New Journal and we have not received input yet 29 (2022) 101203

Contents lists available at ScienceDirect

Thermal Science and Engineering Progress

journal homepage: www.sciencedirect.com/journal/thermal-science-and-engineering-progress

Fluid flow and heat transfer in microchannel heat sinks: Modelling review and recent progress

Jie Gao [a], Zhuohuan Hu [a,*], Qiguo Yang [a], Xing Liang [b], Hongwei Wu [a,c,*]

[a] Institute of Advanced Power Equipment and Technology, School of Energy and Power Engineering, Uni-versity of Shanghai for Science and Technology, Shanghai 200093, China
[b] School of Computing and Mathematical Sciences, University of Greenwich, SE10 9LS, UK
[c] School of Physics, Engineering and Computer Science, University of Hertfordshire, Hatfield AL10 9AB, UK

ARTICLE INFO

Keywords:
Boiling
Heat transfer
Helical coils
Neural network
Terrestrial gravity
Computational fluid dynamics
Molecular dynamics simulation
lattice Boltzmann methods
Genetic algorithm
Taguchi algorithm

ABSTRACT

Nowadays, microchannels have been widely utilized in various multidisciplinary fields, and as a consequence, some new and different requirements for microchannels in the process of practical application are required, such as structure, working fluid, and operating conditions, etc. This article reviews the current research achievement of microchannels, as well as the thermodynamic research on microchannels with different structures in the past five years, but mainly focuses on the numerical methods. The purpose of this review article aims to summarize a comprehensive overview of the latest developments of numerical methods in microchannel heat sinks, as well as to provide a useful benchmark for future research. The present article reviews straightforward on the most commonly used numerical methods for solving governing equations and optimizing data, including conventional computational fluid dynamics (CFD) simulation methods, molecular dynamics simulation (MDS), Lattice Boltzmann methods (LBM), direct simulation Monte Carlo (DSMC), and other techniques such as machine learning (ML) approach, artificial neural network (ANN) method, genetic algorithm (GA), Taguchi algorithm (TA), as well as optimisation methods. This review will not only help to understand the physical mechanism of microchannels in different application fields but also help to fill in the gaps in related research and provide research methods for future numerical studies.

1. Introduction

Microchannels are not new in concept but have attracted considerable attention over the past four decades. With the rapid development of microelectronic devices, the heat generation within the system has been significantly increased exponentially. The high heat flux generation (in the order of 1000 W/cm^2) with reduced surface area becomes one of the major concerns to dissipate the heat to maintain the reliability and performance of electronic devices. It is therefore imperative to propose a more compact and efficient device to tackle the thermal management challenges in electronic components. Microchannel heat sinks (MCHS), as an innovative cooling technology for electronic systems, was firstly proposed by Tuckerman and Pease [1], have attracted growing attention and have been widely used in industrial applications due to their inherent advantages of superior heat transfer performance, smaller geometric size and volume per heat load, lower coolant requirement and lower operational cost.

It is recognized that MCHS with other technologies can deal with complex thermal challenges faced by the industry today such as 5G devices, micro-electromechanical system (MEMS) devices, fuel cells, air-conditioning systems, as well as medical/biological and energy sectors. For example, the use of MCHS in a micro-chip, due to the high integration of microchip, a large amount of heat generated needs to be taken away in time. In addition, due to its small size and high density, microchannel radiator has successfully played an important role in cooling [2]. It was noted that microchannels can be used in the production of biodiesel [3]. Meyari et al. [4] numerically investigated the blood flow process in blood vessels by studying the flow effect in microchannels. The contribution of the microchannel is ineffable, however, with the great progress of science and technology, the traditional microchannel heat sink has been unable to meet the demands of

---

* Corresponding authors at: Institute of Advanced Power Equipment and Technology, School of Energy and Power Engineering, University of Shanghai for Science and Technology, Shanghai 200093, China (Z. Hu). School of Physics, Engineering and Computer Science, University of Hertfordshire, Hatfield AL10 9AB, UK (H. Wu).

E-mail addresses: huzh@usst.edu.cn (Z. Hu), h.wu6@herts.ac.uk (H. Wu).

https://doi.org/10.1016/j.tsep.2022.101203
Received 27 November 2021; Received in revised form 10 January 2022; Accepted 10 January 2022

Nomenclature

Cir	circular
D_h	hydraulic diameters
d	actual data
Hax	hexagonal
Nu	Nusselt number
n	number of repeated experiments
p	prediction parameter
Rec	rectangular
Re	Reynolds number
Tra	trapezoidal
Y	index value

Subscripts

i	output parameter
j	represents a given point in the data set
lb	larger-the-better–type
sb	smaller-the-better–type

Abbreviations

AC	alternating current
ANN	artificial neural networks
ANFIS	adaptive neuro-fuzzy inference systems
BBC	bounce back
BFS	backward-facing step
BP	back propagation
BSR	bounce back and specular reflection
CFD	computational fluid dynamics
CLSVOF	coupled level-set and volume of fraction
DA-HCPV	dense-array high concentrator photovoltaic
DEM	discrete element method
DSMC	direct simulation Monte Carlo
FREI	flame repetitive extinction ignition
FDM	finite difference method
FEM	finite element method
FVM	finite volume method
GA	genetic algorithms
GMDH	group of method data handling
GP	genetic programming
IBM	immersion boundary method
LBM	Lattice Boltzmann method
LMB	Levenberg-Marquardt Backwardpropagation
LVG	longitudinal vortex generator
MCHS	microchannel heat sinks
MCHX	heat exchanger with microchannel coil
MCSHP	microchannel separate heat pipe
MDS	molecular dynamic simulation
MEMS	micro-electromechanical system
MJMC	multi-jet microchannel
MOGA	multi-objective genetic algorithm
Opt-LHD	optimal Latin hypercube design
PEMFC	proton exchange membrane fuel cell
RSM	response surface methodology
SNR	signal-to-noise ratio
SPM	single-phase model
SVM	support vector machines
TEC	thermoelectric cooler
TM	taguchi method
TOPSIS	technique for order preference by similarity ideal solution
VOF	volume of fluid method
WMCCT	wavy microchannels with consistent crests and troughs
WMOCT	wavy microchannels with opposite crests and troughs
XGBoost	extreme gradient boosting
4QMCHS	four quadrants microchannel heat sink

practical applications such as more and more sophisticated instrument cooling, more subtle biological research. Thus, it is important to optimise the design of the microchannel heat sink [5], such as changing the length of the long side and the width of the secondary channel at the entrance of the microchannel. Ling et al. [3] proposed a microchannel structure with a staggered water cooling channel and water heating channel. And various methods, such as adding corrugated fins on cylindrical microchannels [2], changing the wavelength and amplitude of corrugated microchannels [6], and inlaying coils in rectangular microchannels [7] to enhance their heat transfer by changing the microchannel structure, as well as to change the heat transfer characteristics of traditional microchannels were investigated. Al-Rashed et al. [7] numerically investigated the hydrothermal and irreversible behaviour of a biologically synthesized water-silver nanofluid in a wavy MCHS. It was found that the nanofluid showed a better cooling performance in comparison with that of pure water. Arjmandfard et al. [8] performed a molecular dynamic (MD) approach to study the time evolution of nanofluid flow in a microchannel with various sizes of Fe nanoparticles. It was stated that by adding Fe nanoparticles to base-fluid the highest rate of velocity and temperature of base fluid could increase 12% and 37%, respectively.

This paper does not and cannot review all the interesting and important progress related to numerical methods in microchannel, but tries to collect, summarize and discuss the research cases of the microchannel in different fields in the past five years. From the analysis of the research process, this focus not only shows the effectiveness of the methods used by researchers but also helps readers in the choice of research methods in the future. The organization of this paper is as follows: the second section mainly introduces the numerical research methods of the microchannel, including CFD method, lattice Boltzmann method (LBM), molecular dynamics simulation (MDS), direct simulation Monte Carlo (DSMC), Taguchi algorithm (TA), genetic algorithm (GA), and artificial neural network (ANN) method. In the last section, the conclusion and prospects of this paper will be given.

2. Numerical methods

2.1. Classical computational fluid dynamics (CFD)

Over the past sixty years, scientific computation has emerged as the most versatile tool to complement theory and experiments. Modern numerical methods, in particular those for solving nonlinear partial differential equations (PDEs), are at the heart of many advanced scientific computations. The interplay between computation, theory, and experiments was envisioned by John von Neumann in 1949. Numerical solutions for solving nonlinear PDEs were first put into use by von Neumann himself, in the mid-1940 s.

In this section, the analysis of flow and heat transfer in microchannels with state-of-art conventional CFD methodology in recent years will be reviewed systematically. CFD technology was presented in the 1960 s, with the continuous improvement of computer techniques, CFD simulation has developed rapidly. In general, the CFD method is a mathematical technique to solve governing equations (normally partial differential equations) describing fluid flow by computer. Based on the analysis and calculation of various problems of fluid mechanics, the approximate solution of the actual fluid model equation is obtained. Among them, the governing equations are known as the mass conservation equation, the energy conservation equation, and the momentum

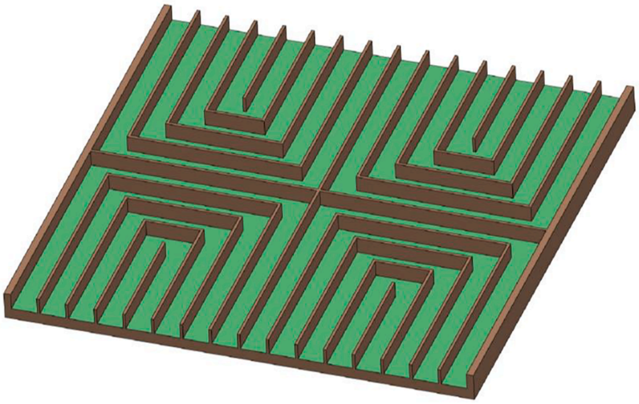


Fig. 1. Four quadrants microchannel heat sink.

equation. Due to the complex and diverse fluid flow state in practical problems, the corresponding solutions have become more and more challenging. It is noted that three typical numerical methodologies, i.e., Finite Difference Method (FDM), Finite Element Method (FEM), and Finite Volume Method (FVM) are normally used to solve the Navier-Stokes equation. Among these, FDM is the first method used by CFD to solve the governing equations, and it is still widely used. FDM divides the calculation area into discrete grids and uses nodes to replace the continuous grids. The original equation is approximately discretized by the continuous variables of the calculation area, and then gradually approximates the solution. In the study of microchannels, FEM could be used to solve various problems described by the Poisson equation and Laplace equation. This method divides the calculation area into finite continuous elements, establishes a correlation function in each element, and then combines the approximate solutions of all element functions to obtain the solution of the calculation area. FVM is now a relatively mature CFD algorithm. This method divides the calculation area into non-overlapping control volumes and replaces them with nodes. The equations to be solved are integrated with each control volume to obtain discrete equations. These computational methods in the field of CFD not only enrich their functions but also gradually expand the applicable scope of CFD. Below we list some examples of microchannel problems where researchers use CFD methods to solve different practical problems.

In the previous studies, the complex structure of microchannels could not be processed and this made the experimental study more difficult. Therefore, the numerical research method could become an effective tool in dealing with complex problems. Feng et al. [6] applied CFD to simulate the laminar flow and heat transfer characteristics of a rectangular microchannel with nested coils, and the governing equations were solved by the FVM. Their results showed that the coil in the microchannel could enhance the fluid disturbance and improve the temperature distribution. The maximum deviation between experimental data and numerical results is 14.2%. Liu et al. [9] employed CFD to study a kind of annular inclined microchannel with multiple staggered entrances and exits. Their simulation results showed that the temperature distribution of staggered arrangement was more uniform than that of sequential arrangement. Yang and Cao [5] proposed a new type of hybrid microchannel heat sink by varying the inlet length, the width of the secondary channel, and the Reynolds number (Re) of the microchannel heat sink. They stated that compared with the traditional MCHS, the new MCHS can effectively reduce the pressure loss due to its flow characteristics. Lu and Zhai [10] applied CFD to investigate the heat transfer and flow characteristics of MCHS combined with dimples and vortex generators. Their results showed that the combination of dimples and vortex generators could improve the heat transfer performance and reduce the pressure loss. Ling et al. [3] performed a combined CFD simulation and experimental study to investigate the heat transfer and flow characteristics of new staggered microchannels and employed a conjugate heat transfer numerical simulation to optimise the structure. The analysis of experimental and simulated results showed that the Nusselt number of the staggered microchannel was 65.4% higher than that of parallel microchannel, and the surface temperature distribution was more uniform. In their study, the deviation between experimental data and simulation results was less than 18%. Ali et al. [11] proposed a four quadrants microchannel heat sink (4QMCHS) and established a three-dimensional conjugate heat transfer model to investigate its temperature distribution. Their results showed that the inlet and outlet directions had a great effect on the temperature nonuniformity and temperature distribution of the heat sink with countercurrent mode was relatively uniform. The specific structure size is shown in Fig. 1. Totally there are 32 channels at the top with the microchannel radiator at the bottom.

