater pressure drops, particularly at high Reynolds numbers, but had a good pressure loss versus heat removal efficiency trade-off

LIST OF PUBLICATIONS

[1] Shansi Rawat and Mohammad Zunaid, " Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number and Nanoparticle Concentration" submitted in The Power and Intelligent Control Systems (PICS-2025), Scopus Indexed, 2025, NIT Hamirpur, (H.P.).

(Accepted)

[2] Shansi Rawat and Mohammad Zunaid, " Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number" submitted in The Power and Intelligent Control Systems (PICS-2025), Scopus Indexed, 2025, NIT Hamirpur, (H.P.).

(Under Review)

TABLE OF CONTENTS

Title		Page No.

Title **Page No.**

LIST OF TABLES

Title **Page No.**

LIST OF FIGURES

Title **Page No.**

Title	**Page No.**

NOMENCLATURE

W_1	Height of the cover plate
H_1	Thermocouple height
W_{ch}	Microchannel width
H_{TH}	Thermocouple height
H_{W1}	Height above microchannel
H_{W2}	Height below Microchannel
L	Length of Microchannel
LIBs	Lithium-ion batteries
CAD	Computer aided design
LEDs	Light emitting diodes
CFD	Computational Fluid Dynamics
MCHSs	Micro-channel heat sinks
VLSI chips	Very Large-Scale Integration chips
PCM	Phase Change Material
FVM	Finite volume method
K_s	Thermal conductivities of the solid
K_f	Thermal conductivities of the fluid
$T_{s, c}$	Temperatures of the solid at the interface
$T_{f, c}$	Temperatures of the fluid at the interface
Re	Renolds number

CHAPTER 1

INTRODUCTION

1.1 Microchannel Heat Sink - An Overview

The accelerated miniaturization of electronics and exponential growth in power density in future high-performance computing devices have prompted the need to produce high-efficiency thermal management devices. Early research by Tuckerman and Pease indicated the potential of MCHSs to cool VLSI chips, as they were able to manage much higher heat fluxes than traditional cooling schemes. Micro-channel heat sinks (MCHSs) are now an icon technology in the field due to their remarkable surface-area-volume ratio and ability to utilize convective heat transfer manage thermal fluxes that are well in excess of 1,000 W/cm^2 [1]. Hajmohammadi and Toghraei [2] maximized the design of a double-layered MCHS employing Al_2O_3 nanofluids and achieved a 10% thermal resistance reduction in comparison to prior designs. More recent studies have also examined innovative microchannel structures and geometries to maximize thermal performance with nanofluids [3].

Computational Fluid Dynamics (CFD), a crucial tool for studying the complex flow and heat transfer processes within MCHSs. Several studies have utilized CFD software, e.g., Fluent, to investigate MCHSs with nanofluids under different operating condition optimization of MCHSs utilizing nanofluids. Here we explore the relationship between flow rate, Reynolds number, and weight% composition of nano particles used, Al_2O_3, resulting in outlet temperature, surface temperature, and pressure difference in MCHS, to develop a predictive model for outlet temperature under various operating conditions.

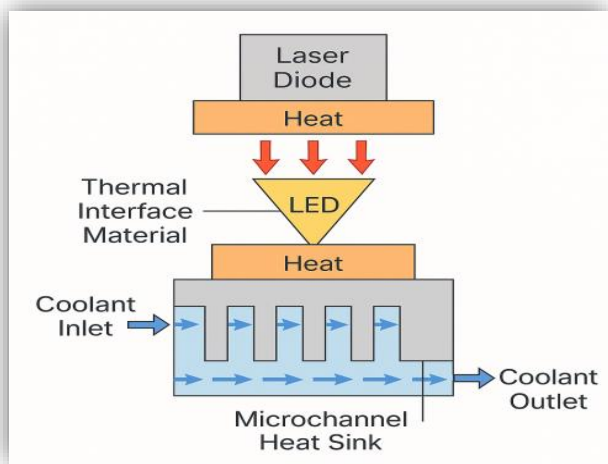


Fig.1.1 A schematic diagram of Laser Diode cooling system

Although their benefits, the behaviour of microchannel heat sinks depends on various parameters such as channel geometry, flow structure, fluid properties, and boundary conditions of temperature. Rectangular microchannels, for example, are widely analysed for convenience of fabrication and analysis but other geometries like trapezoidal, triangular, and circular cross-section are also analysed with aim to improve thermal behaviour and minimize pressure drop [4].

1.2. Basics of Micro-Channel Heat Sinks

Microchannel heat sinks are small-sized thermal devices used to dissipate large heat fluxes from tiny surface areas. They are typically made up of many parallel channels, with the hydraulic diameter ranging from 10 µm to 200 µm, between which a coolant passes**.** The basic operating mechanism of a microchannel heat sink lies in forced convection. Water or dielectric fluid is pumped through the microchannels. As it flows through the channels, it picks up the heat produced by a heat source (e.g., microprocessor or laser diode) mounted at the base of the heat sink. Because of their tiny channel size, there is a large surface area-to-volume ratio, which greatly increases the rate of heat transfer over traditional heat sinks.

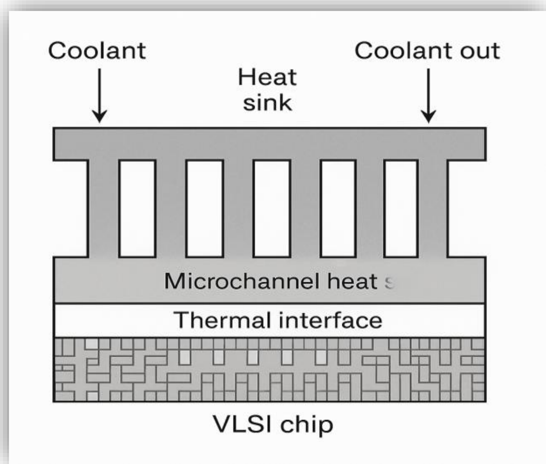


Fig.1.2. Components of the VLSI cooling system

With the persistent pace of semiconductor technology, VLSI (Very Large-Scale Integration) chips have come to feature billions of transistors within very small footprints. This integration has significantly risen the power density of electronic devices, bringing with it quite severe thermal management issues. Waste heat, if not effectively dissipated, can cause performance degradation, reliability problems, and even permanent failure of VLSI circuits. Hermetic air-cooled heat sinks and fans have been found inadequate in satisfying the thermal requirements of current VLSI chips, leading to the investigation of other high-performance cooling methods. A number of design aspects are important in the application of MCHSs to VLSI chips [4]:

a) Pressure Drop: A high heat transfer rate is usually at the expense of a high pressure drop. A balance between thermal performance and pumping power is needed in VLSI systems to ensure energy efficiency.

b) Thermal Resistance: The overall thermal resistance of the system should be minimized, encompassing contributions from the chip-to-sink interface, microchannel wall, and the convective resistance of the coolant.

c) Fabrication Compatibility: MCHSs should be compatible with the semiconductor fabrication processes, particularly when integrated at the package or wafer level rated at the wafer or package level.

d) Channel Geometry: Rectangular, trapezoidal, and circular microchannels are commonly studied where rectangular microchannels are most widely used.

e) Coolant Selection: Water is a popular coolant due to its high thermal conductivity and heat capacity, though dielectric fluids are sometimes used to prevent electrical issues.

MCHS integration into the primary purpose of this structure is to facilitate efficient thermal management in heat-generating components, such as semiconductors, power electronics, laser diodes, and LEDs, where high heat fluxes need to be removed from the component's surface to ensure stable operation and prevent thermal damage. As power densities keep increasing, novel designs and materials for microchannel heat sinks will become absolutely vital to assure thermal reliability and operational lifetimes for the systems [5][6].

1.3 Components and Structure of a Micro-Channel Heat Sink

A conventional microchannel heat sink consists of a number of important constituents that coordinate their efforts to achieve maximum heat extraction due to increased convective heat transport. The principal structural elements are:

a) Base Plate: the base plate contains the microchannels and physically connects with the heat source and makes it possible for heat conductance from the device surface all the way down to the cooling fluid, typically made of high thermal conductivity materials like silicon, copper, or aluminium. Aluminium, for example while having lower thermal conductivity (about 205 W/m·K), is lighter, more cost-effective, and easier to manufacture, making it a popular choice for many applications. Silicon is utilized for application in electronics—namely for semiconductors and power devices—due to its suitability for micro-fabrication processes and thermally suitable with chip materials.

b) Microchannels: thin channels, usually between 10 and 500 μm hydraulic diameter, etched or milled into the substrate. Microchannels significantly enhance surface area for heat exchange and allow laminar or transitional fluid flow, increasing convective heat transfer. Different geometries—rectangular, trapezoidal, circular—have been investigated to achieve the best performance based on application [4]. Parallel channel arrays are the most widely used configuration.

c) Inlet and Outlet: distribute the coolant evenly into the microchannels and return the heated fluid at the exit. Optimum manifold design is important to achieve uniform flow distribution and to prevent flow mal-distribution, which causes thermal hot-spots on the chip surface. In VLSI and LED cooling applications, manifold design has a significant influence on overall thermal performance [6].

d) Coolant: the working fluid (coolant) is responsible for the removal of heat from the system. Water is the most prevalent one because of its high thermal conductivity and specific heat, but dielectric fluids like Fluorinert is used in sensitive electronic applications to avoid electrical hazard during leakage or direct contact [7]. Nanofluids, base fluids with nanoparticles suspended in them, are applied in some advanced designs to further improve the thermal performance.

e) Thermal Interface: between the heat-generating element and microchannel base, the interfacial thermal resistance is usually minimized using a thermal interface. It can be used to fill the microscopic air gaps and enhance conduction, the importance of which is particularly necessary in laser diode and LED cooling applications where the quality of the thermal contact seriously affects the light emission and device lifetime [8].

1.4. Mechanisms of Heat Transfer in Micro-Channel Heat Sinks

These devices emit significant amounts of heat in extremely small quantities, and this generates high heat flux densities that can cause significant performance deterioration, reliability, and life if not dissipated. MCHSs overcome this challenge through the use of sophisticated heat transfer mechanisms.

a) Heat Transfer Through Substrate (Conduction): The initial thermal dissipation is through conduction from the heat-producing element (for example, the semiconductor dies) into the substrate of the microchannel heat sink. The substrate, usually constructed of high thermal conductivity materials like copper, aluminium, or silicon, provides a thermal bridge. The hot junction of the device releases the heat into the base of the MCHS via a thermal interface, which minimizes interfacial resistance [6][9].

b) Heat Removal by Fluid Flow (Convection): these devices emit significant amounts of heat in extremely small quantities, and this generates high heat flux densities that can cause significant performance deterioration, reliability, and life if not dissipated. MCHSs overcome this challenge through the use of sophisticated heat transfer mechanisms [10].

c) Boiling and Two-Phase Heat Transfer: In systems with very high heat fluxes (e.g., power electronics and RF devices), two-phase cooling (boiling) is commonly used in microchannels. The process incorporates the phase transition of the coolant from liquid to vapor within the channels, which enormously enhances the effective heat transfer coefficient by latent heat absorption [3].

d) Heat Spreading: In certain applications, the lateral conduction or "heat spreading" in the substrate is also relevant. Especially in power amplifiers or LED arrays, where heat is being generated non-uniformly, the substrate also serves as a heat spreader to spread the heat uniformly before it enters microchannels. High thermal conductivity materials or stack composites are usually used to amplify this effect.

e) Thermal Interface Management: The role of thermal interface between the heat source and the MCHS base is crucial. Thermal interface fill microscopic gaps and reduce contact resistance that would otherwise act as bottlenecks in the conduction path. Materials like phase-change thermal greases, or nano-enhanced composites are used to minimize the overall thermal resistance of the system [8].