Peng et al. [12] employed a three dimensional CFD to simulate the flow and heat transfer process in the multijet microchannel (MJMC) heat sink with coolant flowing through alternative inlet and outlet jets in the direction normal to the heated surface. Compared with traditional microchannels, it was found that the MJMC combined the advantages of impinging jet flow and entrance effects of microchannel. Lin et al. [13] proposed a new MCHS with variable wavelength and variable amplitude along the flow direction. They found that this change can effectively mix coolant and enhance heat transfer. Hasis et al. [14] performed a CFD study to simulate the laminar flow and heat transfer in twisted sinusoidal microchannels. It was verified that the heat transfer performance of the twisted wavy channel was better than that of the sinusoidal wavy channel. Lei and Chen [15] numerically investigated the heat transfer and pressure drop characteristics of supercritical carbon dioxide in horizontal wavy microchannels with consistent crests and troughs (WMCCT), as well as wavy microchannels with opposite crests and troughs (WMOCT). Their results indicated that the heat transfer coefficient and pressure drop in WMCCT and WMOCT could be increased with the increase of the amplitude and decreased with the decrease of the wavelength. Sreehari and Sharma [16] performed a combined CFD method with experimental work to analyze the overall performance of three different rectangular cross-section serpentine microchannels under different Reynolds numbers and heat fluxes. They reported that the U-serpentine microchannel exhibited the best thermal performance while compared to the other two serpentine microchannels. The experimental data were compared with the simulation results, it was noted that the pressure drop deviation is 10–11%, and the average base temperature deviation was within 1–3%. The shape of the microchannel in the study is illustrated in Fig. 2.

Gomez-Pastora et al. [17] numerically analyzed the hydrodynamics and mass transfer characteristics of solute in Y-Y-shaped microchannels.

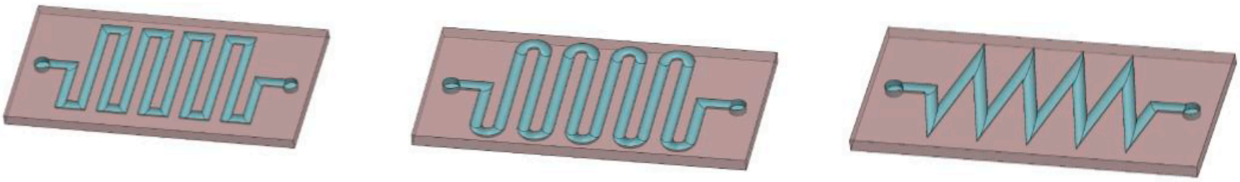


Fig. 2. Channel shape of microchannel [29].

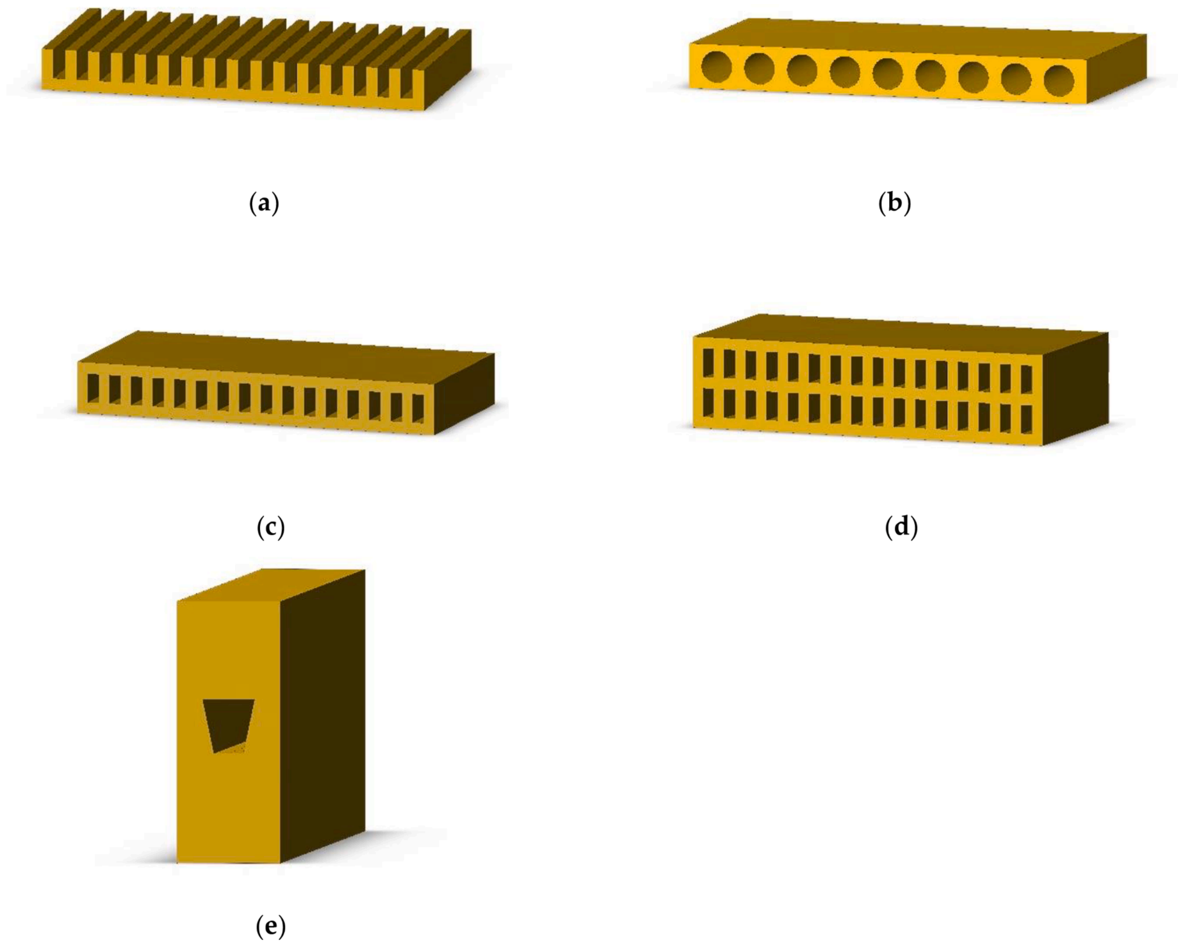


Fig. 3. Classic microchannel cross-section and shape: (a) Rectangular section without cover; (b) Circular cross section; (c) Rectangular covered section; (d) Double-layer rectangular section; (e) Trapezoidal section.

The comparison between experimental data and numerical results proved that the ability of CFD to effectively simulate the concentration gradient of microchannels over a wide range of flow rates. Bayrak et al. [18] established a two-dimensional modle and numerically investigated the effect of surface modification on the heat transfer performance of different MCHS in cooling channels of lithiumion batteries. Their research results showed that the combination of cavity and rib in microchannel has good heat transfer performance. Zheng et al. [19] carried out a combined experimental work and numerical simulations to analyze the mechanism of variable cross-section microchannels with internal components. Their results showed that the rapid change of channel size could lead to the pressure drop of the fluid impacting the wall and the gas phase accelerates in the throat, thus improving the mixing efficiency. Qaderi et al. [20] established a two-dimensional microchannel model of triangular obstacles under the condition of heterogeneous zeta-potential. Their results presented that higher hurdles could improve the mixing efficiency and reduce the mass flow rate in microchannels. Hosseinpour et al. [21] combined the CFD method and Response Surface Methodology (RSM) to analyze the flow and heat transfer characteristics of rectangular microchannels with four different types of fins. Their results showed that the overall performance of the pyramid fin was the best. Derakhshanpour et al. [22] numerically investigated the effects of different ribs (including semi-circular ribs, semi-elliptic ribs, semi-circular ribs, and filleted corner, semi-elliptic ribs, and filleted corner) in the microchannel on the heat transfer and flow characteristics of the MCHS. Their results indicated that the heat transfer could be enhanced by changing the ribs' corner curvature, and the overall performance of the filleted ribs was the best. He et al. [23] applied FVM to numerically investigate the laminar flow characteristics of the mixed nanofluid in the double-layer sinusoidal microchannel. The analyses of the heat transfer performance of the microchannel showed that changing the wavelength, using nanofluids, and increasing the flow rate could improve the thermal performance of the microchannel. In addition, the sinusoidal wavelength of the microchannel could enhance the mixing of the surface and the fluid. From the above research works, the investigation on flow and heat transfer in microchannels has been carried out from different aspects, among which the most typical is the change of channel cross-section, channel number, and channel structure. Fig. 3 showed several typical cross-sections of microchannels. It can be seen from the current literatures that there are still some difficulties in the numerical simulation, such as grid division, turbulence model selection, boundary layer transition, etc. This will make the CFD simulation more difficult and a big deviation is observed [3,6,16,26]. This could be improved when taking several numerical techniques into consideration, such as (1) reasonable meshing and selecting the number of grids to get more accurate numerical results under the minimum computational cost. (2) selection of accurate turbulence model is also important for solving the governing equations. However, there is no universal turbulence model at present, and there are still challenges in dealing with some complex flow problems. (3) accurately judge the boundary layer transition information, reduce the calculation error.

Due to the diversity of solving governing equations by CFD method, it has a strong predictive ability in dealing with single-phase, two-phase flow and flow boiling problems. Yue et al. [24] employed CFD to study the flow and heat transfer characteristics of microchannel heat pipe evaporators with different filling ratios. Their results found that the cooling capacity of the evaporator increased with the increase of the charging ratio. Mandel et al. [25] developed a simplified “2.5D” CFD model for studying the single-phase flow and heat transfer in manifold microchannels. They found that the model had high accuracy when inertia was low enough. Ding et al. [26] combined the CFD method with the experiment to establish a three-dimensional numerical model of R410a convective and condensed annular flow in a rectangular microchannel. Their models could predict the condensation film thickness, liquid flow rate, and heat transfer coefficient with an average error of 5.3%. When compared with the experimental data, most of the error is less than 15%. Burk et al. [27] numerically investigated the transfer of conjugate gradient in a two-phase microchannel array. Their research proposed a method that combined heat transfer correlation with Finite Element Analysis (FEA) to evaluate its effectiveness in microchannels with high heat flux. El-Genk et al. [28] applied CFD to study the water and air convection heat transfer in the process of laminar flow in a microchannel. In their study, the correlation of Nusselt numbers with different parameters in microchannels was obtained. Jiang and Zhou [29] combined the VOF method with dynamic contact angle to analyze the flow characteristics of water in a single straight microchannel. Their results showed that the flow patterns in different water injection channels are also different. Abdollahi et al. [30] combined the CFD method with the experiment to establish a three-dimensional numerical model to analyze the flow and heat transfer of liquid–liquid Taylor flow in a square microchannel. Their results showed that liquid–liquid Taylor flow could increase the heat transfer rate by 700% compared with single-phase flow, and the three-dimensional simulation method of Taylor flow could be extended. Soleimani et al. [31] used VOF to establish a three-dimensional numerical model to study the highly subcooled flow boiling process of HFE-7100 with different concentrations of alumina nanoparticles in the microchannel. It was found that the flow boiling heat transfer was enhanced by the thermal boundary layer disturbance caused by the wall gas-phase motion in a two-phase flow. Chatterjee et al. [32] numerically investigated the straight and spiral microchannels and employed CFD to study the Taylor bubble formation process in the microchannel. Their results indicated that the centrifugal force in the spiral microchannel has a large effect on the Taylor bubble dynamics, and under the same flow state, the vortex intensity generated by the two-phase flow in the spiral channel was higher than that of the single-phase flow. Kumar et al. [33] established a three-dimensional numerical model to study the flow and heat transfer characteristics of trapezoidal microchannels for Reynolds numbers ranging from 96 to 720. According to the numerical simulation, they found that the heat transfer efficiency of the trapezoidal microchannel was 12% higher than that of the rectangular microchannel.

Trofa et al. [34] combined CFD with the Discrete Element Method (DEM) to simulate the adhesion process of particles on the wall of the microchannel. Their results found that the model could effectively capture several characteristics of the scaling process. Mohammadpour et al. [35] combined the single-phase model (SPM) with Eulerian-Lagrangian (DPM) model to analyze the flow and heat transfer characteristics of a new type of microchannel and nanofluid synthetic jet device. Their output indicated that SPM over-predicted the heat transfer enhancement, while DPM realistically took into account the forces on the particles and the base fluid. Zeng et al. [36] proposed a topology optimisation method for designing a planar water-cooled heat sink. Their results showed that the pumping power of the microchannel could be saved by more than 50% under the same cooling requirements. Comparing the simulation results with experimental data, it was found that the errors of pressure and temperature were less than 6% and 10 °C, respectively. Yang et al. [37] combined with the optimal Latin hypercube design (Opt LHD), Pareto chart analysis, RSM, non-dominated

Table 1
Summary of microchannel research data.

Author(s)	Channel shape	Channel material	D_h (mm)	Re	Fluid(s)	Dimension
Feng et al [6].	Rec single, straight	Copper	0.67	188–1458	Water	3
Lin et al [13].	Rec multi, wavy	Silicon	0.12	300,400,500,600,800	Water	3
Kewalramani et al [38].	Rec multi, straight	Silicon	0.3,0.48	2000	Water	3
Soleimanikutanaei et al [50].	Rec multi, straight	Silicon	0.4	200,600,1000	Water	3
Alfaryjat et al [51].	Hex single, straight	Aluminum	0.17	100–1000	CuO, Al_2O_3,ZnO, SiO_2	3
Yue et al [24].	Rec multi, straight	Copper	1.09	–	R22	2
Mandel et al [25].	Rec multi, straight	–	0.18	–	Air and water	2.5
Chai et al [52].	Rec single, straight	Silicon	0.13	443	Water	3
Bayrak et al [18].	Rec single, straight	–	0.67	600	Water	3
Kumar [33].	Tra single, straight;Rec single, straight	Silicon	0.20, 0.348	96–720	Water	3
Burk et al [27].	Rec multi, straight	Silicon	0.073	–	Water	3
Wang et al [53].	Rec multi, straight	Silicon	0.47	100–1000	Water	3
Liu et al [9].	Rec, Cir, multi, straight	Copper	2.3, 2.1	576–2305	Water	3
Yang and Cao [5].	Rec multi, straight	Silicon	0.2	295	Water	3
Lu and Zhai [10].	Rec multi, straight	Silicon	0.27	167–834	Water	3
Vasilev et al [54].	Rec multi, straight	Aluminum	1	500	Water	3
Shen et al [55].	Rec multi, straight	Silicon	119.15	345.18	Water	3
Lei and Chen [15].	Rec multi, wavy	–	1.31	–	CO_2	2
Sreehari and Sharma [16].	Rec single, serpentine	Silicon	0.4	100–400	Water	3
Jiang and Zhou [29].	Rec single, straight	–	0.1	29.9, 66.6, 112.7	Air and water	3
Liu et al [56].	Rec single, straight	Aluminum	0.45, 0.4, 0.3	3.79–67.41	R11 and water	3
Ong [46].	Rec multi, straight	PDMS, PTFE, PDMS/ MWCNT	0.3, 0.38, 0.42	–	Water	3
Ling et al [3].	Rec multi, straight	–	0.75	< 2300	Water	3
Abdollahi [30].	Rec single, straight	Stainless steel 304	1, 2	6–100	Kerosene, hexadecane and water	3
Su et al [57].	Rec single, straight	–	0.4	25–2000	Air	3
Khalesi et al [58].	Rec multi, straight	Stainless Steel	0.53	100	Supercritical CO_2	3
He et al [23].	Rec single, wavy	Silicon	0.1	50,700,1200	Al_2O_3-Cu/water	3
Al-Baghdadi et al [59].	Rec single, straight	Silicon	0.13	900	Water, SiO_2,Al_2O_3, CuO	3
Ansari and Zhou [60]	Rec multi, straight	–	0.49	50–250	Water and air	3
Ali Soleimani [31].	Rec multi, straight	Copper	0.18	530–2000	HFE-7100	3
Derakhshanpour [22].	Rec single, straight	Silicon	0.13	66–396	Water	3

Cir: circular, Rec: rectangular, Hex: hexagonal, Tra: trapezoidal.

sorting genetic algorithm II (NSGA- II), and technique for order preference by similarity ideal solution (TOPSIS), the heat transfer performance of the hybrid microchannel was optimised. The CFD simulation results showed that the proposed optimisation method could also be used for performance optimisation of other types of microchannels.

Kewalramani et al. [38] numerically investigated the thermal–mechanical characteristics of the microchannel heat sink and optimised the porous medium model. Their simulation results of the velocity and temperature distribution consisted with previous research work. Wang et al. [39] conducted a numerical study to establish the film condensation heat transfer model of annular flow in elliptical microchannels. Their simulation results were validated against the experimental data in open published literature. Taher et al. [40] established an analytical model for capillary flow across the back steps in microchannels with and without top surface. Their outputs established a capillary pressure analysis model and concluded the relative law between capillary pressure and gas–liquid interface. Cao et al. [41] performed a combined CFD method and experimental work to analyze the high and low-pressure countercurrent heat exchanger with isotropically etched support pillars. Their results obtained the correlation between Nusselt number (Nu) and Darcy friction coefficient in a staggered channel and the side-by-side channel. Bucci et al. [42] conducted a combined CFD method with experimental study to analyze the flame dynamics in the microchannel. Their results presented that the numerical simulation could effectively plot the phase diagram of the flame position in microchannels. Wei et al. [43] numerically investigated the different heat flux and filling rates in the axial microchannel aluminum ammonia groove heat pipe (GHP). Combining the experiment with numerical simulation, they concluded that the Ω-shape GHP could transfer heat in a long distance with a small temperature difference. Laziz et al. [44] performed a combined CFD method with experiment to explore the effect of KOH catalyst on biodiesel production in T-type microchannel. Their outputs revealed that the residence time of KOH did not significantly increase the conversion rate of diesel after 20 s, but their numerical simulation showed that the annular recirculation structure in the slug could enhance the mixing effect.

At present, still there is a lack of research on the use of new materials to make microchannels. It is recognized that numerical simulation could effectively solve this problem and obtain the availability of materials to a certain extent. Vajdi et al. [45] numerically studied the heat transfer characteristics of the microchannel made of ZrB_2, and the FEM was used to solve the governing equations. Their research results expanded the application of ultra-high temperature ceramics in the field of microchannel fabrication. Ong et al. [46] employed CFD method to study the effect of the heat transfer coefficients of different polymer materials (PDMS, PTFE, PDMS/MWCNT) and metallic aluminum on the thermal performance of microchannels under different working conditions. Their results showed that polymers could transfer heat by enhancing thermal conductivity. Fattahi et al. [47] numerically studied the thermal properties of Aluminum nitride (AlN) microchannel and compared with the one made by Al_2O_3. Their results showed that the thermal conductivity of the aluminum nitride microchannel was higher than that of aluminum oxide, and the heat transfer efficiency could be increased by 26%. Recently, Zhan et al. [48] performed a combined numerical study and theoretical analysis to analyze the hydraulic resistance, mixing efficiency and comprehensive performance of a T-shaped rectangular microchannel. Their results indicated that the hydraulic resistance was independent of the inlet conditions, and also found that the hydraulic resistance increased initially and then decreased with the increase of the width and height of the inlet and outlet, which reflected the extension of the classical Murray's law in the microchannel mixing. Wan et al. [49] reviewed the existing new microchannel design, and pointed out the challenges of improving the structure and working fluid to enhance heat transfer in microchannel research. Their results provided an efficient microchannel design method, which provided a reference for future research.

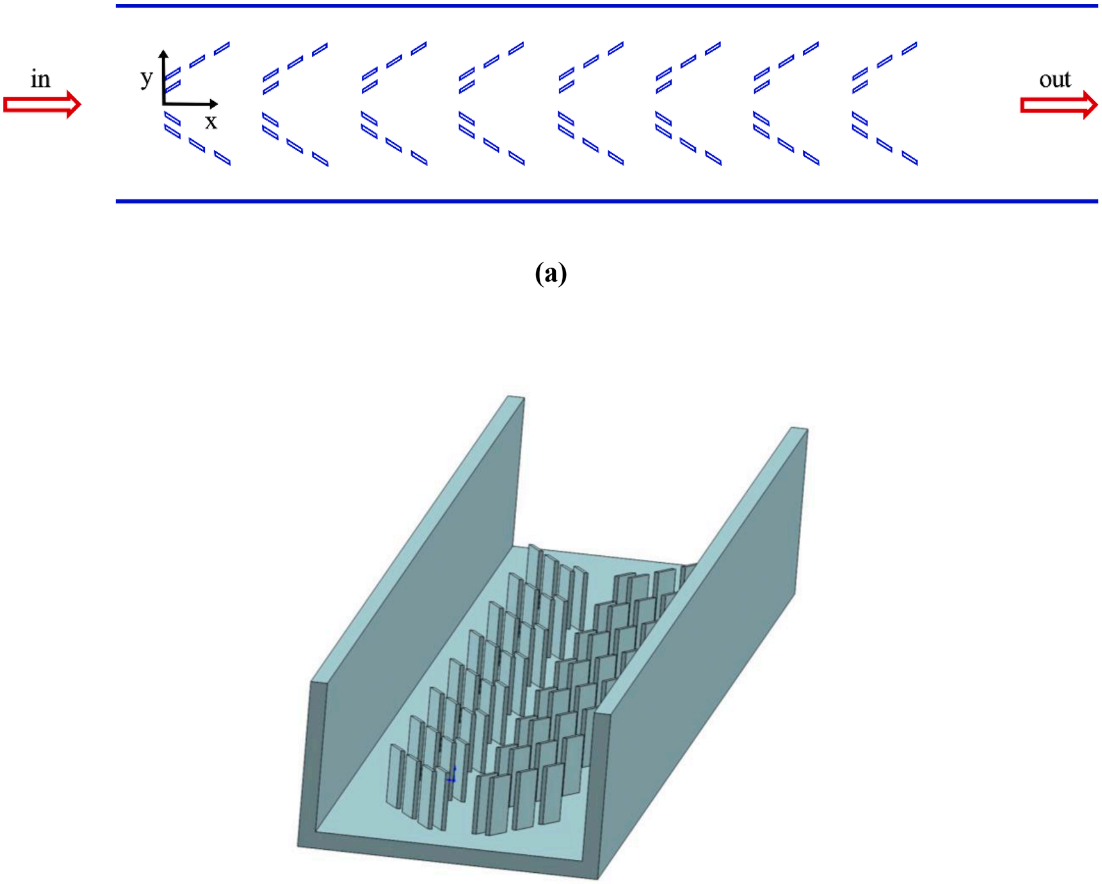
Table 1 summarizes the typical work based on the CFD method. In summary, the CFD could be regarded as a reliable and excellent simulation methodology to simulate a variety of complex conditions. With CFD, it can reduce the cost of the practical experiments and visually displays the required control parameters, which can greatly reduce the research cycle.

2.2. Lattice Boltzmann method (LBM)

LBM is a flow field simulation method developed in the 1980 s. Compared with the traditional CFD method, LBM has its unique advantages. Over the pas decades, LBM method can deal with some complex and irregular structures in practical problems and has obvious advantages in dealing with multiphase and multicomponent fluid problems. Unlike the traditional CFD method, the LBM method adopts the Bhatnagar-Gross-Krook (BGK) model, which can quickly solve the linear equations [74], effectively deal with complex boundary conditions, adopt parallel algorithm, and can be easily applied to multiphase flow [68]. Among all the particle algorithms, the calculation cost of the LBM method is relatively low.

Fallah and Rahni [61] applied the LBM based on the pseudo-potential method to analyze the effect of several control parameters such as capillary number, flow rate ratio, width ratio, and contact angle on droplet formation in conventional and improved T-junction microchannels. Their results found that the improved T-junction microchannel could produce smaller droplets and spacing. Fu et al. [62] performed a numerical simulation to establish a ternary LBM based on the color-gradient model to analyze the formation process of Janus droplets in Y-junction microchannels. Their outputs revealed that the size of the Janus droplet was related to the number of capillaries and the droplet size followed the scaling law. Ghadirzadeh and Kalteh. [63] employed LBM to investigate the laminar forced convection heat transfer characteristics of nanofluids in a slip annular microchannel. Their results indicated that the Nusselt number could be increased by changing the slip coefficient, particle diameter, radius ratio, and volume fraction. Kamali et al. [64] established a numerical model of coupled LBM for studying the effect of roughness in a two-dimensional flat microchannel, and the proposed model used the Poisson equation and the Nernst-Planck equation to solve the electric potential and ion concentration. Their results presented the effect of different roughness heights and roughness spacings on the flow velocity in the channel. Cai et al. [65] conducted the multiphase LBM to simulate the liquid film on the wall in the microchannel hydraulic cavitation flow. Their results proved the effectiveness of the multiphase Lattice Boltzmann model in solving the problem of hydraulic cavitation flow. Zhou et al. [66] combined the LBM, Immersion Boundary Method (IBM), and D3Q19 model to develop the motion model of magnetic particles under alternating gradient magnetic field. Their results found that the magnetic particles vibrated along the flow direction in an alternating gradient magnetic field and disturbed the flow field, which could increase the intensity of turbulence.

Afterwards, the research of [67] proposed a doubled-population LBM to analyze the heat transfer enhancement effect of the motion of magnetic nanoparticles within alternating gradient magnetic field. Their results showed that the magnetic nanoparticles were influenced by the alternating gradient magnetic field produced transverse velocity component. D' Orazioa and Karimipour [68] applied the LBM to establish a model of air mixing convective heat transfer in two-dimensional microchannels. Their results demonstrated that the LBM method can effectively simulate constant heat flux boundary conditions along the microchannel wall when there is slip velocity and buoyancy. Combined with the LBM method, Yang et al. [69] proposed an improved slip boundary condition to simulate several typical channel flows. It was found that this simulation method could be effectively applied to the Couette flow, the force driven Poiseuille flow, the time-dependent force driven Womersley flow, the porous plate flow, and the channel flow with



(a)

(b)

Fig. 4. Louvered microchannel structure diagram: (a) One-dimensional floor plan; (b) Three-dimensional structure diagram.

a surface-mounted block, and could avoid the leakage of wall mass and eliminate the error caused by an external force. Ahangar et al. [70] employed the LBM method to simulate the flow of rarefied gas in the microchannels with backward facing step, and used a model of two relaxation times when slipping and transient flow. Their results concluded that the LBM was consistent with the DSMC method, and the LBM could be better than the DSMC method. In addition, compared with the multi relaxation time model, the calculation time with the proposed LBM was greatly reduced. Saravani and Kalteh [71] applied the Lattice Poisson–Boltzmann method to analyze the Newtonian nanofluid in the microchannel under thermostat boundary conditions, and solved the governing equations by the D2Q9 model. Their results indicated that the velocity of nanoparticles in microchannels decreased with increasing volume fraction.