1.5 Microchannel Geometry

Microchannel heat sink geometry is critical to the thermal-hydraulic performance of MCHSs, particularly for cooling high-heat-flux devices such as semiconductors, power electronics, laser diodes, and LEDs. These devices are highly sensitive to temperature variations, and poor heat removal can lead to thermal hot-spots, performance degradation, and even catastrophic failure. It is essential to optimize geometric parameters to obtain low thermal resistance, high heat transfer coefficients, and reasonable pressure drop while ensuring chip temperature uniformity and reliability [8][11].

a) Importance of Geometry in High Heat Flux Applications - High-power devices such as RF transceivers, and laser arrays require accurate thermal management. The geometry of MCHS influences not just the convective heat transfer rate but also temperature distribution uniformity over the surface of the device. Parameters like channel width, height, hydraulic diameter, aspect ratio, number of channels, and fin thickness are all important considerations in this context. For instance, in power modules and semiconductors, rectangular microchannels of high aspect ratio are commonly utilized to optimize surface area within restricted space.

b) Common Microchannel Geometries - Rectangular microchannels are most prevalent because they are easy to fabricate and compatible with planar silicon substrates. The large surface-area-to-volume ratio provides increased convective heat transfer. Trapezoidal channels are usually formed as a secondary effect of wet etching techniques on silicon and are more mechanically stable than ideal rectangular channels. Circular shapes minimize pressure drop and help prevent the creation of sharp corners where pieces of debris or bubbles can stick. Less prevalent because of the complexity of fabrication but provide very good flow characteristics and can prove useful in some applications. Pin-Fin and Cross-Linked Microchannel Designs higher-order designs feature pin-fin geometries or cross-connected channel networks to prevent thermal boundary layers and promote mixing in the flow.

c) Geometric Parameters Influencing Performance Some of the important parameters characterize the thermal and hydraulic performance of an MCHS. Channel width (W) and height (H) impacts hydraulic diameter and flow regime whereas aspect ratio (H/W): Increased ratios enhance surface area but may enhance fabrication complexity. Hydraulic diameter (D_h) defined as $D_h = 4A/P$, where A is cross-sectional area and P is wetted perimeter; critical for calculating Reynolds number and convective coefficients.

1.6 Coolant Properties

The choice of an appropriate coolant is one of the most important considerations in microchannel heat sink (MCHS) design, especially for the thermal management of high heat flux components like semiconductors, power electronics, laser diodes, and LEDs. Coolant selection has a direct influence on the performance of a heat sink in cooling and ensuring temperature homogeneity in the surface area of a device. The thermophysical characteristics of the coolant are an important factor in establishing the overall efficiency of heat transfer i.e. thermal conductivity, specific heat capacity, density, dynamic viscosity, and boiling point. Low electrical conductivity is also required for some applications—particularly where the coolant could come into contact with electronic components—to avoid short circuits. Higher thermal conductivity facilitates the faster conduction of heat into the fluid, whereas high specific heat enables the fluid to hold more heat. At the same time low viscosity decreases pumping power requirements and lowers the pressure drop in the microchannels, which is especially crucial in compact systems with low energy budgets.

Water is the most common coolant used in MCHSs for its superior heat conductivity (~0.6 W/m·K) and high specific heat capacity (~4.18 kJ/kg·K). These characteristics are particularly ideal for cooling VLSI chips and semiconductors, where both high thermal and space limitations exist. Water's electrical conductivity is a hazard in direct-contact systems, requiring electronic components to be isolated or encapsulated. In electrically sensitive applications, power electronics and laser packaging, electrically insulating dielectric fluids such as 3M Fluorinert™. These fluids are chemically stable, non-flammable, and electrically insulating, but have much lower thermal conductivity and specific heat and are more costly than water [8].

In applications where freezing is a problem ethylene glycol and water mixtures are used most commonly, these antifreeze protection mixtures are high-end applications for cooling, particularly within research and development settings. Nanofluids are engineered by suspending nanoparticles like aluminium oxide, copper oxide, or graphene in base fluids for enhancing their thermal conductivity and heat transfer. Both theoretical and experimental studies have proved that nanofluids can be utilized to enhance convective heat transfer coefficients by 20–30% and thus are of great interest for cooling high-power laser diodes [12]. The choice of a coolant will depend on the particular application, taking into account factors such as necessary thermal performance, electrical insulation, risk of corrosion, pumping requirement, and cost of the system. Deionized water or water-based nanofluids, can be best for high-performance computing chips, whereas dielectric fluids are required for power electronics in order to eliminate electrical hazards. Hence, successful coolant selection is a matter of balancing thermal and hydraulic performance with electrical, chemical, and cost factors, to provide reliability and peak performance over the long term in high heat flux electronic devices.

1.7 Objective of Thesis

The primary objective of this thesis is to investigate the design, performance, and applications potential of single-phase micro-channel heat sinks for cooling semiconductors with varying heat flux, Reynolds number and nano particle concentration. The thesis work examines the performance of a liquid cooling design based on small microchannel dispersed along the length of the cell and cooled by water. A three-dimensional Computer-aided design (CAD) model was created for the proposed design using Ansys Design Modeller, and numerical simulations were run using the CFD solver in the software Ansys Fluent to solve the continuity equation and Energy equation. The specific objectives of the thesis can be summarized as follows:

i. Establish a Core Understanding - Create a computational model in order to predict the thermal behaviour of MCHSs under fluctuating heat flux, Reynolds number and nano particle concentration with the aim of finding optimal configurations for particular application in high-power microelectronics. A stable computational model is an in silicon tested where it is easy to iterate at speed and undertake exhaustive parametric studies without having physical hardware. It gives detailed information about fluid flow patterns, temperature profiles, and local heat transfer coefficients that are hard to identify experimentally.

ii. Research and analyse different microchannel configurations (rectangular, trapezoidal, pin-fin, etc.) based on their ability to promote heat evacuation without increasing pressure drop and flow maldistribution. This goal addresses the important interrelationship between the geometry of a microchannel and both its hydraulic and thermal performance. The challenge is to increase the heat transfer (evacuation of heat) while at the same time offsetting the drawback of increased pressure drop and uniform flow.

iii. Investigate and analyse various nano particle weight percent concentration, the impact achieved on the rise in outlet temperature i.e. removal of heat and reduction in pressure drop. This sub-objective examines the use of nanofluids as a coolant, specifically examining how particle concentration influences both thermal behaviour and hydraulic properties.

CHAPTER 2

LITRATURE REVIEW

Effective thermal management is crucial in high-power density and high-performance applications like lithium-ion batteries (LIBs), semiconductors, power electronics, and optoelectronic devices in which high temperatures can enormously degrade performance, lower efficiency, and decrease operational lifespan. In order to enhance the need for miniaturized and reliable thermal control technologies, microchannel heat sinks (MCHSs) now have gained interest as next-generation solutions owing to their compact dimensions, giving excellent surface area-to-volume ratio, and effective heat dissipation [1]. Microchannel design optimization has been the focus of recent research in order to meet the demands of high-power applications. A thorough understanding of the thermal and fluid dynamics properties of MCHSs was provided by Kandlikar et al. (2006), who emphasized the impact of channel design by optimizing convective heat transfer while minimizing pressure drop. The performance of straight microchannels may be limited at low coolant flow rates or small temperature gradients, despite their simpler flow pathways and comparatively lower pressure drops [4]. Here, two leading configurations in this regard are straight and wavy microchannels. Though straight microchannels are characterized as having relatively lower pressure drops with simpler flow paths, their performance can be constrained at low coolant flow rates or small temperature gradients [4]. While on the other hand, wavy microchannels have come into focus due to their capability of causing higher turbulence and mixing of fluid, leading to better heat transfer coefficients even in situations when a high flow rate is difficult to maintain.

The paper entitled "A Critical Review of the Straight and Wavy Microchannel Heat Sink and the Application in Lithium-Ion Battery Thermal Management" by M. Hajialibabaei and M. Z. Saghir [11] presents a critical review of straight and wavy microchannel heat sinks' thermal performance, specifically applied to lithium-ion battery (LIB) thermal management. Effective heat control in LIBs is a critical challenge, particularly in high-power applications where excessive temperature increase can significantly degrade battery efficiency, safety, and lifespan. Owing to their minimal size and high surface-area-to-volume ratio, microchannel heat sinks have emerged as promising candidates for efficient heat enhancement. The authors present an extended comparison of the thermal performances of wavy and straight microchannels. Since straight microchannels have a linear, simple shape and no complex flow patterns, they are often less vulnerable to pressure drops but at the same time, their capacity to transfer heat may be impaired when the fluid flow rate is low or the coolant and channel wall have a minor temperature differential. Even with lower flow rates, thermal dissipation due to increased turbulence and fluid mixing brought can be slightly improved by wavy microchannels on by their curved form. Because of this, wavy microchannels are more useful for applications that need better thermal control without sacrificing tolerable pressure drops [13].

2.1 Surface Modifications and Strengthened Structures

a) Surface Roughness:

- Mechanism: At the microscopic scale as well, surface roughness due to manufacturing processes or even engineered intentionally can drastically change the flow field close to channel walls. These micro-scale asperities are "turbulators" and induce local flow separation and reattachment. This incessant disturbance of the thermal boundary layer forecloses its complete growth, sustaining high temperature gradients near the wall and thereby augmenting the local convective heat transfer coefficient [21]
- Boiling Enhancement: In applications for two-phase cooling, controlled surface roughness becomes even more important. Roughness features can become good nucleation sites for bubble growth during boiling. While offering favourable fluid dynamic characteristics (lower pressure drop), their fabrication in MCHSs can be more complex. By offering additional sites for initiation of phase change, roughness can result in an earlier nucleate boiling onset, stabilize and enhance

bubble growth and departure, and eventually enhance the critical heat flux prior to dry-out. This is critical in realizing increased heat removal rates and steady operation in boiling microchannels.

b) Internal Fins/Ribs:

- Mechanism: Passive flow manipulators in the form of small ribs or fins embedded within the walls of the microchannels, usually transverse or inclined to the primary flow direction, create secondary flows, swirls, and vortices within the microchannels. These secondary motions improve the mixing of fluid, reducing it near the heated walls. This scouring mechanism is instrument in removing the thermal boundary layer, and it boosts the heat transfer coefficient [23].

c) Pin-Fins:

- Mechanism: Instead of a straight fin, pin-fin microchannels consist of arrays of discrete, small cylinders or posts in the flow stream. The pin-fins substantially enhance the total heat transfer surface area. Of more importance, they induce robust flow disturbances, including flow impingement, separation, and vortex shedding about each pin This produces strong local mixing and boundary layer disruption, generating heat transfer coefficients much higher than straight channels [23].
- Performance vs. Pressure Drop: Although pin-fins provide better heat transfer and commonly superior temperature uniformity, they generally involve a larger pressure drop as a result of greater form drag. Experimental work concentrates on the optimization of pin-fin shape (e.g., circular, elliptical, diamond), arrangement (in-line, staggered), and size to achieve maximum thermal performance with the least hydraulic penalty [24].

d) Re-entrant Cavities and Porous Structures:

- Mechanism: These are more sophisticated, frequently three-dimensional, interior geometries. Re-entrant cavities are inward-facing features at the walls of the channel that can trap vapor bubbles, inducing stable nucleation and augmenting boiling heat transfer. Porous structures (e.g., metal foams, sintered particles, or architected porous media) incorporated within or as a component of the microchannel walls deliver an extremely enormous internal surface area and complex flow paths. This greatly improves the convective heat transfer (because of enhanced surface area and tortuous flow) and boiling heat transfer (by offering an abundance of nucleation sites and capillary wicking for liquid delivery.