Wang et al. [72] used LBM to simulate the fluid flow in a new type of louvered microchannel heat sink (see Fig. 4) and they observed the characteristics of laminar forced convection heat transfer in the channel. Their results showed that the fluid flow in the new microchannel enhanced the guiding effect of the louver, and generated the vortex structure, thereby strengthening the heat transfer.

Similarly, Ahangar et al. [73] applied the lattice Boltzmann method with two Relaxation Times to analyze the flow state of rarefied gas in multi throats microchannels with slip and transition flow regimes. They reported that compared with a single relaxation time, the slip velocity predicted by this model has higher accuracy. It was also found that, compared with multi relaxation time, this model could reduce the cost. (The structure is shown in Fig. 5)

Afrouzi et al. [74] applied the incompressible version of LBM with precondition factor to analyze the flow and heat transfer characteristics of the fluid in microchannels with superhydrophobic surfaces. Their results showed that the friction coefficient and Nusselt number in this model were proportional to the volume fraction of nano-particles and the Hartman number. Wang et al. [75] employed a three-dimensional ternary color-gradient lattice Boltzmann model to analyze the formation process of Janus droplets in a Y-shaped co-flow microchannel. Their results showed five flow patterns during Janus droplet formation. He et al. [76] used the LBM and D3Q19 velocity model to investigate the motion laws and characteristics of magnetic particles in microchannels. Their results indicated that the model can effectively calculate the interaction between magnetic moments, they also found that the external magnetic field and flow field change the fluid flow by changing the structure of magnetic particles. Zhang et al. [77] applied the LBM to study the effects of boundary conditions, buoyancy, and sparsity on the heat transfer of asymmetric walls in a horizontal microchannel. They stated that the heat transfer intensity of the microchannel inlet region was strongly depends on those three factors.

Unlike CFD, the model characteristic scale studied by the lattice Boltzmann method can reach the order of magnitude of micro and nano. In the simulation process, the LBM method is not limited to the mesh quality, thus, it has a great advantage over CFD in simulating fluid and other complex boundary conditions. It can be seen from the above that most of the current LBM methods are focused on the study of fluid flow and heat transfer in the channel. Moreover, due to the advantages of the LBM method, it can simulate the working fluids in a variety of complex and irregular structure microchannels, such as the new shutter microchannel, multi throat microchannel. It should be also noted that LBM is suitable for fluid simulation under complex components and special driving forces. (Table 2 summarizes the application examples of the LBM method in specific experimental research.)

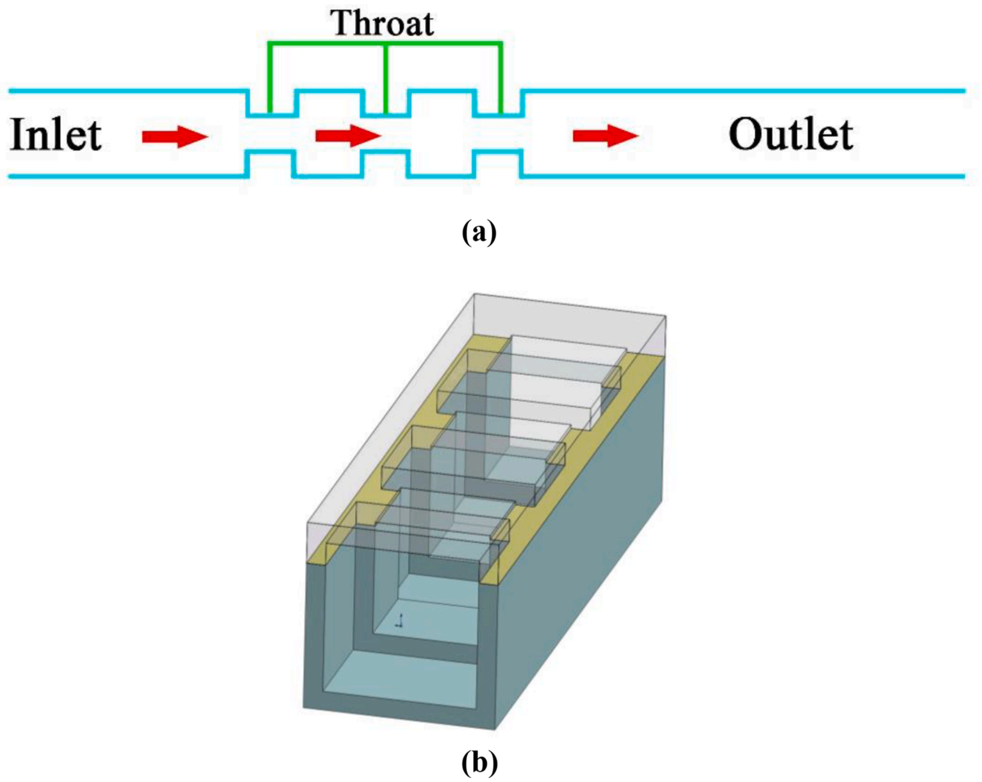


Fig. 5. Multi throat long microchannel structure diagram: (a) One-dimensional floor plan; (b) Three-dimensional structure diagram.

Table 2
Summary of the application of the LBM method on microchannels.

Author(s)	Channel shape	Latticemodel	Working fluid(s)	Dimension	Highlights
Fallah and Rahni [61].	T-junction	D2Q9	Water	2	Apply Pseudo-Potential LBM to simulate the droplets generated in the microchannel
Fu et al [62].	Y-junction	D2Q9	Janus droplet	2	Ternary color-gradient LBM Model to simulate Janus droplets
Ghadirzadeh and Kalteh [63].	Rec single, straight	D2Q9	Water-alumina nanofluid	2	Taking into account the wall temperature jump and slip speed
Kamali et al [64].	Rec single, straight	D2Q9	–	2	Considered the EDL layers fully overlap
Cai et al [65].	Rec multi, straight	D2Q9	Water	2	Simulated hydraulic cavitation flow using LBM
Zhou et al [66].	Rec multi, straight	D3Q19	Fe_3O_4	3	Combines the LBM method and the IBM method
D' Orazioa and Karimipour [68].	Rec single, straight	D2Q9	Air	2	LBM method can simulate the constant heat flux along the wall in the presence of slip velocity and buoyancy
Ahangar et al [70].	microchannel with a backward-facing step	D2Q9	Rarefied gas	2	Bounce back and Specular Reflection (BSR) boundary condition instead of the Bounce back (BBC) boundary condition.
Saravani and Kalteh [71].	Rec single, straight	D2Q9	Electrolyte fluid with Al_2O_3	2	–
Ahangar et al [73].	Rec variable cross-section	D2Q9	Rarefied gas	2	Proposed a Power-law approach
Afrouzi et al [74].	Rec single, straight	D2Q9	Al_2O_3-Water	2	To optimize low-energy, high-performance cooling tools
He et al [76].	Rec single, straight	D3Q19	Magnetic particles	3	Propose an implicit particle velocity method
Zhang et al [77].	Rec single, straight	D2Q9	Air	2	Mixed convection in a non-uniform wall heat flux channel is studied
Mehrizi et al [131].	Rec single, straight	D2Q9	Water/Ag nanofluid	2	Considered coupling heat transfer and viscous dissipation
Liou et al [132].	Louvered microchannel	D2Q9	Al_2O_3-Water	2	Combines nanfluids with louver structure
Yuan et al [133].	Rec single, straight	D3Q19	-	3	Migration characteristics of particles in square microchannels
Lalami et al [134].	Rec single, straight	D2Q9	functional multi-walled carbon nanotubes - water	2	Effects of magnetic field intensity, wall hydrophobicity and nanoparticle volume fraction on flow and heat transfer characteristics were investigated.

2.3. Molecular dynamic simulation (MDS) and direct simulation Monte Carlo (DSMC)

The molecular dynamics simulation (MDS) method is a simulation method based on classical mechanics, statistical mechanics, and quantum mechanics in recent years. Since molecular dynamics is a simulation method based on a molecular and atomic level, this method is also widely used in the problem of fluid flow and heat transfer in microchannels. In molecular dynamics simulation, once the velocity and position of particles are determined, the past and future states can be budgeted [128]. Under suitable boundary conditions, the motion path of a large number of interacting particles can be effectively predicted by solving relevant governing equations [79]. However, as one of the most accurate methods to simulate complex systems in engineering, this method requires high computational resources and is currently only applicable to small systems, and the scope of application of [78] remains to be expanded.

Zarringhalam et al. [78] investigated the effect of conical barrier on argon flow in microchannels under different wall temperatures by molecular dynamics methods and compared the argon flow characteristics in smooth and rough microchannels at temperature ranging from 84 K to 133 K. Their results indicated that the tapered rough components in the microchannel could strengthen the boiling heat transfer. Peng et al. [79] conducted MDS to analyze the effect of silver nanoparticles on the flow of Argon base fluid in cubic microchannels in the case of boiling. They found that nanoparticles in Argon base fluid could strengthen the boiling heat transfer and reduce the time required for boiling. Yan et al. [80] employed the MDS to simulate the effects of boundary temperature on Argon flow in smooth and rough microchannels at different wall temperatures ranging from 84 K to 133 K. They concluded that rough elements can effectively reduce the amount of Argon atoms entering the channel centre from the wall layer. Rostami et al. [81] used the MDS to investigate the effects of barriers with cubic geometry in smooth and rough microchannels on the boiling flow of argon driven by different external forces. Their report indicated that due to the effect of the arrangement of Argon atoms, the cubic roughness element could increase the density distribution oscillations. Goldanlou et al. [82] applied the MDS to analyze the effect of roughness with a cone shape in microchannels on the change in argon flow characteristics in phase change conditions. It was found that the cone geometry of roughness elements could effectively enhance the heat transfer between the walls and the fluid, and the rough element has little effect on the flow velocity of the fluid. Che et al. [83] used the non-equilibrium MDS to analyze the process of Helium Oscillation in Microchannel Pulse Tube. Their simulation results showed that the energy flow density of acoustic waves was high when the forced oscillation time was lower than the natural oscillation time. After that, the research of [84] used MDS to effectively simulate the thermodynamic process of the coupling between the microchannel pulse tube and the active piston. Their simulation results presented that when the piston was located in the middle of the tube, the pressure amplitude was smaller than that at both ends of the tube. Dehkordi et al. [85] applied the MDS to predict the movement of water / Fe3O4 nanofluids in copper microchannels with the application of an electric field. They found that the nano-particles could enhance the heat transfer under the external electric field. Arjmandfard et al. [86] employed the MDS to analyze the thermal behavior of water and water/ Fe nanofluids in nonidea microchannels with atomic porosity. Their results indicated that the density, speed, and temperature of the nanofluid reached the optimum value in the case of three nanoparticles and 5% porosity. Afterwards, the research of [8] simulated water/iron nano fluid containing one nanoparticle. Their simulation results showed that the velocity, temperature, and density of nanofluids could be increased with the increase of the radius of nanoparticles.

Similarly, Mosavi et al. [87] used the MDS method to study the effect of spherical roughness barrier on the boiling flow of argon atoms in a square cross-section microchannel. Their research analyzed the distribution of flowing atoms at different temperatures and concluded that the spherical roughness barrier layer in the microchannel not only has no destructive effect on the boiling flow but also enhances the boiling effect. Dehkordi et al. [88] applied the MDS method to analyze seven important parameters such as potential energy, atomic energy, and kinetic energy of H_2O/Fe_3O_4 nanofluids in different atomic microchannels under an electric field. Their results found that the external electric field had a significant influence on the exercise behavior of nanofluid.

As mentioned above, the characteristic scale of the LBM method model can reach micron and nanometer. Unlike the LBM method, the molecular dynamics method can simulate the motion details of atoms and molecules, similar to the simulation of argon atom and water/Fe_3O_4 nanofluid mentioned above by analyzing the control parameters such as particle density, radius, velocity, and temperature, the thermodynamic properties of the fluid are analyzed in detail. In other words, compared with the CFD method and LBM method, the MDS method could be the most suitable method to study the change characteristics of particles in the process of fluid motion and the influence factors of particles on the thermodynamic properties of the fluid.

Over the past few decades, the DSMC method has become one of the main tools to predict the flow state of thin gas. It solves the Boltzmann equation for direct statistical simulation of molecular processes based on kinetic theory. Therefore, the DSMC method can describe the gas flow in the microchannel, and it has been widely used in predicting the flow of thin gas in the microchannel. The basic principle of this method is to decouple the motion and collision of particles in a time step [125]. The simulation procedure of DSMC mainly includes the following steps: (1) read the number of grids and record the boundary condition information. (2) initialize the flow field and calculate the number of particles entering. (3) simulation of particle motion and interaction. (4) mark all simulated particles. (5) probability selection and collision of simulated particles. (6) using grid cell and wall information, repeat steps (3)-(5), until the flow field is stable. (7) write flow filed and wall information [89].