- Fabrication Impact: The possibility of fabricating such complex structures has been greatly enabled by additive manufacturing (3D printing) methods, which allow for the production of highly optimized and complicated internal geometries that were not attainable before [18]. Such structures have the ability to drastically enhance the effective surface area and create many nucleation points for boiling, resulting in very high heat flux and enhanced stability in two-phase systems.

2.2 Challenges and Limitations

While having their benefits, MCHSs incur a number of design and operating difficulties through process:

a) High Pressure Drop: Due to the low channel dimensions, unavoidable high viscous friction and pressure drops result, strong pumps and leading to system increased power consumption [9].
b) Manufacturing Complexity: Microchannel fabrication with high repeatability and precision may be challenging and expensive, requiring sophisticated micro-machining methods (e.g., deep reactive ion etching, micro milling, or additive manufacturing) [18].
c) Flow Instabilities: Microchannels are prone to some flow installations (e.g., parallel channel instability, excursive instability) in two-phase flow, and these instabilities may introduce pressure and temperature fluctuations, resulting in early critical heat flux and device failure [6, 7].
d) Critical Heat Flux: Although two-phase cooling provides high rates of heat removal, exists above which boiling ceases to be a stable process, resulting in dry-out and

sudden increase in surface temperature. This prediction and extension in microchannels are a current area of active research [6].

e) Clogging: The small channel size makes the MCHSs susceptible to clogging by foreign particles or by coolant impurities, which can substantially impair performance.

2.3 Passive Enhancement for Heat Transfer:

a) Peng and Wang [12]: One of the first scientists to explore with heat transfer and fluid flow in rectangular microchannels they made an important observation: as the hydraulic diameter of these microchannels was reduced, there was a marked enhancement in the Nusselt number and convective heat transfer. This result emphasized the promise of miniaturization to improve heat transfer since improved surface area to volume ratio at smaller scale channels enables improved heat dissipation. Their work provided foundation knowledge in microchannel heat sink design.

b) Hetsroni [17]: Following up from the idea of intensifying heat transfer, Hetsroni's work concentrated on the effect of surface enhancement such as rough surfaces and internal fins in channels. The most important point was that minute geometric modifications would contribute significantly to intensification of thermal performance. This is due to two prominent mechanisms:

- Disturbance of the thermal boundary layer: Roughness and fins induce turbulence and disturb the smooth flow of the fluid close to the heated surface, which keeps the thermal boundary layer from being very thick and insulating.
- Increased fluid mixing: The geometric disturbances enhance the mixing of the fluid, introducing cooler fluid nearer to the hot surface and evacuating hotter fluid more effectively. This passive technique for enhancement presents a method for enhancing heat transfer without external input of power.

c) Luan and Li [18]: Their laboratory experiments particularly proved the potential of microchannel-based cold plates in stabilizing the temperatures of diode lasers, even under dynamic loading conditions. This is an important factor for applications in lasers, where temperature variations cause wavelength shifts, poorer beam quality, and even destruction of the laser parts. Their capability to keep up stable temperatures makes them efficient in regulating transient heat loads that are prevalent in most sophisticated electronic and optoelectronic systems.

d) Garimella and Sobhan [19]: Their study focused on the convective heat transfer performance of FC-72, a dielectric fluid, in microchannels. On compared it with water, which is a widely used cooling fluid. They compared its performance to that of water, a common cooling fluid. Their findings revealed that while FC-72 exhibited a lower heat transfer coefficient compared to water.

- Lower Heat Transfer Coefficient: This implies that under identical conditions, FC-72 may not transfer heat as well as water on a unit area basis.
- Safer Choice for Electrical Insulation: The most important advantage of FC-72 is its electrical insulation characteristic. This makes it a perfectly suitable option for cooling delicate electronic devices were coming into contact with an electrically conductive fluid such as water may result in short circuits or other electrical failures. Thus, where applications require efficient cooling and intense electrical isolation, FC-72 offers a valuable safer option even though it has relatively inferior thermal performance.

2.4 Importance of Coolant Selection in Optimization:

a) Beyond Geometry: The first text points out that microchannel geometry is important for heat transfer but the choice of the coolant itself is equally, if not more, important in optimizing the performance of Microchannel Heat Sinks (MCHSs) in general. This means that an optimally designed channel will not operate at its best without an appropriate working fluid.

b) Influence of Thermophysical Properties: An MCHS's performance is essentially determined by the thermophysical properties of the coolant used. They are

- Thermal Conductivity: How efficiently the fluid conducts heat. Higher conductivity generally means better heat dissipation

- Viscosity: It is a measure of resistance to flow of the fluid. Lower viscosity normally results in lower pressure drop, which is desired for reducing pumping power.
- Electrical Conductivity: Especially important in electronic cooling, since excessive electrical conductivity can result in short circuits when the fluid contacts delicate electronic components.
- The interplay between these characteristics determines the efficiency of the fluid in absorbing, carrying, and emitting heat while keeping in mind the related pumping power and safety.

Zhang et al. [20] illustrated that the integration of PCMs into microchannel geometries—either in cavities in close proximity to the channels or as inclusions encapsulated—quite notably enhances thermal stability when the device is under dynamic operation. Phase-change materials (PCMs) have become an interesting class of passive thermal management materials especially for managing intermittent and high-density heat loads in compact systems. Main characteristic is that they can absorb or release large quantities of latent heat associated with phase changes within a relatively small temperature range. From their research, it was revealed that these PCM-aided microchannel geometries can prolong the onset delay in temperature increase, keeping device temperatures in safe operating ranges even with instantaneous power spikes or pulsed operation regimes characteristic of high-frequency electronics, laser diodes, and processors. A principal concern is the inherently poor thermal conductivity of the majority of organic and paraffin-type PCMs, which can slow the rate of heat transfer into and out of the material, reducing the rate of phase change and overall cooling capacity To counter this, there have been attempts to look into composite PCMs, where materials of high conductivity like copper foams, carbon nanotubes, graphene, or metal oxide nanoparticles are incorporated into the PCM matrix to increase its thermal responsiveness. Nevertheless, these composites bring in complexity regarding material homogeneity, long-term stability, and cost.

In the quest for improved thermal performance of microchannel heat sinks (MCHSs), Chamoli et came up with a numerical investigation "Numerical optimization of design parameters for a modified double-layer microchannel heat sink," examines the effect of geometrical design on heat transfer efficiency and fluid flow behaviour. The redesigned device consists of two parallel layers of microchannels with staggered inlet and outlet configurations to provide more uniform coolant distribution and increase thermal spreading within the heat sink structure. used a multi-parameter optimization methodology through the implementation of computational fluid dynamics (CFD) simulations to investigate major design parameters including channel height, width, thickness of the fins, and the gap between the two layers. Identifying the shortcomings of standard single-layer MCHSs for handling very high heat fluxes, the authors designed a new double-layer structure for enhancing the effective surface area and promoting heat dissipation while not excessively raising the device's footprint.

The research identified that geometrical parameter optimization would result in a significant enhancement of thermal performance—higher heat removal rates with acceptable pressure drop levels. In addition, the authors emphasized the trade off between hydraulic and thermal performance, indicating that excessively narrow channels or dense fin structures, although beneficial for heat transfer, may be more flow-resistance and pumping power-consuming. It is a valuable reference for future experimental validation and practical applications of multi-layer microchannel structures in thermal management systems. The paper is a valuable guide for future experimental validations and engineering designs of multi-layer microchannel architectures in thermal management systems The findings of this work are particularly significant for high-heat-density applications such as power electronics, VLSI chips, and laser modules, where heat densities and constrictive space necessitate innovative cooling concepts.

CHAPTER 3

METHODOLOGY AND MODEL DESCRIPTION

In high-performance electronic devices such as semiconductors, power electronics, laser diodes, and LEDs, unwanted heat generation is a major issue that can heavily degrade performance, reliability, and life. Analytical treatment is performed through a representative unit cell model consisting of one microchannel together with the neighbour solid substrate material. The microscale geometry, however, greatly enhances the surface-area-to-volume ratio, providing for fast and localized heat extraction. By controlling the temperature of the device within operating specifications, the MCHS provides thermal stability, system efficiency, and component lifetime enhancement. The unit cell model is a computational effective approach to predict the intricate are important towards realizing the overall thermal performance of the heat sink, components tend to run at high power densities, which result in localized areas of significant thermal energy, which are usually referred to as high heat flux zones.

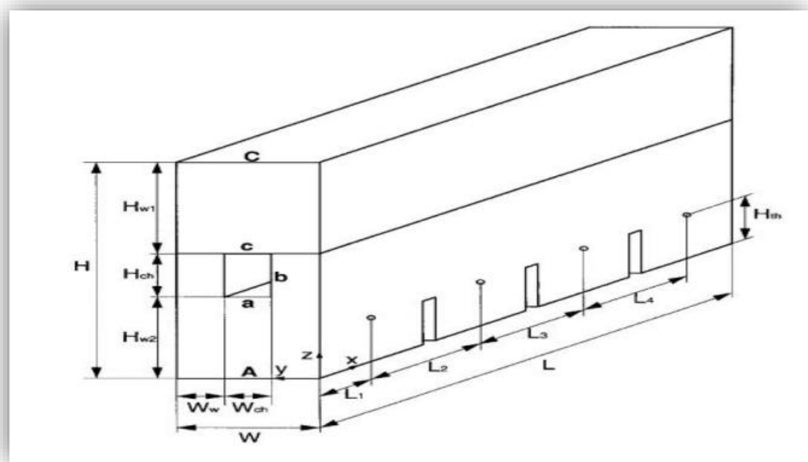


Fig. 3.1 A schematic diagram of the Unit Cell [3]

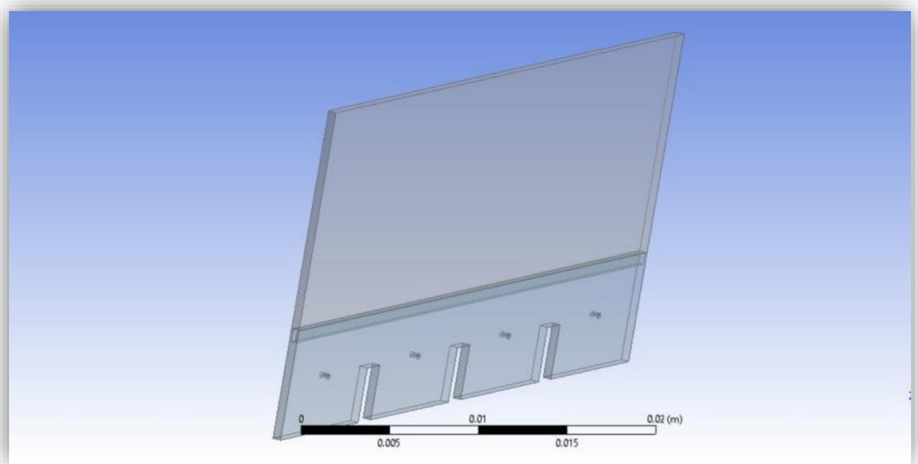


Fig. 3.2 3D view of the Unit Cell in Design Modele

The unit cell in this research is used to represent an individual microchannel implanted in an element of the enveloping solid substrate, which usually is the base material of the heat sink e.g., silicon, copper, or aluminium and in this case, copper is considered. This cell incorporates both the fluid region and the neighbouring solid region through which heat is transferred before being transferred to the fluid. Since the heat sink consists of several, identical parallel channels, thermal and hydrodynamic behaviour in one is representative for the rest. In the unit cell, comprehensive simulation of the conjugate heat transfer is conducted—involving conduction within the solid and convection within the fluid. This enables a detailed examination of important performance parameters like the local and average temperature fields, velocity profiles. Through integrating latest cooling technology with numerical modelling of high fidelity, microchannel heat sinks, which are computed using CFD, provide a practical and effective solution for thermal load management in contemporary high-density electronic and photonic systems.

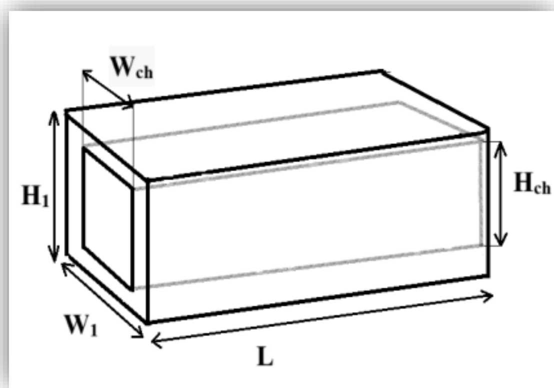


Fig. 3.3 A schematic diagram of the fluid domain

Table 3.1 Dimension of the Unit Cell

Parameters Sign	Parameters	Values (mm)
W_{ch}	Microchannel width	0.335
H_{TH}	Thermocouple height	3.01
H_{W1}	Height above microchannel	12.5
H_{W2}	Height below Microchannel	4.637
L	Length of Microchannel	48.664
H_{ch}	Microchannel Height	0.913

Table 3.2 Dimension of fluid domain with bumps

Parameters Sign	Parameters	Values (mm)
W_{ch}	Microchannel width	0.421
H_1	Thermocouple height	1.313
H_{ch}	Height of the microchannel	0.913
W_1	Height of the cover plate	0.821
L	Length of Microchannel	35.2

Table 3.3 Dimension of simple fluid domain.

Parameters Sign	Parameters	Values (mm)
W_{ch}	Microchannel width	0.235
H_{ch}	Height of the microchannel	0.750
L	Length of Microchannel	47

3.1 Material selection

Copper is extensively employed as the base material for microchannel heat sinks, and water is widely selected as the base coolant, because of their excellent thermophysical properties that substantially augment the heat transfer efficiency of cooling systems at the microscale. The following is the detailed description of why these materials are utilized [24]:

a) High Thermal Conductivity - Copper possesses one of the highest room temperature thermal conductivities among all commercially used metal (~385 W/m·K). High conductivity provides consistent temperature distribution and reduces thermal resistance within the solid realm.
b) Mechanical Strength and Manufacturability - Copper has very good mechanical properties, which qualify it for the precise micromachining or etching processes necessary to develop microchannels.
c) Corrosion Resistance (with Proper Treatment) - Though pure copper may be prone to corrosion—particularly in water-cooling applications—the corrosion resistance of copper can be enhanced through passivation, surface coating,
d) Compatibility with Semiconductor Substrates - Copper is widely applied in the electronics industry and is compatible with most packaging materials and interface technologies employed in VLSI chips, power electronics, and LEDs.

Table 3.4 Material properties of aluminium and coolant

Material	Density (kg/m^3)	Specific heat capacity (J/kg. K)	Thermal conductivity (W/m. K)
Copper	8978	381	387.6
Water	998.2	4182	0.6

Water is the most widely used coolant in microchannel heat sinks because of its outstanding heat transfer characteristics [25]:

a) High Specific Heat Capacity - The specific heat capacity of water is about 4.18 kJ/kg·K, which is one of the highest values among liquids and enables it to absorb a large quantity of heat with very little rise in temperature.

b) High Thermal Conductivity - The thermal conductivity of water (~0.6 W/m·K) is much larger than most other liquids, and thus allows easy heat transfer from the channel walls to the bulk fluid.

c) Low Viscosity and Good Flow Characteristics - Water has a low viscosity, and it is advantageous for having greater flow rates at lower pressure drops. It is especially advantageous in microchannels.

d) Environmentally Friendly and Non-Toxic - Water is safe, easily accessed, non-toxic, and non-flammable, making it the best for laboratory and commercial purposes.

3.2 Numerical Formulation and Boundary Conditions

Conjugate heat transfer, which involves consideration of conduction in the solid wall and convection in the fluid domain, is simulated using the finite volume method (FVM) in ANSYS Fluent. Simplifying assumptions are included to establish the governing equations for fluid flow and heat transfer in the unit cell. These assumptions are as follows [26] -

- Steady-state condition with no influence on flow;
- Non-compressible fluid;
- Under streamline flow conditions;
- Radiation heat transfer is minimized;
- All solid and liquid properties are treated as constant, except for water viscosity;
- Natural convection of air trapped in the heat sink's deep slots is minimized.

The conjugate heat transfer is simulated here considering heat transfer—considering conduction through the solid walls and convection through the fluid domain—employing the finite volume method (FVM) as applied in ANSYS Fluent. Both types of heat transfer are coupled with continuity conditions for temperature and heat flux across the interface of the solid and fluid. These conditions are stated mathematically as where $T_{s,c} = T_{f,c}$ represents (temperature continuity) while K_s (∂T/∂n) s, C = K_f (∂T/∂n) f, C represents (heat flux continuity) [27].

$$Ts, c = Tf, c \tag{3.1}$$

where $T_{s,c}$ and $T_{f,c}$ are the temperatures of the solid and fluid at the interface and,

$$Ks \ (\frac{\partial y}{\partial x})s, C \ = \ Kf \, (\frac{\partial y}{\partial x})f, C \tag{3.2}$$

where K_s and K_f are the thermal conductivities of the solid and fluid, respectively ∂T/∂n represent temperature gradient normal to the interface.

In Computational Fluid Dynamics (CFD) simulations of microchannel heat sinks, three primary sets of equations control the physical behaviour of fluid flow and heat transfer. These equations serve as the basis to simulate the physical behaviour of the coolant (such as water) in motion through the microchannels and thermally interacting with the solid walls usually copper or silicon (and in this case copper is considered) [28] [29].

3.2.1. Continuity Equation

Conservation of mass, this equation is responsible for conserving mass in the fluid domain, i.e., fluid cannot simply appear or vanish. This formula says that the net flow of volume into any small volume within the fluid must be equal to the net flow of volume out, it keeps the fluid balanced and the formula mathematically represented as [17]

$$\nabla \cdot V = 0 \tag{3.3}$$

where V (u, v, w) is the velocity vector in the x, y, and z directions

3.2.2. Momentum Equation

This formula says that the net flow of volume into any small volume within the fluid must be equal to the net flow of volume out and explains how momentum varies in a fluid because of pressure forces and viscous action and the formula mathematically represented as [26]

$$\rho\,(u\,\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\,\frac{\partial u}{\partial z}) = -\,\frac{\partial \rho}{\partial x} + \mu\,(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}) \tag{3.4}$$

where ρ is density of fluid, μ is dynamic viscosity, p is pressure, $(u.\nabla)u$ is convective acceleration (change in velocity due to motion) and $\nabla^2 u$ = viscous diffusion term (resistance to flow due to viscosity).

3.2.3. Energy Equation

This equation controls the transfer of heat within the system, both conduction through the solid and convection through the fluid. This equation indicates that the fluid convects heat along its flow (convective term on the left), and it diffuses heat (conductive term on the right). It is critical to predicting how heat is extracted from the hot walls and the formula mathematically represented as [30]

$$\rho \, cp \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z}\right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \quad (3.5)$$

where cp is specific heat capacity of the fluid, T is temperature and k is thermal conductivity of the fluid and solid.

These equations are numerically solved by CFD codes, generally by the finite volume method (FVM), which discretizes the domain into small control volumes and iteratively solves the equations at each node or cell. The heat fluxes taken were $100 W/cm^2$, $150 W/cm^2$, and $200 W/cm^2$, with varying Reynolds numbers and fluent containing Al_2O_3 weight compositions of 1.5% and 2.5%. For fluid domains, the fluid's inlet temperature is set to 22 °C, and flow is fully developed through the channel. The SIMPLE algorithm is then applied to solve equations.

3.3 Boundary Conditions

It is critical to define proper boundary conditions for the fluid domain in order to simulate the heat transfer and fluid flow accurately. The boundary conditions represent the physical boundary conditions and operating conditions of the real system, especially in cases with high heat-generating devices like semiconductors, power electronics, laser diodes, and LEDs. This condition includes defining a velocity profile that depends on varying Reynolds number from 100 to 900 and a constant inlet temperature, taken as 295 K according to operating conditions. Care needs to be taken in such that at the inlet, the flow is laminar and hydrodynamically developing, particularly since microchannel flows tend to be operated in the low Reynolds number regime because of their small hydraulic diameters [32]. Velocity for different Reynolds number is to be calculated by the formula mathematically represented as

$$Re = \frac{\rho D v}{\mu} \tag{3.6}$$

where,

$$D = \frac{2A}{P} \tag{3.7}$$

and ρ is density of fluid, μ is dynamic viscosity, v is velocity of the fluid, D is the diameter of inlet, A is cross section area of inlet and P is perimeter of inlet [31].

Thermally, walls are addressed based on conjugate heat transfer concepts. Heat transmitted through solid microchannel walls is transferred to fluid through convection. A condition of constant heat flux is generally employed in simulating electronic devices with specified power dissipation, here three different values of heat flux is utilized over bottom of heat sink which are $100W/cm^2$, $150W/cm^2$, and $200W/cm^2$. Furthermore, in simulations employing a unit cell strategy—in which only a representative section of the complete microchannel array is simulated to conserve computational resources—a symmetry boundary condition is imposed on the sidewalls of the fluid domain [30]. Together, these boundary conditions constitute the basis for the simulation of the fluid flow and heat transfer in a microchannel heat sink. They make the simulation realistic with respect to operating conditions and allow important analysis of thermal performance across different geometries and coolants [31]. As here two different fluid domains were considered, one of simple geometry and another with bumped geometry. Also fluent contain nano particles weight percentage of 1.5% and 2.5%.