Rath et al. [90] numerically analyzed the two-dimensional Burnett equation of gas flow in a long microchannel. The pressure boundary conditions were obtained by DSMC, and used the perturbation analysis momentum equation. Their results showed that the pressure distribution was related to the flow direction and the normal position of the wall, and obtained the distribution expression of gas flow pressure in the microchannel. Shah et al. [91] used the DSMC method to simulate the nitrogen flow in the straight microchannel under seven different boundary conditions. Their results presented that the pressure, velocity and translation temperature of microchannels under different boundary conditions would have the same variation trend. Taassob et al. [92] applied the DSMC method to analyze the flow characteristics of the thin gas at the corner of the microchannel, and the thermal behaviour of the thin gas at the corner was studied by using the pressure-driven implicit boundary conditions. Their results found that increasing the corner radius could improve the quality flow. Shariati et al. [93] applied the DSMC method to study the fluid flow and gas transport in porous microchannels. Their results verified the ability of DSMC to solve microporous media, and concluded that this method could effectively simulate micro-porous media with porosity of 40%. Ebrahimi et al. [94] used the DSMC method to predict the flow and heat transfer characteristics of pressure-driven nitrogen in divergent microchannels, and analyzed the Knudsen number from slip to free molecular rarefaction regimes, as well as studied the effects of microchannel divergence angle, inlet and outlet pressure ratio and sparsity on the thermal field and flow field. Their results found that the effect of the heat flux gradient on the direction of net heat flux increases with the increase of divergence angle. Mozaffari et al. [95] applied the DSMC method to simulate the thermal creep flow in the microchannel, and analyzed the Mach number, pressure and other parameters in the channel under different Knudsen numbers. Their results concluded that the gas flow was mainly affected by the wall temperature gradient, viscous forces and regional expansion effect. Gavasane et al. [96] employed the DSMC method to analyse the temperature change generated by the flow of rarefied gas in two-dimensional microchannels at high Knudsen number, and then a three-dimensional simulation was carried out [135]. Their outputs proposed the functional relationship between the temperature drop and aspect ratio, aspect ratio and pressure ratio of microchannels. In addition, Teschner et al. [97] reviewed five particle-based multi-scale and hybrid numerical simulation methods, including LBM, MDS and DSMC, and other dissipative particle dynamics (DPD) and smoothed particle hydrodynamics (SPH). Their results systematically summarized the application examples, differences, advantages and disadvantages of five particle methods, and provided reference for future research. Rumyantsev et al. [129] applied the DSMC method to simulate the propagation process of cesium atoms in microchannels. Their results

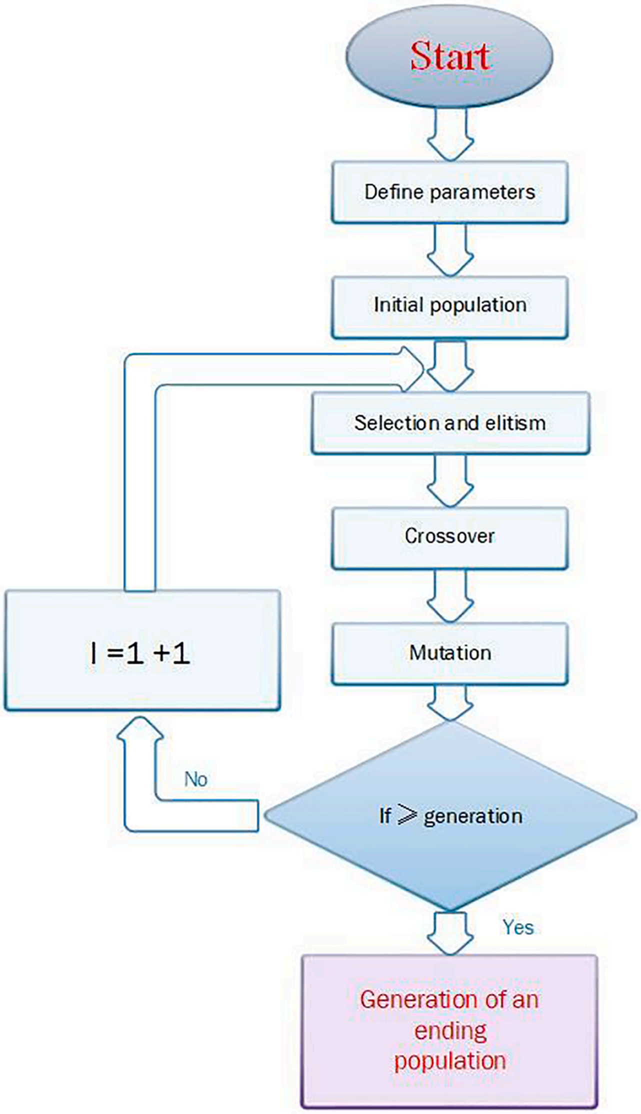


Fig. 6. Genetic algorithm flow chart.

indicated that the total flow rate of the atom outflow is 1.35 – 1.69 times lower than the theoretical flow rate. Roohi et al. [130] used the DSMC method to simulate the transition flow regimes at different Knudsen numbers. They found that a more accurate physical understanding could be obtained by applying back pressure in the buffer zone far from the outlet of the channel [127]. Afterwards, they simulated the subsonic flow of the gas in the microchannel when heated or cooled under different hot wall conditions. They stated that the gas heating will increase the compression effect in the channel, while the gas cooling will increase the mass flow rate.

2.4. *Taguchi method (TM), and genetic algorithms (GA)*

Taguchi method is normally used to optimize the design and improve the robustness of the product [109]. This method uses the orthogonal method to design the experimental plan and the signal-to-noise ratio (SNR) to determine the diversity of the experimental design [101]. Finally, the experimental data can be analyzed and the best combination of impact factors can be achieved. In general, the higher the signal-to-noise ratio, the better the performance of the optimized target. It should be noted that, however, due to the different content of the experiment, the requirements for the evaluation index are different, which is similar to Zhang's experiment, Eq. (1) and Eq. (2) [109] are used as evaluation criteria. Where Y is the index value and n is the number of repeated experiments. Different calculation formulas are used for different experiments.

$$SNR_{1b} = -10lg\left(\frac{1}{n}\sum_{i=1}^{n}\frac{1}{Y_i^2}\right) \tag{1}$$

$$SNR_{sb} = -10lg\left(\frac{1}{n}\sum_{i=1}^{n}Y_i^2\right) \tag{2}$$

It is recognized that the genetic algorithm (GA) is a computing method inspired by nature and made by Darwin's theory of evolution. This method can use the characteristics of genetic algorithms to seek the optimal solution in more complex experimental data, and because of the high efficiency of genetic algorithms in data processing, the GA can be applied in many fields. The calculation steps of the genetic algorithm are population initialization, selection, crossover, and mutation. Fig. 6 showed a typical calculation step of genetic algorithm which is similar to biological genetics.

Javadpour et al. [98] combined the Taguchi method with the multi-objective genetic algorithm (MOGA) method to predict the Nu and friction coefficient in the rectangular microchannel. They applied the Taguchi method to seek the best experimental scheme, and used the MOGA to optimise the rib shape and volume fraction of nano-particles in the microchannel. Their computed results showed that the efficiency of the predicted parameters for ideal response function design with a total absolute difference of 4% in Nu and 16% in friction coefficient. The proposed method could be an alternative way to design an effective microchannel for thermal systems. Moradkhani et al. [99] developed a new general semi-empirical model with 7328 data points collected from 35 sources to predict the frictional pressure drop in single and multiport mini/micro and macro channels using genetic programming (GP). They stated that the proposed model can predict well under different conditions when taking several control parameters into accounts, such as mass flux, channel diameter, saturation temperatures, and working fluids. Hosseini et al. [100] proposed a new general explicit correlation based on GP to predict the condensation heat transfer coefficient in horizontal mini and macro channels, their model was validated with a dataset consisting of 6521 data samples from 40 different sources that considering several control parameters such as various types of fluids, different cross-sectional geometries, mass fluxes, and saturation temperatures, into account. It was concluded that the proposed generalized correlation could fit the wide range of data points used with an average relative error of 17.82%, which gave better prediction performance compared with the other eight correlations available from the open published literature using the same database.

Jafari et al. [101] employed micro-Wire electrical discharge machining (μ-WEDM) to fabricate metal-based microchannel heat sinks with different surface textures and an experimental study based on the Taguchi technique was carried out. An artificial neural network model was also utilized to evaluate the variation of the surface roughness with process parameters. It was stated that the predictions were in very good agreement with results yielding a coefficient of determination of 99.5%, which is helpful to find the desired surface roughness to have a well-controlled flow and heat transfer characteristics for microchannels. Shi et al. [102] took advantage of GA, surrogate model, and CFD method, a three-dimensional ceramic microchannel heat exchanger model was established. Their results showed that the optimized model presented a stronger ability to predict the flow nonuniformity, they also reported that the fluid nonuniformity and pressure in the heat exchanger channel were reduced by 68.2% and 6.6% respectively. Al-Neama et al. [103] conducted a combined CFD simulation with experimental work to analyze the effect of herringbone fin on thermal resistance and convective heat transfer capacity of heat exchanger in the serpentine microchannel and applied the GA to optimize the design condition. Their results showed that the designed structure could reduce the

Table 3
Summary of Taguchi Method Optimization Experiments.

Author(s)	Target factors	Adjustable parameters	Test number
Jafari et al. [101]	Surface roughness	Pulse on, Pulse off, Peak voltage, Feed rate, Capacitance, Wire-speed	25
Lin et al. [108]	LED substrate temperature	TEC current, ambient temperature, water flow velocity, water inlet temperature	16
Zhang et al. [109]	The overall efficiency	Length, width, longitudinal spacing, and number of LVGs	16
Bazkhane and Zahmatkesh [113]	Mean temperature, thermal resistance, Pumping power	Inlet velocity, nanoparticles fraction, the material of the porous substrates, the thickness of the vertical substrates, the thickness of the horizontal substrates	27
Park et al. [114]	Mixing indices	Applied electric potential, position, size, heating intensity, inlet velocity ratio, aspect ratio	18

pressure drop by 60% and the total thermal resistance by 10%, and could increase the average Nusselt number by 15%. McCann et al. [104] used the TM to establish a model for predicting the width and depth of the microchannel. After analyzing the correlation coefficient and signal-to-noise ratio, their results showed that the Taguchi experiment model may have some limitations in detecting the edge of process space. Lin et al. [105] combined the FEM and GA to design a micro multichannel heat sink with minimum thermal resistance, the control parameters such as channel number and channel size were considered as optimisation variables. Their results showed that the thermal resistance was effectively reduced by 0.144 W/K. Wu et al. [106] proposed a two-layer compact microchannel model with a non-uniform heat source, they used the GA prediction model to obtain the best parameters for fluid velocity and fluid temperature and set the channel width as the prediction parameter. Their outputs revealed that the model can accurately predict the microchannel temperature, and the optimized model has uniform surface temperature distribution under extremely uneven heat sources. Yoshimura et al. [107] proposed a passive micromixer optimization method based on GA, after analyzing the characteristics of the groove and convex groove at the bottom of the microchannel using CFD, the design parameters were predicted using GA. Their results pointed out the effects of different width groove combinations, different number grooves, and different types of groove combinations on the performance of microchannels. Lin et al. [108] proposed a cooling device integrating thermoelectric cooler (TEC) and water-cooled microchannel heat sink based on Taguchi's experimental method. Through Taguchi orthogonal experiment and variance analysis, the effect of four control parameters such as temperature coefficient, water flow speed, water flow temperature, and ambient temperature on equipment cooling was obtained. Their results indicated that the combination of a thermoelectric cooler and water-cooled microchannel heat sink showed a good cooling effect on the LED. Table 3 lists some of the output parameters and adjustment parameters optimised by the Taguchi method. Table 4 summarizes the optimisation design parameters used by some researchers using genetic algorithms, including the number of individuals, crossover rate, mutation rate, and the maximum number of generations. Zhang et al. [109] combined the TM and RSM to establish a rectangular cross-section microchannel model with a longitudinal vortex generator (LVG). According to the basic principles of TM, the effect of the four control parameters such as LVG's length, width, number, and longitudinal spacing on the overall performance is analyzed, and the optimal Nu number is obtained. Their experimental results showed that after TM optimisation, Nu and the total efficiency could be improved to 23.6% and 7.2%, respectively. Their results also found that the number and spacing of longitudinal vortex generators could have a significant influence on the Nusselt number. Alperen et al. [110] predicted the geometric size, fluid flow rate, and Re of microchannels with rectangular cross-sections by multi-objective GA. The prediction has been carried out for 20 generations, and the population number is 30. Their optimisation results showed that the microchannel height has the best performance within 0.5 – 0.67 mm. Wang et al. [111] used the Multi-objective GA to predict the semi-porous ribbed double-layer microchannel heat sink structure. Their results indicated that the thermal resistance and pumping power in multi-objective optimization cannot meet the best requirements at the same time, and the optimised model cooling performance could be increased by 14.06%, and the pumping power could be reduced by 16.4%. Glazar et al. [112] combined RSM and GA to optimise the four geometric parameters of the microchannel coil air–water heat exchangers. Their study found that the best structural options available for different needs. Rabiei et al. [2] numerically investigated the influence of wavy-shaped fins on flow and heat transfer in cylindrical microchannels.

It employed the GA and the compromise programming method to optimise the fin structure, and the scheme with the best overall performance was obtained. Their results found that the fin can change the flow direction, disturb the boundary layer and enhance the heat transfer. The optimised results also showed that the concentration of nanoparticles and the fin amplitude could affect the pumping power and thermal attributes, respectively. Bazkhane and Zahmatkesh [113] applied Taguchi method to analyze the flow and heat transfer characteristics of alumina-water nanofluid in an MCHS with vertical/horizontal porous substrate.