CHAPTER 4

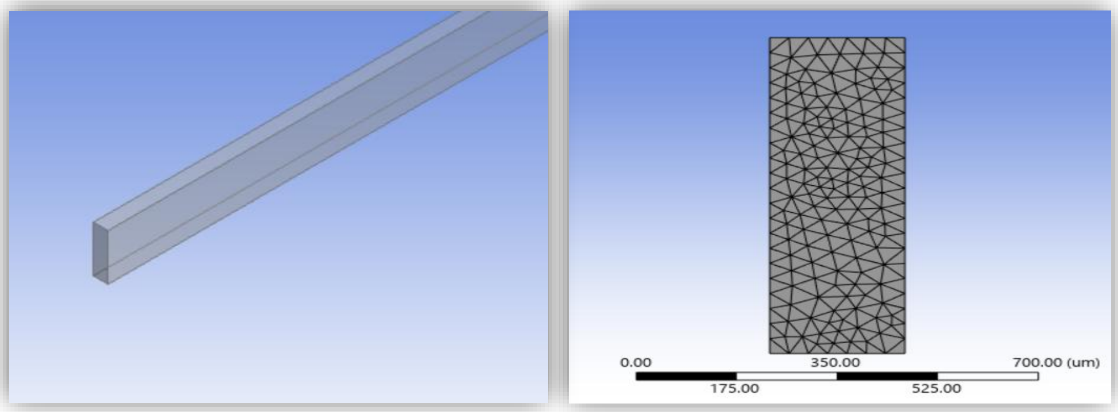
VALIDATION AND GRID INDEPENDENCE TEST

4.1 Mesh Generation and Grid Independence Test

In the context of Computer-Aided Engineering (CAE), which includes simulation software for other engineering analyses (such as Computational Fluid Dynamics - CFD, Finite Element Analysis - FEA, and heat transfer simulations), mesh creation and the following grid independence test form core steps. Mesh generation, or discretization or meshing, is the process of partitioning the continuous geometric domain of an engineering problem into a large number of small, discrete, interacting sub-regions known as "elements" or "cells". Physical phenomena (such as fluid flow, heat transfer, structural deformation) are characterized by continuous partial differential equations. Numerical methods that are the heart of CAE software convert these continuous equations into a set of algebraic equations that can be solved computationally. The choice of element type depends on the geometry, physics, and solver capabilities:

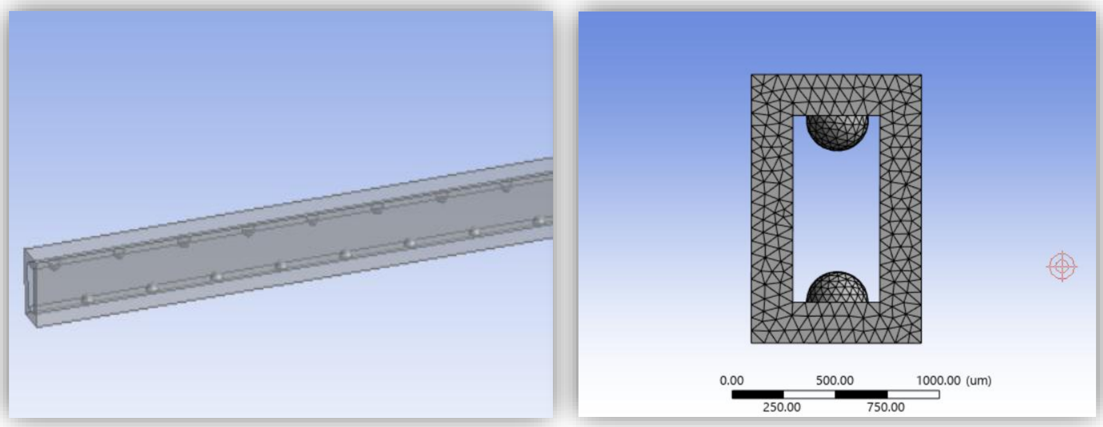
a) 2D Meshes - triangular mesh a versatile for intricate geometries, simple to create and quadrilateral mesh may be able to provide better results for certain flow types, particularly with structured meshing.
b) 3D Meshes - tetrahedral mesh a very flexible for intricate 3D geometries, automatic generation is typical whereas hexahedral mesh generally used in preference for greater accuracy and efficiency in areas where structured meshing is feasible (e.g., boundary layers) and polyhedral mesh a fairly new variety, providing a good compromise of accuracy, efficiency, and flexibility for intricate geometries.

The mesh quality has a direct influence on the simulation convergence and accuracy. CAE tools allow the analysis of: The ratio of the longest side to the shortest side of an element. Large aspect ratios can result in inaccuracies, particularly if the gradients are in the perpendicular direction to the long side. A measure of how twisted an element is from its perfect form e.g., equilateral triangle, ideal cube high skewness reduces accuracy. Smoothness or grading which is how well the cell size varies.



(a) (b)

Fig. 4.1. shows (a) simple fluid domain, (b) simple fluid domain mesh size with 252519 nodes and 1268146 elements size



(a) (b)

Fig.4.2. shows (a) geometry of fluid domain with bumps of 0.2mm in radius (b) bumped fluid domain mesh size with 249346 nodes and 1192671 elements

After generating a mesh, one needs to carry out a grid independence test. This is a rigorous investigation to check that the solution obtained through the simulation is no longer influenced by additional refinement of the mesh to any significant extent. It gives assurance that the solution obtained through computation is a true reflection of the physical problem at hand, and not a function of the resolution of the mesh. Asymptotic behaviour which means to look at the trend on graph where in the beginning, when the mesh is being refined, the output parameter will probably change dramatically. When the mesh is fine enough, however, the output parameter value should start asymptoting towards a constant value. One general requirement for grid independence is where the difference in the major output parameter between one refinement and the previous one is less than a specified tolerance. The grid at which the solution is grid independent (or perhaps the next finer one, to be sure) is selected for the ultimate, production-level computation.

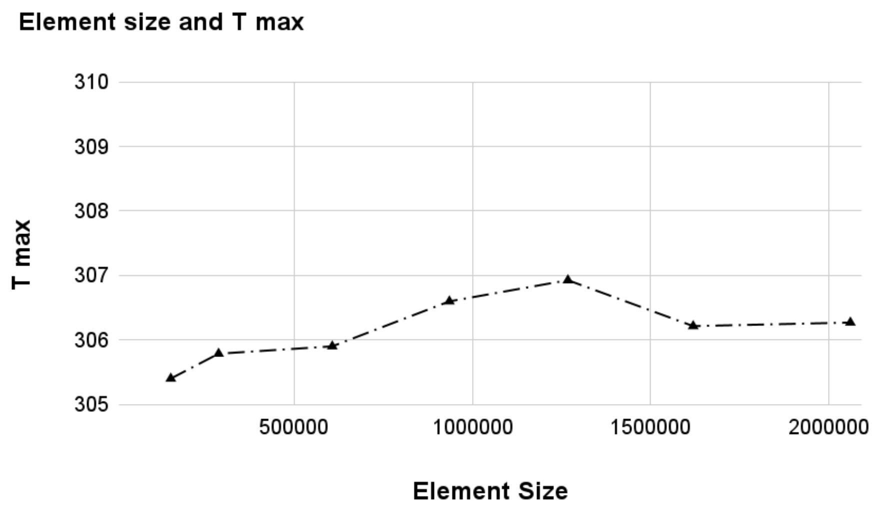


Fig. 4.3 Grid independence test

From the above graph it can be concluded that for simple fluid domain mesh size with 252519 nodes and 1268146 elements size, graph shows its maximum value then after it tends to converge. That means on extremely coarse meshes the numerical solution has a great deal of discretization error. This is due to the fact that the continuous physical equations are being approximated on a discrete mesh. The location where the graph indicates a maximum value (252,519 nodes and 1,268,146 elements) indicates, that as the mesh is progressively refined from being very coarse, the discretization error reduces. This maximum may be one such overshoot where the solution goes beyond the actual value temporarily before stabilizing. This particular mesh resolution might be the first instance where important physical details e.g. small re-circulation regions or steep temperature gradients are adequately resolved.

The important aspect is that having reached this peak, the graph subsequently tends to converge. As we further refine the mesh beyond 252,519 nodes, the size of the individual elements continues to decrease. This enables the numerical scheme to better resolve the velocity, temperature, and pressure gradients in the flow domain. With decreasing discretization error, the numerical solution increasingly approximates the exact, continuous solution of the governing partial differential equations. The following convergence establishes that further increasing the density of the mesh gives diminishing returns in the accuracy of the solution, confirming that the selected mesh will deliver trustworthy results.

4.2 Validation

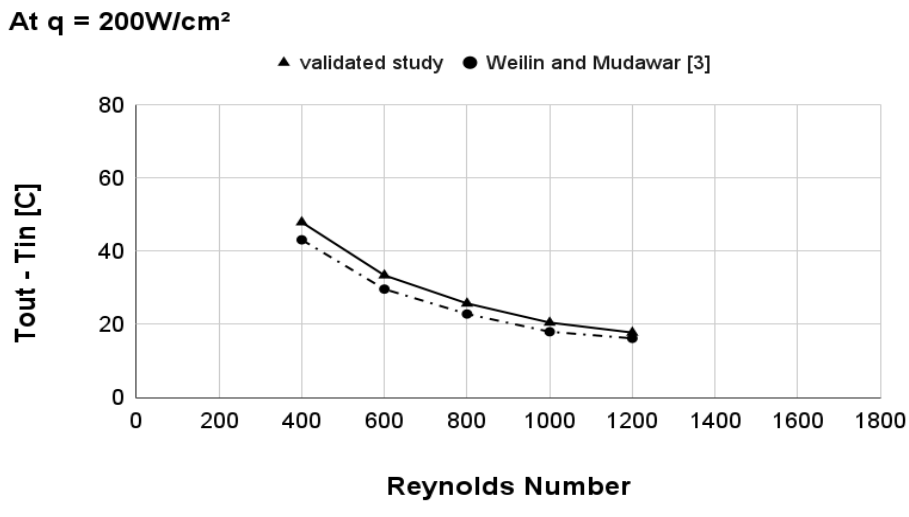


Fig. 4.4 Variation of change in maximum temperature difference of actual and validated results with increase in velocity

Figure 4.4 and table 4.1 displays the variation in the maximum temperature difference between the actual and validated results of the cell surface temperature as velocity increases, calculating the percentage inaccuracy.

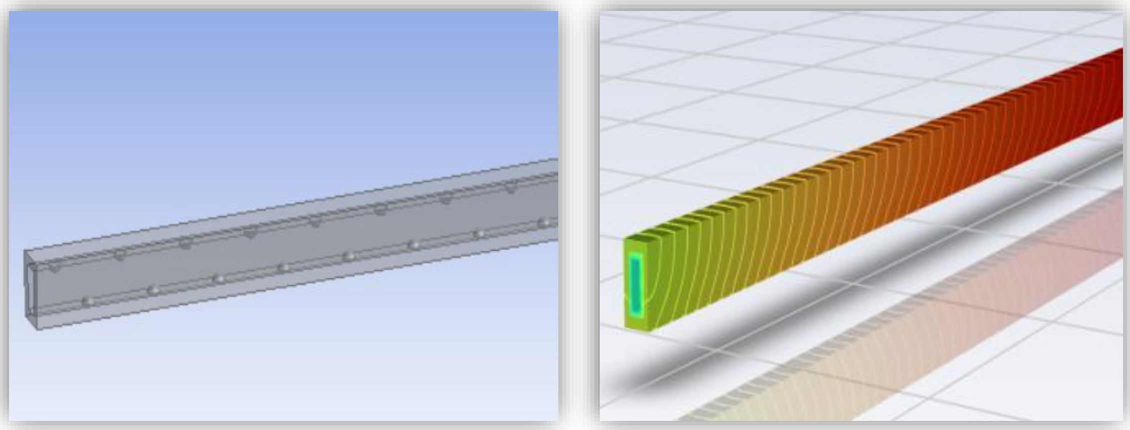
Table 4.1 Variation $\Delta \mathbf{T_{max}}$, actual & $\Delta \mathbf{T_{max}}$, validated with velocity to calculate % error

S.No.	Re	T_{outlet}, actual (K)	T_{outlet}, validated (K)	Error%
1	200	57.01	55.01	26.6
2	400	43.09	47.893	11.1
3	600	29.62	33.391	12.35
4	800	22.82	25.707	12
5	1000	17.97	20.47	13
6	1200	16.17	17.749	9.7

CHAPTER 5

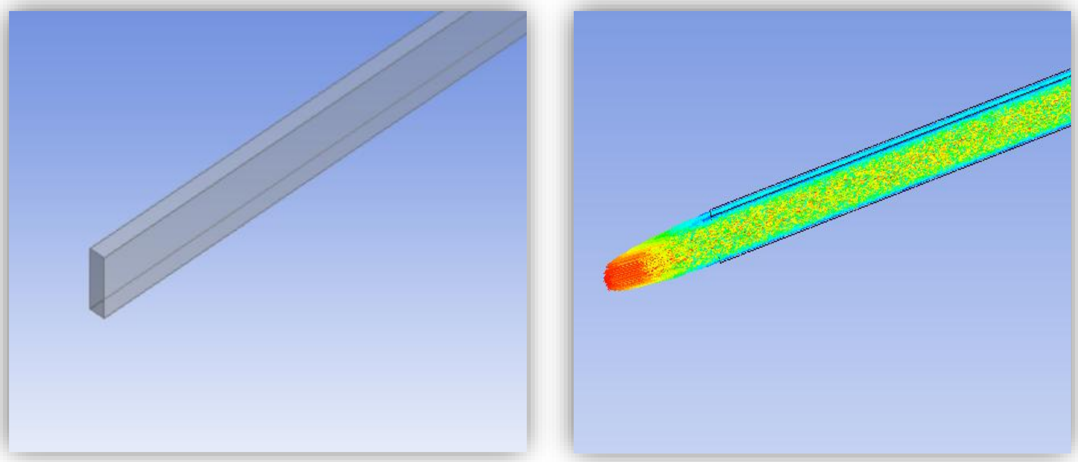
RESULT AND DISCUSSION

Taking heat fluxes taken were 100W/cm^2, 150W/cm^2, and 200W/cm^2, with varying Reynolds numbers and fluent containing Al_2O_3 weight compositions of 1.5% and 2.5%. For fluid domains, the fluid's inlet temperature is set to 22 °C, and flow is fully developed through the channel. The SIMPLE algorithm is then applied to solve equations. This section provides a comprehensive discussion of the results of the simulation for a microchannel heat sink under varying conditions of heat flux, Reynolds number, and concentration of nanoparticles.