Table 4
Optimising data of genetic algorithm in microchannel research.

Author(s)	Transfer		Parameters of Optimization Algorithms			
	Objective Functions	Design Variables	Number of Individuals	Crossover Rate	Mutation Rate	Maximum number of Generations
Shi et al [102].	Mass flow parameter, pressure drop	Rib origin position	500	0.8	–	300
Al-Neama et al [103].	Total pressure drop, total thermal resistance	Mini channel width, number of the mini channels, oblique angle	50	–	–	–
Lin et al [105].	Thermal resistance	Channel number, channel aspect ratio, the ratio of the channel width pitch	–	0.6	0.01	–
Alperen et al [110].	Averaged Nusselt number, pumping power	Channel height, channel width, inlet Re	30	0.6	0.1	20
Wang et al [111].	Thermal resistance, total pumping power.	The channel number of the heat sink, channel aspect ratio, channel-to-pitch width ratio, the ratio of volumetric flow rate	40	–	–	30
Glazar et al [112].	Heat transfer per mass, heat transfer per volume, heat transfer, and air/water pressure drops	Fin pitch, transversal MCHX tube row pitch, number of small channels per multiport tube, multiport tubes wall thickness	100	–	0.075	–

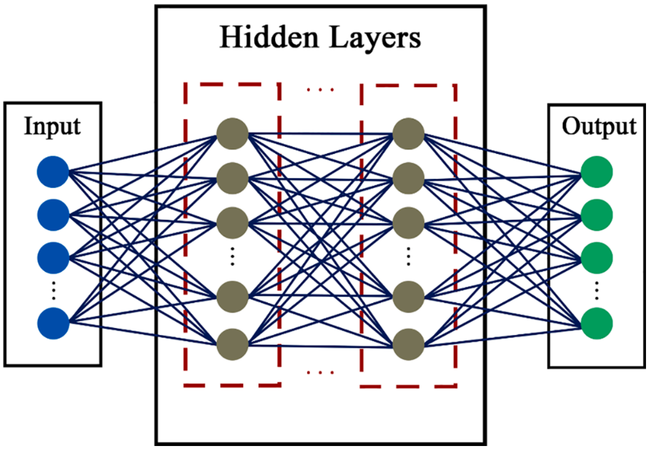


Fig. 7. Artificial neural network model configuration.

Their results indicated that the vertical/horizontal porous substrates could effectively reduce the total thermal resistance and pumping power, and the use of nanofluids could increase the pumping power while improving the overall performance. Park et al. [114] employed the Taguchi method to optimise the fluid mixing by alternating current electrothermal flow in a Y-shaped microchannel. Their results stated that the mixing efficiency of the two fluids can be affected by external potential, channel cross-section, and micro-heater. The mixing efficiency of the two fluids optimised by the Taguchi method is 90.4%.

It is knows that bothTM and GA are experimental optimisation methods. First of all, the TM can be seen from the above cases that the most critical step of this method is to establish an orthogonal table, and then introduce the signal-to-noise ratio. The appropriate orthogonal table will affect the accuracy and credibility of the final optimisation data, and the signal-to-noise ratio is the standard to evaluate the quality of optimisation parameters. According to the research of [101], the similarity between the prediction results and the experimental results using the Taguchi algorithm is 99.5%. It can be seen that if the TM can be used effectively, the experimental time and cost will be reduced. Secondly, the GA is different from the TM. The key point of the GA lies in the accuracy and applicability of coding. In the process of generating population, evaluation, selection, crossover, and mutation, this method can select the optimal data for researchers based on a large number of experimental data. For example, Moradkhani et al. [99] successfully predicted the desired control parameters from 7328 data, which showed the efficiency, accuracy, and wide applicability of the GA.

2.5. Machine learning algorithms

Most recently, machine learning-based modeling techniques, such as Artificial Neural Networks (ANNs), Decision Tree, Random Forest, Gradient Boosting, Adaptive Neuro-Fuzzy Inference Systems (ANFIS), and Support Vector Machines (SVM), have attracted growing attention and have been considered as a very promising methodology for analyzing thermal systems. Similar to genetic algorithm, as its name is general, ANN is to use the computer to imitate human brain nerve and establish different kinds of neural network models, to process a large number of data and information, and finally output the optimal results. In the actual calculation process, the input data is first provided to the input node, and then the information is forwarded to the node through the network until the output node, the calculation process is illustrated in Fig. 7.

Tafarroj et al. [115] predicted the heat transfer coefficient and Nusselt number in TiO2/water nanofluid flow in a microchannel heat sink with very limited data from their previous experimental study using a simple ANN model. Four inputs, i.e. volume fraction of nanoparticles, Re, heating rate, and model number, were introduced to the neural network. It was concluded that despite the limited number of experimental data, the average relative errors in the prediction of Nusselt number and heat transfer coefficients were 0.3% and 0.2, respectively. López-Belchí et al. proposed an ANN coupled with Group of Method Data Handling (GMDH) to predict the pressure drop with an accuracy of 88.63% and condensation heat transfer coefficient with an accuracy of 98.70% in two-phase flow systems in minichannels with hydraulic diameters of 1.16 and 0.70 mm and five different refrigerant fluids as the working fluid [116]. The methodology could be used for thermodynamic systems with complex flow dynamics in different operating conditions. Khosravi et al. [117] employed an ANN model with a single hidden layer and 10 neurons to predict the entropy generation rate for a cylindrical MCHS in an accurate manner compared with numerical simulation results. They stated that the ANN can properly predict entropy generation rates. Naphon et al. [118] conducted a combined Eulerian two-phase CFD model and ANN model with Levenberg-Marquardt Backwardpropagation (LMB) training algorithms to analyze the nanofluids jet impingement heat transfer and pressure drop in the microchannel heat sink. Both model results were verified with the measured data. The results revealed that the majority of the data obtained from the optimal ANN model were within ± 1.5% of the Nusselt number and pressure drop. The proposed ANN model is helpful to better understand the heat transfer and nanofluid characteristics in the microchannel heat sinks with various configurations. Qiu et al. [119] developed an ANN model based on the universal consolidated database of 16,953 data points that is amassed from 50 sources that include 16 working fluids with several control parameters such as reduced pressure, hydraulic diameter, mass velocity, liquid-only Reynolds number, and flow qualities to predict the saturated flow boiling heat transfer coefficients in mini/microchannels. They concluded that the ANN model was superior to universal correlations in predicting saturated flow boiling heat transfer even predicting individual databases with high accuracy. The stated that the ANN model worked very well when working fluid data was included in the training dataset. They concluded that the ANN model can become an extremely useful tool to predict heat transfer coefficients for saturated flow boiling in mini/microchannels. Most recently, with the taking off of deep learning, Liang et al. [120] proposed a deep ANN model to predict the boiling heat transfer in helical coils under high gravity conditions, which is also compared with experimental data. With the utilization of deep learning, the proposed model can successfully predict the heat transfer performance in helical coils, and especially achieved excellent performance in predicting outputs that have a very large range of value differences. Zhou et al. [121] applied four machine learning-based models, i.e. ANNs, Random Forest, AdaBoost, and Extreme Gradient Boosting (XGBoost), to predict condensation heat transfer coefficients in mini/microchannel with a consolidated database of 4,882 data points and compared for predicting accuracy. It was found that the ANN and XGBoost models gave the best predicting accuracy. The results also showed that the optimal ANN and XGBoost models could perform better than a highly reliable generalized flow condensation correlation. The results of this research work demonstrate that machine learning algorithms can become a robust new predicting tool for condensation heat transfer coefficients in mini/microchannels.

Various verification statistics can be regularly employed to analyse the predictive results to present a regression model performance, including Coefficient of Determination (R2), Mean Squared Error (MSE), Mean Absolute Error (MAE), Mean Absolute Percentage Error (MAPE) (i.e. Mean Relative Error), Cosine Proximity, and Root Mean Squared Error (RMSE) [126].

The above metrics can be defined by the following Eqs. (3) - (9). R2 metric obtained by the following Eqs. (6) and (7) indicates how well the prediction of the regression model approximates the real data points. R2 value of 1 indicates that the regression predictions perfectly fit the data.

$$y_{mean}\frac{1}{n}\sum_{k=1}^{n} y_{true} \tag{3}$$

$$R^2 = 1 - \frac{\sum_{k=1}^{n}\left(y_{true} - y_{pred}\right)^2}{\sum_{k=1}^{n}\left(y_{true} - y_{mean}\right)^2} \in [0, 1] \tag{4}$$

$$MSE = \frac{\sum_{k=1}^{n}\left(y_{true} - y_{pred}\right)^2}{n} \in [0, +\infty] \tag{5}$$

$$MAE = \frac{\sum_{k=1}^{n}\left|y_{true} - y_{pred}\right|}{n} \in [0, +\infty] \tag{6}$$

$$MAPE = \frac{1}{n}\sum_{k=1}^{n}\left|\frac{y_{true} - y_{pred}}{y_{true}}\right| \in [0, +\infty] \tag{7}$$

$$\text{Cosine Proximity} = \frac{\sum_{k=1}^{n} y_{true_k} \times \mathrm{y}_{\mathrm{pred}_\mathrm{k}}}{\sqrt{\sum_{k=1}^{n} y_{true_k^2}} \times \sqrt{\sum_{\mathrm{k=1}}^{\mathrm{n}} \mathrm{y}_{\mathrm{true}_\mathrm{k}^2}}} \in [-1, 1] \tag{8}$$

$$RMSE = \sqrt{\frac{\sum_{k=1}^{n}\left(y_{true} - y_{pred}\right)^2}{n}} \in [0, +\infty]$$

where $\mathrm{y}_{\mathrm{true}}$ is the ground truth output (experimentally observed output data), $\mathrm{y}_{\mathrm{pred}}$ is the predicted output, $\mathrm{y}_{\mathrm{mean}}$ is the mean of the ground truth data, n is the total number of data.

MSE is the most commonly used regression loss function. It is the sum of squared distances between ground truth value and predicted value. It is always non-negative, and values closer to zero are better. MAE is the average of all absolute errors, which is another useful measure that is extensively employed in model evaluation. MAPE is the most common measure used to forecast error, and is sometimes reported as a percentage, which is the above equation multiplied by 100 to make it a percentage error. Cosine proximity or cosine similarity is the measure of similarity between two vectors. High cosine proximity indicates a higher accuracy. Perfectly opposite vectors have cosine proximity of -1; perfect orthogonal vectors have cosine proximity of 0, and identical vectors (such as the perfect fit of predicted values to the ground truth values) have cosine proximity of 1. RMSE is the square root of the ratio of the square of the deviation between the predicted value and ground truth value to the number of data n. The RMSE has a value equal to or greater than 0, where 0 depicts a statistically perfect fit to the ground truth data.

The following experimenters are also used as an evaluation standard for ANN methods. Heshmatian and Bahiraei [122] combined the CFD method with the ANN method to analyze the thermodynamics of nanoparticle migration in microchannels. According to the CFD simulation, they obtained the required fluid parameters, such as heat transfer coefficient at different Re. Then, they used a multilayer perceptron ANN to simulate the total entropy, friction entropy, and entropy yield of nanofluids. They concluded that the ANN algorithm can effectively save computing time, and the output data also has greater accuracy. Xiang et al. [123] based on the ANN predicted the thermal conductivity of the liquid alloy coolant in the microchannel heat sink, according to the existing experimental data, the researchers established two back-propagation artificial neural networks to predict the unknown and specific thermal conductivity. Combined with the classical theoretical model, they calculated the viscosity and density of the liquid and compared the predicted data with the measured data, they found that the prediction accuracy of the ANN is high, which proves the ANN is a powerful tool to optimize the microchannel heat sink. Giannetti et al. [124] established a model to predict the distribution of two-phase flow in the microchannel heat exchanger by the ANN. Comparing the previous experimental data with the output data of their model, it is found out that most of the deviations are less than 10%, and the highest correlation index is more than 98%. Based on the unsteady Bernoulli equation, Shen et al. [125] combined the backpropagation neural network method, CFD method, particle swarm optimization algorithm, and GA, to predict the resistance factor of the rectangular snake-shaped microchannel, and established the inertial transient flow model. According to the numerical simulation, they concluded that the width, height, and radius of the microchannel are the important control parameters that affect the resistance factor, the BP algorithm is used to optimize the simulation results, and the experimental data are compared with the simulation data. It was found that this model can effectively predict the change of flow with time, which further verifies the feasibility of the neural network to predict the resistance coefficient.

3. Conclusions and research directions

Due to its small size and high heat transfer performance, the microchannel heat sink has applications in several important fields including aerospace, automotive, microelectronics, power and process industries, refrigeration and air conditioning, cooling of gas turbine blades, etc. As far as we know, this is the first comprehensive review of microchannel research by analog methods. We tried to provide some research cases from different perspectives, to overcome the problems of incompatibility, inappropriateness, and low efficiency in specific research content and use methods such as CFD, LBM, MDS, followed by GA, TM, ANN, and other data optimisation methods. The characteristics, scope of application, advantages, and disadvantages of different methods are discussed, respectively. In this review, some typical application cases are cited and critical discussions are presented. This review can provide an opportunity for other researchers in selecting suitable methods for different research problems.

Based on the above literature review, the following specific conclusions may be achieved:

1 With the rapid development of computers and CFD techniques, most of the complex and diverse problems can be solved numerically. This method becomes the preferred method for most researchers in microchannel research. It is recognized that CFD methods normally include the finite difference method, finite element method, finite volume method, and other solving algorithms. Besides, the CFD simulation method can simulate a variety of different boundary conditions and complex model structures, but the requirements for mesh quality would be high. For different experimental contents, the CFD method can modify the boundary conditions to determine the influencing control parameters. Most of the research on microchannel heat transfer and flow characteristics can be solved by this method.

2 Since the LBM method can study the characteristic scales in micrometers and nanometers, and its application is not limited by the quality of the mesh. Thus, most researchers use the LBM method to study the heat transfer and flow characteristics of fluids in complex structures. It is worth noting that, the LBM method is difficult to accurately capture the dynamic behavior of the interface when studying the phase change problem, and it will have a certain impact on this issue. Unlike the CFD method, the LBM method can determine the effects of particles on flow and heat transfer by observing the characteristics of the particles being studied. Therefore, this method is mostly used in the study of fluids in complex structures and complex boundary conditions.

3 The MDS method, which has a finer feature size than that of the LBM method model, can simulate the motion law of molecules and atoms, and study the thermodynamic properties of fluids by analyzing the radius, density, velocity, or temperature of atoms and molecules. However, it should be noted that there are little researches on the MDS method in the current microchannel. Therefore, there is still much room for further application of this method in the microchannel.

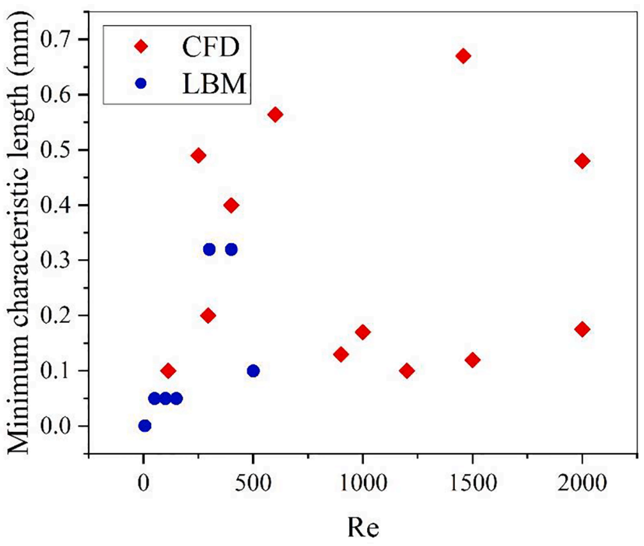


Fig. 8. Application scale and Reynolds number comparison: CFD and LBM.

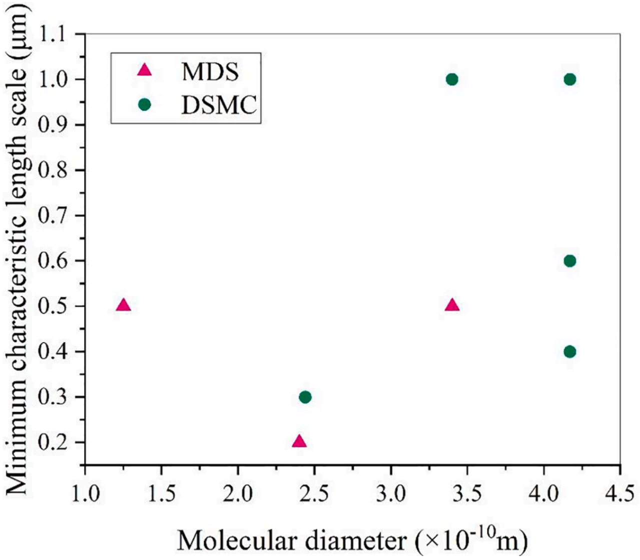


Fig. 9. Application scale comparison: MDS and DSMC.

4 The DSMC method is one of the main tools to predict the flow state of thin gas, which solves the Boltzmann equation for direct statistical simulation of molecular processes based on dynamics theory. Therefore, the DSMC method can describe the gas flow in the microchannel, and it has been widely used in the flow prediction of thin gas in the microchannel. However, DSMC has a huge amount of calculation. Compared with the LBM method, the high calculation cost makes DSMC difficult to be widely used. However, nowadays many researchers are attracted by the high accuracy of DSMC and continue to overcome the problem of high computational cost. Therefore, the DSMC method could need an effective method to solve the huge amount of calculations.

5 Fig. 8 and Fig. 9 draw the application range of various methods at different scales. Since the simulation can be performed using dimensional and non-dimensional units, the data selection in the figure can express the article of physical dimensions. Fig. 8 shows the application scale and Reynolds number range of the CFD method and LBM method. The minimum characteristic length means the hydraulic diameter of the microchannel. It can be seen intuitively that the LBM method can be widely used at low Reynolds number (Re ⟨500) and small size (Dh less than 0.32 mm) structures, whereas the CFD method has a wide range of scales and Reynolds numbers due to its mature development. Similarly, Fig. 9 shows the scope of application of MDS and DSMC methods. The minimum characteristic scale means studying the hydraulic diameter or channel minimum height (2D) in the computational domain. According to the above content, in MDS and DSMC methods, since most studies take argon atoms as the research content, therefore they all have the same molecular scale and computational domain scale. However, MDS is mostly applied to smaller calculation scales.Fig. 9.

6 Taguchi method (TM) is an effective statistical method which could reduce experimental cost and improve experimental quality. This method can be used to design an orthogonal array to observe the data and use the signal-to-noise ratio and variance to evaluate the experimental data in order to obtain a set of optimal experimental strategies. Using Taguchi's optimisation method, the operation is simple and the convergence speed is fast, which can enable researchers to have a deeper understanding of the control parameters.

7 A genetic algorithm (GA) is used to design a process similar to heredity through coding, the four steps of population initialization, selection, crossover, and mutation. This method can filter out the optimal data group from a large amount of data, and processing from a group of data at the same time. This method has strong robustness, which not only increases the calculation speed but also avoids the algorithm from falling into the local optimal solution [126]. One can select this method when dealing with large amounts of data or solving non-linear problems.

8 The artificial neural network (ANN) method and the genetic algorithm have the same capability, which can have strong robustness and have a strong ability to solve nonlinear problems. Their difference is that the ANN method consists of an input layer, a hidden layer, and an output layer. The parameters are input to the hidden layer through the input node, and then the information is transmitted to the output node. The ANN method has the ability of autonomous diagnosis and can process complex information and data. However, the current ANN method is not accurate enough and the speed is relatively slower than other algorithms, therefore this method has much room for further improvement. Further researches need to be explored in the future work.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] D.B. Tuckerman, R.F.W. Pease, High-performance heat sinking for VLSI, IEEE. 2 (1981) 126–129.
[2] S. Rabiei, R. Khosravi, M. Bahiraei, M. Raziei, A.A. Hosseini, Thermal and hydraulic characteristics of a hybrid nanofluid containing graphene sheets decorated with platinum through a new wavy cylindrical microchannel, Appl. Therm. Eng. 181 (2020), 115981.
[3] W. Ling, W. Zhou, C. Liu, F. Zhou, D. Yuan, J. Huang, Structure and geometric dimension optimization of interlaced microchannel for heat transfer performance enhancement, Appl. Therm. Eng. 170 (2020), 115011.
[4] M. Meyari, Z. Salehi, R. Zarghami, M. Saeedipour, Numerical investigation of particle separation in Y-shaped bifurcating microchannels, Particuology. 56 (2021) 142–151.
[5] M. Yang, B. Cao, Numerical study on flow and heat transfer of a hybrid microchannel cooling scheme using manifold arrangement and secondary channels, Appl. Therm. Eng. 159 (2019), 113896.
[6] Z. Feng, X. Luo, F. Guo, H. Li, J. Zhang, Numerical investigation on laminar flow and heat transfer in rectangular microchannel heat sink with wire coil inserts, Appl. Therm. Eng. 116 (2017) 597–609.
[7] A.A.A.A. Al-Rashed, A. Shahsavar, O. Rasooli, M.A. Moghimi, A. Karimipour, M. D. Tran, Numerical assessment into the hydrothermal and entropy generation characteristics of biological water-silver nano-fluid in a wavy walled microchannel heat sink, Int. Commun. Heat Mass Transf. 104 (2019) 118–126.

[8] A. Arjmandfard, D. Toghraie, B. Mehmandoust, M. Hashemian, A. Karimipour, Study the time evolution of nanofluid flow in a microchannel with various sizes of Fe nanoparticle using molecular dynamics simulation, Int. Commun. Heat Mass Transf. 118 (2020), 104874.

[9] H.-L. Liu, D.-H. Qi, X.-D. Shao, W.-D. Wang, An experimental and numerical investigation of heat transfer enhancement in annular microchannel heat sinks, Int. J. Therm. Sci. 142 (2019) 106–120.

[10] G. Lu, X. Zhai, Analysis on heat transfer and pressure drop of a microchannel heat sink with dimples and vortex generators, Int. J. Therm. Sci. 145 (2019), 105986.

[11] A.Y.M. Ali, E.M. Abo-Zahhad, M.F. Elkady, S. Ookawara, A.H. El-Shazly, A. Radwan, Temperature uniformity enhancement of densely packed high concentrator photovoltaic module using four quadrants microchannel heat sink, Solar Energy. 202 (2020) 446–464.

[12] M. Peng, L. Chen, W. Ji, W. Tao, Numerical study on flow and heat transfer in a multi-jet microchannel heat sink, Int. J. Heat Mass Transfer. 157 (2020), 119982.

[13] L. Lin, J. Zhao, G. Lu, X.-D. Wang, W.-M. Yan, Heat transfer enhancement in microchannel heat sink by wavy channel with changing wavelength/amplitude, Int. J. Therm. Sci. 118 (2017) 423–434.

[14] F.B. Abdul Hasis, P.M. Mithun Krishna, G.P. Aravind, M. Deepu, S.R. Shine, Thermo hydraulic performance analysis of twisted sinusoidal wavy microchannels, Int. J. Therm. Sci. 128 (2018) 124–136.

[15] Y. Lei, Z. Chen, Cooling heat transfer and pressure drop of supercritical CO2 in wavy microchannels with consistent and opposite crests and troughs, Int. J. Refrig. 109 (2020) 64–81.

[16] D. Sreehari, A.K. Sharma, On thermal performance of serpentine silicon microchannels, Int. J. Therm. Sci. 146 (2019), 106067.

[17] J. Gómez-Pastora, C. González-Fernández, M. Fallanza, E. Bringas, I. Ortiz, Flow patterns and mass transfer performance of miscible liquid-liquid flows in various microchannels: Numerical and experimental studies, Chem. Eng. J. 334 (2018) 487–497.

[18] E. Bayrak, A.B. Olcay, M.F. Serincan, Numerical investigation of the effects of geometric structure of microchannel heat sink on flow characteristics and heat transfer performance, Int. J. Therm. Sci. 135 (2019) 589–600.

[19] C. Zheng, M. Zhang, S. Qiu, H. Li, T. Wang, H. Wang, Numerical simulation and experimental investigation of gas-liquid two-phase flow in a complex microchannel, Chem. Eng. Sci. 230 (2021), 116198.

[20] A. Qaderi, J. Jamaati, M. Bahiraei, CFD simulation of combined electroosmotic-pressure driven micro-mixing in a microchannel equipped with triangular hurdle and zeta-potential heterogeneity, Chem. Eng. Sci. 199 (2019) 463–477.

[21] V. Hosseinpour, M. Kazemeini, A. Rashidi, Developing a metamodel based upon the DOE approach for investigating the overall performance of microchannel heat sinks utilizing a variety of internal fins, Int. J. Heat Mass Transfer. 149 (2020), 119219.

[22] K. Derakhshanpour, R. Kamali, M. Eslami, Effect of rib shape and fillet radius on thermal-hydrodynamic performance of microchannel heat sinks: A CFD study, Int. Commun. Heat Mass Transf. 119 (2020) 104928, https://doi.org/10.1016/j.icheatmasstransfer.2020.104928.

[23] W. He, R. Mashayekhi, D. Toghraie, O.A. Akbari, Z. Li, I. Tlili, Hydrothermal performance of nanofluid flow in a sinusoidal double layer microchannel in order to geometric optimization, Int. Commun. Heat Mass Transf. 117 (2020), 104700.

[24] C. Yue, Q. Zhang, Z. Zhai, L.i. Ling, CFD simulation on the heat transfer and flow characteristics of a microchannel separate heat pipe under different filling ratios, Appl. Therm. Eng. 139 (2018) 25–34.

[25] R. Mandel, A. Shooshtari, M. Ohadi, A "2.5-D" modeling approach for single-phase flow and heat transfer in manifold microchannels, Int. J. Heat Mass Transfer. 126 (2018) 317–330.

[26] Y. Ding, L.i. Jia, L. Yin, Y. Zhang, Z. An, Theoretical investigation on convective condensation annular flow of R410a inside rectangular microchannel, Int. J. Heat Mass Transfer. 131 (2019) 698–708.

[27] B.E. Burk, T.P. Grumstrup, T.A. Bevis, J. Kotovsky, T.M. Bandhauer, Computational examination of two-phase microchannel heat transfer correlations with conjugate heat spreading, Int. J. Heat Mass Transfer. 132 (2019) 68–79.