(a) (b)

Fig.5.1 shows (a) bumped fluid domain geometry (b) temperature contour

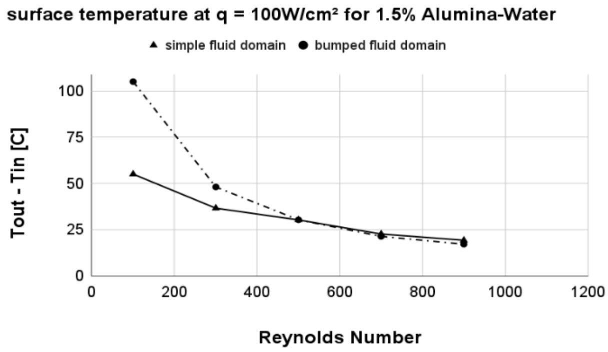


(a) (b)

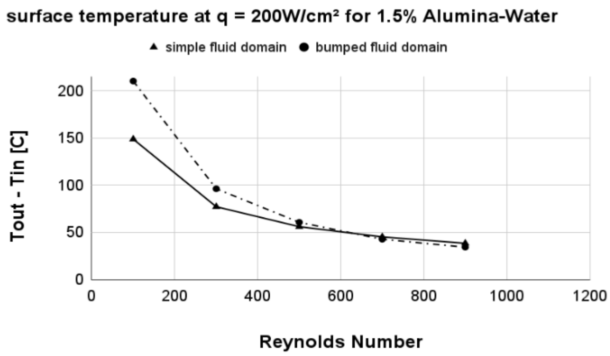
Fig.5.2 shows (a) simple fluid domain geometry (b) shows temperature contour

5.1 Surface Temperature Comparison of Fluent with AL_2O_3 1.5% Weight Composition

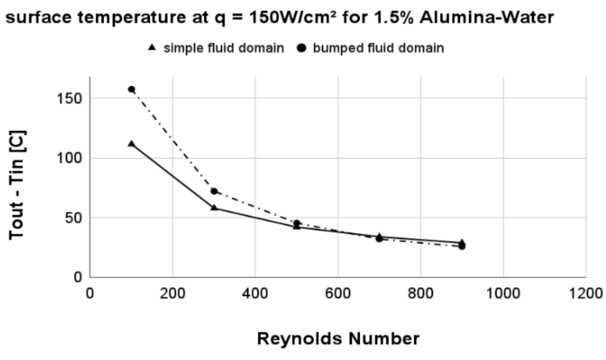
Figure 5.3 presents the surface temperature profile for various Reynolds numbers with 1.5% Al_2O_3 concentration for both geometries. With increasing Reynolds number, there is a noticeable reduction in surface temperature in both geometries. This trend can be explained by increased convective heat transfer rates for higher flow speeds. At higher Reynolds numbers, the fluid travels faster through the channel, decreasing its residence time as a consequence, while the fluid moves more quickly and extracts heat more effectively per unit time, it spends less time for extracting heat from the wall in any particular area. These laminar boundary layer disturbances induce increased fluid mixing, augment the interaction between the warmer channel wall and the colder fluid core, and decrease thermal boundary layer thickness. By contrast, the plane geometry does not have such surface topographies, promoting more thermally stratified flow where fluid in the vicinity of the centerline receives less heat. Nevertheless, the bumped microchannel still has an advantage in low to moderate flow regimes, which is crucial in compact electronics cooling where minimal pumping power is desired.



(a)



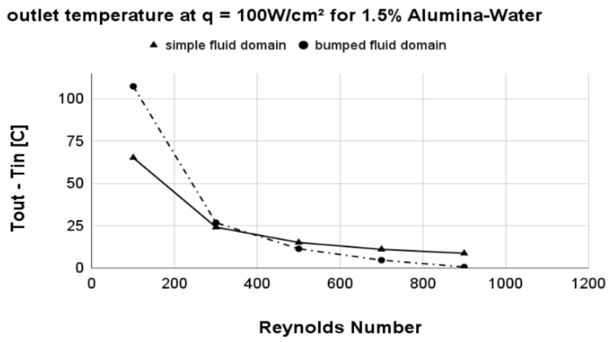
(b)



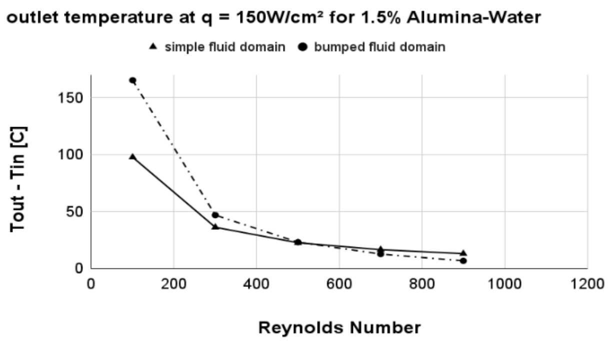
(c)

Fig. 5.3 Comparison of surface temperature at varying flux for 1.5% of weight composition of Alumina nanoparticles in water

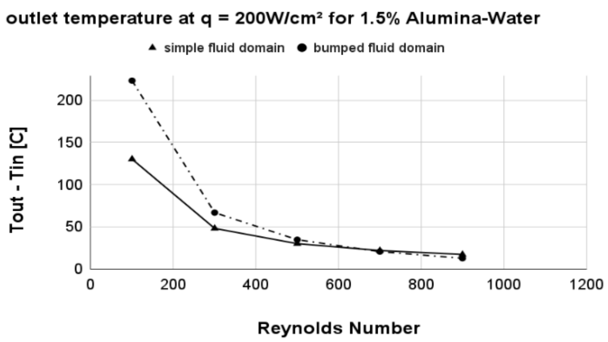
5.2 Output Temperature Comparison of Fluent with AL_2O_3 1.5% Weight Composition



(a)



(b)



(c)

Fig. 5.4 Comparison of outlet temperature at varying flux for 1.5% of weight composition of Alumina nanoparticles in water

Figure 5.4 shows the variation of outlet temperature in relation to operating parameter changes—namely applied heat flux and Reynolds number—at a constant nanoparticle concentration of 1.5% Al_2O_3 in water. The expected trend is observed, with the working fluid's outlet temperature rising as the heat flux increases. This is logical because a higher heat flux means that there is more thermal energy transferred from the hot channel walls to the fluid. The nanofluid takes up this extra energy, leading to a higher fluid temperature upon exiting the microchannel the outlet temperature, on the other hand, has a declining tendency with rising Reynolds number. Under smaller Reynolds numbers, the fluid flows at a slower speed, providing sufficient time for heat transfer between the channel walls and the fluid. More efficient absorption of heat by the fluid in the bumped geometry, which is mainly due to the increased thermal mixing caused by surface features.

This behaviour verifies that structuring near the surface, i.e., adding bumps or obstructions, improves considerably the heat transfer efficiency of microchannel heat sinks, particularly for the low Reynolds number regime. Finally, outlet temperature behaviours shown in Figure 5.4 demonstrate the vital interdependence among flow dynamics, geometry, and thermal loading. They confirm the effect of microchannel geometry alteration and nanofluid choice in improving cooling performance, especially for low-flow, high-heat conditions.

5.3 Pressure Comparison of Fluent with AL_2O_3 1.5% Weight Composition

Pressure drop is a key variable in the thermal-hydraulic characteristics of microchannel heat sinks since it impacts directly on the pumping power needed and the general efficiency of the cooling system. The trend is significant since geometric features like bumps are commonly expected to be pressure drop increasing due to increased flow disturbances, additional surface area, and re-circulation or separation zones. Simplified bump design where bumped microchannel employed probably has optimally sized and positioned bumps that ensure smoother flow paths and less hydraulic resistance in general. Here increased flow mixing and boundary layer thickness decreases, which are surface characteristics that will normally enhance mixing or turbulence, they can also serve to decrease the hydrodynamic boundary layer development length. Hence creating a thinner boundary layer along most of the channel length, decreasing the viscous friction contribution to pressure drop. Whereas in flow redistribution and secondary vortices the bumps in the microchannel can create desirable secondary vortices that redistribute the velocity profile.

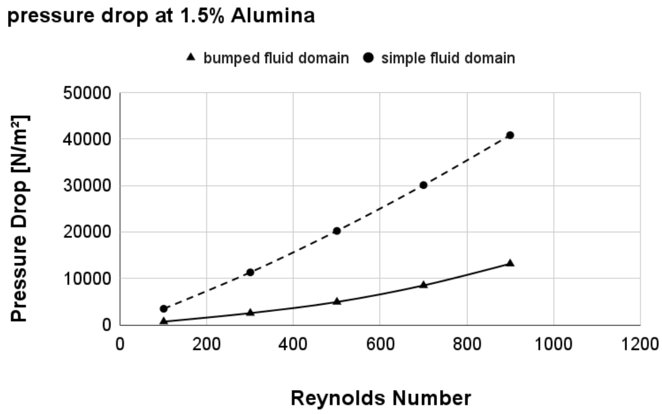
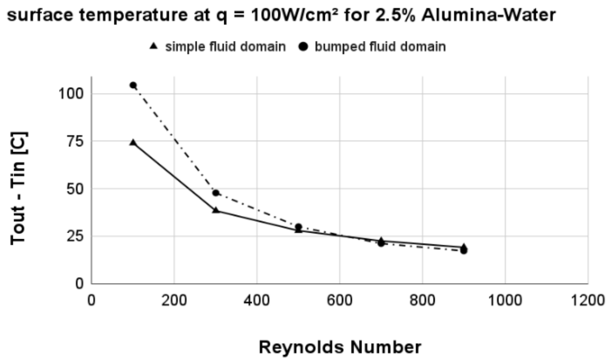


Fig. 5.5 Pressure drop variation concerning Reynolds number in the fluid domain of simple shape and fluid domain with bumps at 1.5% weight composition of nanoparticles

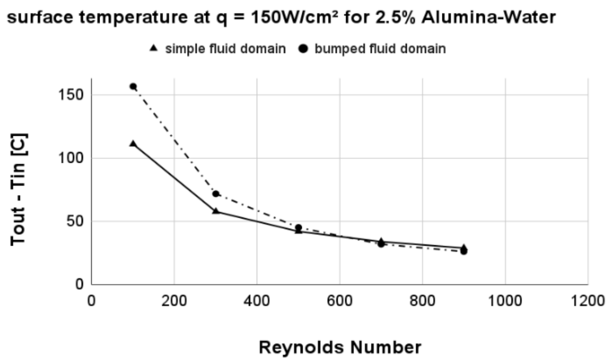
In conclusion, figure 5.5 shows that carefully designed surface optimizations, like bumps, not only enhance heat transfer (as observed in previous figures) but also provide hydraulic benefits through pressure drop reduction. This advantage not only makes bumped microchannels extremely promising for thermal management in small electronic systems, where both efficient heat dissipation and minimal pumping power are essential.

5.4 Surface Temperature Comparison of Fluent with AL_2O_3 2.5% Weight Composition

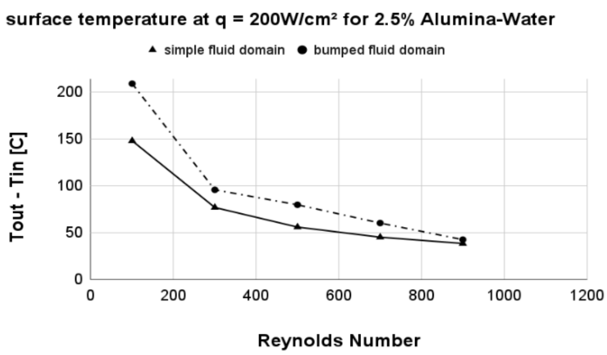
The plots show that for both plain and bumped fluid domains, the Reynolds number has a great effect of decreasing the temperature difference to reflect better heat transfer performance.



(a)



(b)



(c)

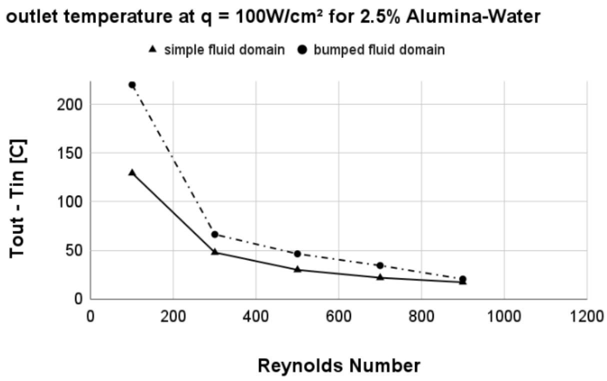
Fig. 5.6 Comparison of surface temperature at varying flux for 2.5% of weight composition of Alumina nanoparticles in water

A higher Reynolds number indicates an increase in flow velocity, which reduces the residence time of the coolant within the microchannel. Consequently, the fluid gets less time to take up heat from the hot channel walls, and thus, the increase in fluid temperature from inlet to outlet is smaller. Regardless of the flow rate, the fluid takes up more heat, and the temperature difference between the outlet and the inlet is higher, increased thermal input leads to larger temperature gains by the fluid, under steady-state conditions and with no losses. Higher $T_{out} - T_{in}$ in the bumped regime means the fluid takes up more heat per unit mass, possibly interpreted as improved heat capture. The bumped shape, though it localizes mixing and may increase surface area, can also cause higher flow resistance. Such resistance can decrease the effective flow rate in local areas or enhance flow maldistribution, especially at low Reynolds numbers. The bumps perturb the thermal boundary layer, best reducing it and increasing heat transfer.

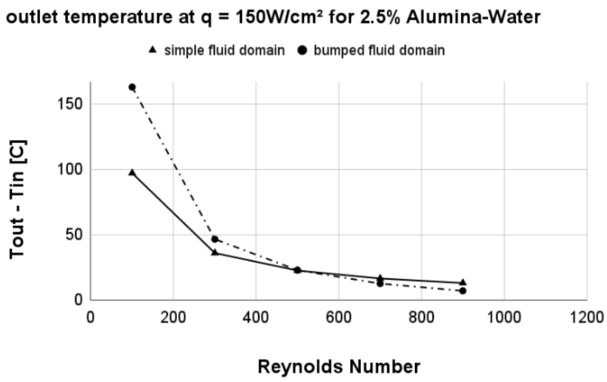
Overall, the increased $T_{out} - T_{in}$ seen in the bumped fluid regime most probably stems from a combination of greater fluid mixing, changed thermal boundary layers, and perhaps increased flow resistance. Although it will superficially indicate enhanced heat absorption, this measurement by itself does not guarantee better thermal performance.

5.5 Output Temperature Comparison of Fluent with AL_2O_3 1.5% Weight Composition

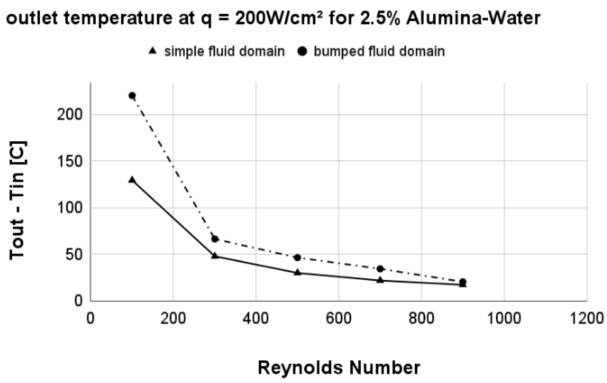
The initial observation from Figure 5.7 is that the difference in outlet temperature (Tout – Tin) rises in proportion with heat flux. When the applied heat flux is higher, greater thermal energy is being transferred to the fluid per unit time and area. Assuming constant mass flow rate, this increased input of energy results in the fluid's temperature rising more significantly, thus increasing. the observation that at higher Reynolds numbers the temperature profiles for both the bumped and simple geometries are converging. It suggests that the thermal performance differences resulting from geometry diminish as the flow evolves into higher velocity regimes.



(a)



(b)



(c)

Fig. 5.7 Comparison of outlet temperature at varying flux for 2.5% of weight composition of Alumina nanoparticles in water

The initial observation from Figure 5.7 is that the difference in outlet temperature (Tout – Tin) rises in proportion with heat flux. When the applied heat flux is higher, greater thermal energy is being transferred to the fluid per unit time and area. Assuming constant mass flow rate, this increased input of energy results in the fluid's temperature rising more significantly, thus increasing. the observation that at higher Reynolds numbers the temperature profiles for both the bumped and simple geometries are converging. It suggests that the thermal performance differences resulting from geometry diminish as the flow evolves into higher velocity regimes.

With an increase in Reynolds number, the velocity of the fluid is higher, and the residence time of the fluid in the channel is shorter. The reduced time for thermal interactions implies that any fine differences from channel geometry have less chance to impact the temperature profile significantly. Consequently, the thermal response of the fluid becomes dominated by its overall motion and not by localized geometrical features. This diminishes the relative contribution of increments such as bumps or surface irregularities.

5.6 Pressure Comparison of Fluent with AL_2O_3 1.5% Weight Composition

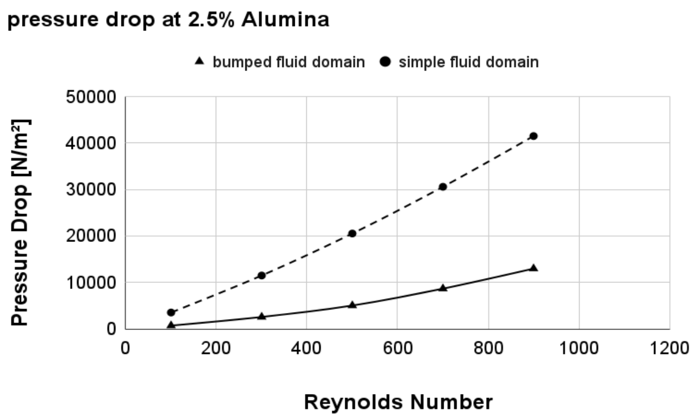


Fig. 5.8 Pressure drop variation concerning Reynolds number in the fluid domain of a simple shape and the fluid domain with bumps at 2.5% weight composition of nanoparticles

Figure 5.8 shows the pressure drop behaviour of the simple and bumped microchannel geometries at a concentration of 2.5% nanoparticles. The addition of nanoparticles into a base fluid (e.g., water) alters its thermo-physical properties. One of the most notable alterations is an increase in the viscosity. Increased viscosity results in increased internal fluid friction, which translates to an elevated pressure loss as the fluid flows through the microchannel. At elevated concentrations, nanoparticles start to collide more regularly, possibly resulting in micro-clustering. Interestingly, although the bumped surface poses more complicated flow paths (which could normally increase resistance), the pressure drop in the bumped channel does not exceed that of the uncomplicated geometry by a significant amount. The bumped geometry still has thermal benefits and thus remains appropriate when high-efficiency cooling in a compact manner is required—e.g., in VLSI chips, power electronics, or LED heat management. Pressure drop to thermal performance balance needs to dictate pump selection, material compatibility, and choice of coolant. In summary, although a concentration of 2.5% nanoparticles increases the pressure drop to a small extent due to elevated viscosity, the thermal performance improvements sufficient justification for the compromise. The bumped geometry does achieve high heat dissipation efficiency by effectively utilizing improved surface interaction and fluid mixing.

5.7 Future Scope

a) Transient and Pulsating Flow Conditions - The present work is restricted to steady-state simulations. Work in the future must include transient simulations to elucidate performance under actual time-variable thermal loads, such as those encountered in high-performance computing or electric vehicle use.
b) Optimization via Machine Learning and AI Methods - With many design parameters (flow rate, concentration, bump pattern, channel shape, size), data-driven optimization via neural networks, genetic algorithms, or surrogate modelling can be utilized to forecast and optimize optimal configurations.
c) Application of Phase-Change Materials (PCMs) in Bumped Geometries - Incorporating PCMs in the wall or cavities of bumped microchannels can provide

dual-mode cooling: one by convection and the other by latent heat absorption, thus leveling down temperature spikes in oscillating thermal environments.

d) Miniaturization and 3D Printing for Custom Geometries - Future designs will be able to utilize more sophisticated fabrication methods like additive manufacturing to produce complex and non-standard geometries that will enable better surface area improvement and flow paths designed to specification

CHAPTER 6

CONCLUSION

6.1 Conclusion

In this research, the thermal-hydraulic performance of microchannel heat sinks are the key factors in the improvement of compact heat exchangers thermal management capacity for high heat-flux devices like semiconductors, power electronics, and LEDs through comparison of two different channel geometries: a simple rectangular microchannel and a modified bumped channel. Both surface and exit temperatures decrease as Reynolds number goes up This is mainly because increasing flow rates enhance convective heat transfer, hence decreasing the thickness of thermal boundary layer However, temperature gain decreases as Reynolds number gets larger, signifying lower thermal gradient resulting from shorter fluid residence time within the channel.

The bumped microchannel design performs better than the traditional straight channel in reducing surface temperature and enhancing heat transfer, especially under low flow rates. This is due to enhanced surface area and created turbulence, which enhance fluid mixing and energy transfer. Enhanced heat dissipation capability also the results from the application of Al_2O_3-water nanofluids with performance improvements of 25–37% depending on concentration and operating conditions. Although higher nanoparticle concentrations increase pressure drops slightly, thermal performance benefits exceed the hydraulic costs. In the bumped geometry at 1.5% Al_2O_3 concentration, the surface temperature was improved by 35% and 25% under the flux ranges of 100-150 W/cm^2 and 150-200 W/cm^2, whereas the output temperature improved by 35% and 37%, respectively indicating more effective absorption by the fluid. Whereas at 2.5% Al_2O_3 concentration, the surface temperature improvements of 33% and 25%, and the output temperature improvements of 35% and 33%, under the same conditions were observed.

Pressure drop analysis shows that geometric modifications, rather than the conventional expectation, do not necessarily enhance hydraulic resistance, optimized structures like the bumped structure can minimize flow resistance and promote beneficial pressure characteristics. Here total hydraulic resistance is alleviated even though the flow path is more complex, hence suggesting that the smart geometric design can minimize pumping power which demands without sacrificing any heat transfer. Increased nanoparticle concentration (1.5% to 2.5%) led to a marginal increase in pressure drop, that is, 1.4% for simple geometry and 1.9% for bumped geometry.

Comparative study of the geometries was found to indicate that the bumped channel invariably had better thermal performance. At low Reynolds numbers, the bumped geometry produced higher outlet temperatures and lower surface temperatures because of the augmented surface area and the formation of secondary flows and turbulence caused by the bumps Another significant finding is the similarity in temperature profiles for both geometries at high Reynolds numbers. This suggests that, in the high-turbulent regimes, microchannel geometry has a lesser impact as the flow is increasingly dominated by inertia over wall effects or induced mixing. In conclusion, the combination of engineered microchannel geometries with nanofluid coolants gives a strong and effective approach to compact and high-performing thermal management systems. Here bumped geometry with 1.5–2.5% Al_2O_3 nanofluids provides a balanced solution with considerable thermal benefits and negligible hydraulic detriments which is a good contender for sophisticated cooling in microelectronics and photonic devices.

REFERENCES

1. Tuckerman, David B., and Roger Fabian W. Pease. "High-performance heat sinking for VLSI." *IEEE Electron device letters* 2.5 (2005): 126-129.
2. Phor, Lakshita, et al. "Al2O3-water nanofluids for heat transfer application." *MRS Advances* 4.28 (2019): 1611-1619.
3. Qu, Weilin, and Issam Mudawar. "Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink." *International journal of heat and mass transfer* 45.12 (2002): 2549-2565
4. Jain, Harshit, Kanishk Yadav, Kapeesh Kumar, and M. Zunaid. "Performance analysis of inclined microjet impingement heat sink with porous medium." Materials Today: Proceedings, 56.6, 3604-3611, (2022).
5. Bhatt, Himanshu, Himanshu Gola, M. Zunaid "Variation of heat transfer characteristics of a microchannel heat sink with geometry." Materials Today: Proceedings, 56.2, 936-942, (2022).
6. Kandlikar, Satish, et al. *Heat transfer and fluid flow in minichannels and microchannels*. elsevier, 2005.
7. Tiwari, Raj Kamal, and Manab Kumar Das. "Heat transfer augmentation in a two-sided lid-driven differentially heated square cavity utilizing nanofluids." *International Journal of heat and Mass transfer* 50.9-10 (2007): 2002-2018.
8. Narumanchi, Sreekant, et al. "Thermal interface materials for power electronics applications." *2008 11th Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems*. IEEE, 2008.
9. Colgan, Evan G., et al. "A practical implementation of silicon microchannel coolers for high power chips." *IEEE transactions on components and packaging technologies* 30.2 (2007): 218-225.
10. Narendran, Nadarajah, et al. "Solid-state lighting: failure analysis of white LEDs." *Journal of Crystal Growth* 268.3-4 (2004): 449-456.
11. Pop, Eric, Sanjiv Sinha, and Kenneth E. Goodson. "Heat generation and transport in nanometer-scale transistors." *Proceedings of the IEEE* 94.8 (2006)

12. Das, Sarit K., et al. *Nanofluids: science and technology*. John Wiley & Sons, 2007.
13. Hajialibabaei, Mahsa, and M. Z. Saghir. "A critical review of the straight and wavy microchannel heat sink and the application in lithium-ion battery thermal management." *International Journal of Thermofluids* 14 (2022): 100153.
14. Sui, Y., P. S. Lee, and C. J. Teo. "An experimental study of flow friction and heat transfer in wavy microchannels with rectangular cross section." *International journal of thermal sciences* 50.12 (2011): 2473-2482.
15. Chai, Lei, et al. "Heat transfer enhancement in microchannel heat sinks with periodic expansion–constriction cross-sections." *International Journal of Heat and Mass Transfer* 62 (2013): 741-751.
16. Peng, X. F., G. P. Peterson, and B. X. Wang. "Frictional flow characteristics of water flowing through rectangular microchannels." *Experimental Heat Transfer An International Journal* 7.4 (1994): 249-264.
17. Hetsroni, G., et al. "Heat transfer in micro-channels: Comparison of experiments with theory and numerical results." *International Journal of Heat and Mass Transfer* 48.25-26 (2005): 5580-5601.
18. Luan, Y., & Li, Y. (2014). Experimental investigation of thermal performance of a microchannel cooling system for high-power laser diode. Applied Thermal Engineering, 71(2), 129–138
19. Garimella, Suresh V., and Choondal B. Sobhan. "Transport in microchannels-a critical review." *Annual review of heat transfer* 13 (2003).
20. Zhang, Peijing, et al. "miR-205 acts as a tumour radiosensitizer by targeting ZEB1 and Ubc13." *Nature communications* 5.1 (2014): 5671.
21. Gamrat, Gabriel & Favre-Marinet, Michel & Person, S.. (2009). Modelling of roughness effects on heat transfer in thermally fully-developed laminar flows through microchannels. International Journal of Thermal Sciences - INT J THERM SCI. 48. 2203-2214. 10.1016/j.ijthermalsci.2009.04.006.
22. Datta, Aparesh & Sanyal, Dipankar & Agrawal, Amit & Das, Ajoy. (2019). A review of liquid flow and heat transfer in microchannels with emphasis to electronic cooling. Sādhanā. 44. 234. 10.1007/s12046-019-1201-2.

23. Kaijie Lu, Chunju Wang, Changrui Wang, Xueliang Fan Topological structures for microchannel heat sink applications - a review Published by EDP Sciences 2023
24. Li, Yantao, et al. "Investigation of flow and heat transfer performance of double-layer pin-fin manifold microchannel heat sinks." *Water* 14.19 (2022): 3140.
25. Ndao, Sidy, Yoav Peles, and Michael K. Jensen. "Multi-objective thermal design optimization and comparative analysis of electronics cooling technologies." *International Journal of Heat and Mass Transfer* 52.19-20 (2009): 4317-4326.
26. Koşar, Ali, and Yoav Peles. "Thermal-hydraulic performance of MEMS-based pin fin heat sink." (2006): 121-131.
27. Li, Z., & Peterson, G. P. (2006). Temperature and pressure characteristics in microchannel heat sinks. Microelectronics Journal, 37(10), 1081–1090
28. Hooman, Kamel, Faegheh Hooman, and Saeed R. Mohebpour. "Entropy generation for forced convection in a porous channel with isoflux or isothermal walls." *International Journal of Exergy* 5.1 (2008): 78-96.
29. Hetsroni, G., et al. "Heat transfer in micro-channels: Comparison of experiments with theory and numerical results." *International Journal of Heat and Mass Transfer* 48.25-26 (2005): 5580-5601.
30. Hung, Tu-Chieh, et al. "Heat transfer enhancement in microchannel heat sinks using nanofluids." *International Journal of Heat and Mass Transfer* 55.9-10 (2012): 2559-2570.
31. Chamoli, Sunil, et al. "Numerical optimization of design parameters for a modified double-layer microchannel heat sink." *International Journal of Heat and Mass Transfer* 138 (2019): 373-389.
32. Sathiyan, S., et al. (2020). "CFD analysis of nanofluid flow in microchannel heat sinks: a review." International Communications in Heat and Mass Transfer, 110, 104389
33. Kumar, Rajesh, Mohammad Zunaid, and Radhey Shyam Mishra. "Multi-objective optimization of hydrothermal performance of a porous minichannel

heat sink using RSM and NSGA-II algorithm." *International Journal of Heat and Fluid Flow* 110 (2024): 109600

34. Bhandari, Prabhakar, and Yogesh K. Prajapati. "Thermal performance of open microchannel heat sink with variable pin fin height." *International journal of thermal sciences* 159 (2021): 106609

35. Adham, Ahmed Mohammed, Normah Mohd-Ghazali, and Robiah Ahmad. "Thermal and hydrodynamic analysis of microchannel heat sinks: A review." *Renewable and Sustainable Energy Reviews* 21 (2013): 614-622

36. Rajalingam, A., and Shubhankar Chakraborty. "Effect of shape and arrangement of micro-structures in a microchannel heat sink on the thermo-hydraulic performance." *Applied Thermal Engineering* 190 (2021): 116755.

LIST OF PUBLICATIONS

[1] Shansi Rawat and Mohammad Zunaid, " Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number and Nanoparticle Concentration" submitted in The Power and Intelligent Control Systems (PICS-2025), Scopus Indexed, 2025, NIT Hamirpur, (H.P.).

(Accepted)

[2] Shansi Rawat and Mohammad Zunaid, " Numerical Analysis of 3D Microchannel Heat Sink Under Varying Heat Flux, Reynolds Number" submitted in The Power and Intelligent Control Systems (PICS-2025), Scopus Indexed, 2025, NIT Hamirpur, (H.P.).

(Under R

e-Receipt for State Bank Collect Payment

NIT HAMIRPUR

NIT INSTITUTE HAMIRPUR , , HAMIRPUR-177005
Date: 29-May-2025

SBCollect Reference Number :	DUO1296953
Category :	WORKSHOP STC FDP CONFRENCE
Amount :	₹3000
NAME OF THE PARTICIPANT :	Shansi Rawat
DESIGNATION :	PG Student
MOBILE NUMBER :	8936978547
e-mail :	shansirawat@gmail.com
ADDRESS :	NIT Hamirpur
NAME OF INSTITUTE/ORGANIZATION :	NATIONAL INSTITUTE OF HAMIRPUR
TITLE OF STC/FDP/WORKSHOP/CONF :	PICS-2025 Conference
DATES OF PROPOSED PROGRAMMES :	July 4-5 2025
ORGANIZING DEPARTMENT :	Electrical Engineering
AMOUNT OF FEE PAID :	3000
REMARKS :	Conference Paper
COURSE FEE :	3000

15% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

Filtered from the Report

- Bibliography
- Quoted Text
- Cited Text
- Small Matches (less than 8 words)

Match Groups

157 Not Cited or Quoted 15%
Matches with neither in-text citation nor quotation marks

0 Missing Quotations 0%
Matches that are still very similar to source material

0 Missing Citation 0%
Matches that have quotation marks, but no in-text citation

0 Cited and Quoted 0%
Matches with in-text citation present, but no quotation marks

Top Sources

11% Internet sources
8% Publications
11% Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

No suspicious text manipulations found.

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A Flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Main Bawana Road, Delhi-42

PLAGIARISM VERIFICATION

Title of the Thesis____

____

____________ Total Pages ________________ Name of the Scholar____________________________________ Supervisor (s)

(1)_______________________________________

(2)_______________________________________

(3)_______________________________________

Department____

This is to report that the above thesis was scanned for similarity detection. Process and outcome is given below:

Software used: _______________________ Similarity Index: ______, Total Word Count: ________

Date: ___________

Candidate's Signature **Signature of Supervisor(s)**