[28] M.S. El-Genk, M. Pourghasemi, Nusselt number and development length correlations for laminar flows of water and air in microchannels, Int. J. Heat Mass Transfer. 133 (2019) 277–294.

[29] M. Jiang, B. Zhou, Numerical study of flow regimes in microchannel with dynamic contact angle, Int. J. Hydrogen. Energ. 45 (2020) 29782–29790.

[30] A. Abdollahi, S.E. Norris, R.N. Sharma, Fluid flow and heat transfer of liquid-liquid Taylor flow in square microchannels, Appl. Therm. Eng. 172 (2020), 115123.

[31] A. Soleimani, A. Sattari, P. Hanafizadeh, Thermal analysis of a microchannel heat sink cooled by two-phase flow boiling of Al2O3 HFE-7100 nanofluid, Therm. Sci. Eng. Prog. 20 (2020), 100693.

[32] S. Chatterjee, K.C. Ghanta, A. Hens, Study of multiphase flow inside straight and spiral microchannel and effect of two phase flow on Dean's vortices, Chem. Eng. Res. Des. 165 (2021) 398–408.

[33] P. Kumar, Numerical investigation of fluid flow and heat transfer in trapezoidal microchannel with groove structure, Int. J. Therm. Sci. 136 (2019) 33–43.

[34] M. Trofa, G. D'Avino, L. Sicignano, G. Tomaiuolo, F. Greco, P.L. Maffettone, S. Guido, CFD-DEM simulations of particulate fouling in microchannels, Chem. Eng. J. 358 (2019) 91–100.

[35] J. Mohammadpour, A. Lee, M. Mozafari, M.R. Zargarabadi, A.S. Mujumdar, Evaluation of Al2O3-Water nanofluid in a microchannel equipped with a synthetic jet using single-phase and Eulerian-Lagrangian models, Int. J. Therm. Sci. 161 (2021), 106705.

[36] S. Zeng, P.S. Lee, Topology optimization of liquid-cooled microchannel heat sinks: An experimental and numerical study, Int. J. Heat Mass Transfer. 142 (2019) 118401, https://doi.org/10.1016/j.ijheatmasstransfer.2019.07.051.

[37] M. Yang, B. Cao, Multi-objective optimization of a hybrid microchannel heat sink combining manifold concept with secondary channels, Appl. Therm. Eng. 181 (2020), 115592.

[38] G.V. Kewalramani, A. Agrawal, S.K. Saha, Modeling of microchannel heat sinks for electronic cooling applications using volume averaging approach, Int. J. Heat Mass Transfer. 115 (2017) 395–409.

[39] J.i. Wang, J.M. Li, Y. Hwang, Modeling of film condensation flow in oval microchannels, Int. J. Heat Mass Transfer. 126 (2018) 1194–1205.

[40] A. Taher, B. Jones, P. Fiorini, L. Lagae, Analytical, numerical and experimental study on capillary flow in a microchannel traversing a backward facing step, Int. J. Multiphas. Flow. 107 (2018) 221–229.

[41] H.S. Cao, S. Vanapalli, H.J. Holland, C.H. Vermeer, H.J.M. ter Brake, Heat transfer and pressure drop in microchannels with isotropically etched pillars at sub-ambient temperatures, Int. J. Refrig. 98 (2019) 334–342.

[42] M.A. Bucci, A.D. Stazio, C. Chauveau, G. Dayma, G. Legros, P. Dagaut, S. Chibbaro, Numerical and experimental analysis of combustion in microchannels with controlled temperature, Chem. Eng. Sci: X. 4 (2019), 100034.

[43] A. Wei, X. Ren, S. Lin, X. Zhang, CFD analysis on flow and heat transfer mechanism of a microchannel Ω-shape heat pipe under zero gravity condition, Int. J. Heat Mass Transfer. 163 (2020), 120448.

[44] A.M. Laziz, K.Z. KuShaari, B. Azeem, S. Yusup, J. Chin, J. Denecke, Rapid production of biodiesel in a microchannel reactor at room temperature by enhancement of mixing behaviour in methanol phase using volume of fluid model, Chem. Eng. Sci. 219 (2020), 115532.

[45] M. Vajdi, F. Sadegh Moghanlou, E. Ranjbarpour Niari, M. Shahedi Asl, M. Shokouhimehr, Heat transfer and pressure drop in a ZrB2 microchannel heat sink: A numerical approach, Ceram. Int. 46 (2) (2020) 1730–1735.

[46] Y.S. Ong, KuZilati KuShaari, CFD investigation of the feasibility of polymer-based microchannel heat sink as thermal solution, Chinese. J. Chem Eng. 28 (4) (2020) 980–994.

[47] M. Fattahi, K. Vaferi, M. Vajdi, F. Sadegh Moghanlou, A. Sabahi Namini, M. Shahedi Asl, Aluminum nitride as an alternative ceramic for fabrication of microchannel heat exchangers: a numerical study, Ceram. Int. 46 (8) (2020) 11647–11657.

[48] X. Zhan, G. Chen, D. Jing, Optimal analysis of the hydraulic and mixing performances of symmetric T-shaped rectangular microchannel mixer, Fractals. 29 (2021) 2150042.

[49] M. Wan, N. Sidik, R. Saidur, Y. Asako, S.N.A. Yusof, A review of passive methods in microchannel heat sink application through advanced geometric structure and nanofluids: Current advancements and challenges, Nanotechnol. Rev. 9 (2020) 1192–1216.

[50] S. Soleimanikutanaei, E. Ghasemisahebi, C.-X. Lin, Numerical study of heat transfer enhancement using transverse microchannels in a heat sink, Int. J. Therm. Sci. 125 (2018) 89–100.

[51] A.A. Alfaryjat, H.A. Mohammed, N.M. Adam, D. Stanciu, A. Dobrovicescu, Numerical investigation of heat transfer enhancement using various nanofluids in hexagonal microchannel heat sink, Therm. Sci. Eng. Prog. 5 (2018) 252–262.

[52] L. Chai, L. Wang, X. Bai, Thermohydraulic performance of microchannel heat sinks with triangular ribs on sidewalls – Part 1: Local fluid flow and heat transfer characteristics, Int. J. Heat Mass Transfer. 127 (2018) 1124–1137.

[53] G. Wang, N. Qian, G. Ding, Heat transfer enhancement in microchannel heat sink with bidirectional rib, Int. J. Heat Mass Transfer. 136 (2019) 597–609.

[54] M.P. Vasilev, R.S. Abiev, R. Kumar, Effect of microchannel heat sink configuration on the thermal performance and pumping power, Int. J. Heat Mass Transfer. 141 (2019) 845–854.

[55] H. Shen, G. Xie, C. Wang, The numerical simulation with staggered alternation locations and multi-flow directions on the thermal performance of double-layer microchannel heat sinks, Appl. Therm. Eng. 163 (2019), 114332.

[56] D. Liu, X. Ling, H. Peng, J. Li, L. Duan, Experimental and numerical analysis on heat transfer performance of slug flow in rectangular microchannel, Int. J. Heat Mass Transfer. 147 (2020), 118963.

[57] L. Su, Z. Duan, B. He, H. Ma, X. Ning, G. Ding, Y. Cao, Heat transfer characteristics of thermally developing flow in rectangular microchannels with constant wall temperature, Int. J. Therm. Sci. 155 (2020), 106412.

[58] J. Khalesi, N. Sarunac, Z. Razzaghpanah, Supercritical CO2 conjugate heat transfer and flow analysis in a rectangular microchannel subject to uniformly heated substrate wall, Therm. Sci. Eng. Prog. 19 (2020), 100596.

[59] M.A.R.S. Al-Baghdadi, Z.M.H. Noor, A. Zeiny, A. Burns, D. Wen, CFD analysis of a nanofluid-based microchannel heat sink, Therm. Sci. Eng. Prog. 20 (2020), 100685.

[60] M.Q. Ansari, G. Zhou, Flow and heat transfer analysis of microchannels structured with rectangular surface roughness, Chem. Eng. Process. 156 (2020), 108066.

[61] K. Fallah, M. Taeibi Rahni, Lattice Boltzmann simulation of drop formation in T-junction microchannel, J. Mol. Liq. 240 (2017) 723–732.

[62] Y. Fu, L. Bai, K. Bi, S. Zhao, Y. Jin, Y.i. Cheng, Yi Cheng, Numerical study of Janus droplet formation in microchannels by a lattice Boltzmann method, Chem. Eng. Process. 119 (2017) 34–43.

[63] S. Ghadirzadeh, M. Kalteh, Lattice Boltzmann simulation of temperature jump effect on the nanofluid heat transfer in an annulus microchannel, Int. J. Mech. SCI. 133 (2017) 524–534.

[64] R. Kamali, M.N. Soloklou, H. Hadidi, Numerical simulation of electroosmotic flow in rough microchannels using the lattice Poisson-Nernst-Planck methods, Chem. Phys. 507 (2018) 1–9.

[65] J. Cai, X. Huai, B. Liu, Z. Cui, Numerical prediction of thin liquid film near the solid wall for hydraulic cavitating flow in microchannel by a multiphase lattice Boltzmann model, Int. J. Heat Mass Transfer. 127 (2018) 107–115.

[66] J.-F. Zhou, S. Zhang, F. Tian, C.-L. Shao, Simulation of oscillation of magnetic particles in 3D microchannel flow subjected to alternating gradient magnetic field, J. Magn. Magn. Mater. 473 (2019) 32–41.

[67] J. Zhou, G. Gu, X. Meng, C. Shao, Effect of alternating gradient magnetic field on heat transfer enhancement of magnetoreological fluid flowing through microchannel, Appl. Therm. Eng. 150 (2019) 1116–1125.

[68] A. D`Orazio, A. Karimipour, A useful case study to develop lattice Boltzmann method performance: Gravity effects on slip velocity and temperature profiles of an air flow inside a microchannel under a constant heat flux boundary condition, Int. J. Heat Mass Transfer. 136 (2019) 1017–1029.

[69] L. Yang, Y. Yu, H. Pei, Y. Gao, G. Hou, Lattice Boltzmann simulations of liquid flows in microchannel with an improved slip boundary condition, Chem. Eng. Sci. 202 (2019) 105–117.

[70] E.K. Ahangar, M.B. Ayani, J.A. Esfahani, Simulation of rarefied gas flow in a microchannel with backward facing step by two relaxation times using Lattice Boltzmann method – Slip and transient flow regimes, Int. J. Mech. SCI. 157-158 (2019) 802–815.

[71] M.S. Saravani, M. Kalteh, Heat transfer investigation of combined electroosmotic/pressure driven nanofluid flow in a microchannel: Effect of heterogeneous surface potential and slip boundary condition, Eur. J. Mech. B-Fluid. 80 (2020) 13–25.

[72] C.S. Wang, T.C. Wei, P.Y. Shen, T.M. Liou, Lattice Boltzmann study of flow pulsation on heat transfer augmentation in a louvered microchannel heat sink, Int. J. Heat Mass Transfer. 148 (2020), 119139.

[73] E.K. Ahangar, M.B. Ayani, J.A. Esfahani, K.C. Kim, Lattice Boltzmann simulation of diluted gas flow inside irregular shape microchannel by two relaxation times on the basis of wall function approach, Vacuum. 173 (2020), 109104.

[74] H.H. Afrouzi, M. Hosseini, D. Toghraie, E. Mehryaar, M. Afrand, Thermo-hydraulic characteristics investigation of nanofluid heat transfer in a microchannel with super hydrophobic surfaces under non-uniform magnetic field using Incompressible Preconditioned Lattice Boltzmann Method (IPLBM), Physica A. 553 (2020), 124669.

[75] H. Wang, Y. Fu, Y. Wang, L. Yan, Y. Cheng, Three-dimensional lattice Boltzmann simulation of Janus droplet formation in Y-shaped co-flowing microchannel, Chem. Eng. Sci. 225 (2020), 115819.

[76] Q. He, Y. Li, W. Huang, Y. Hu, D. Li, Y. Wang, Lattice Boltzmann simulations of magnetic particles in a three-dimensional microchannel, Powder. Technol. 373 (2020) 555–568.

[77] Y. Zhang, G. Xie, A. Karimipour, Comprehensive analysis on the effect of asymmetric heat fluxes on microchannel slip flow and heat transfer via a lattice Boltzmann method, Int. Commun. Heat Mass Transf. 118 (2020), 104856.

[78] M. Zarringhalam, H. Ahmadi-Danesh-Ashtiani, D. Toghraie, R. Fazaeli, Molecular dynamic simulation to study the effects of roughness elements with cone geometry on the boiling flow inside a microchannel, Int. J. Heat Mass Transfer. 141 (2019) 1–8.

[79] Y. Peng, M. Zarringhalam, M. Hajian, D. Toghraie, S.J. Tadi, M. Afrand, Empowering the boiling condition of Argon flow inside a rectangular microchannel with suspending Silver nanoparticles by using of molecular dynamics simulation, J. Mol. Liq. 295 (2019), 111721.

[80] S.R. Yan, N. Shirani, M. Zarringhalam, D. Toghraie, Q. Nguyen, A. Karimipour, Prediction of boiling flow characteristics in rough and smooth microchannels using molecular dynamics simulation: Investigation the effects of boundary wall temperatures, J. Mol. Liq. 306 (2020), 112937.

[81] S. Rostami, M. Zarringhalam, A. Alizadeh, D.a. Toghraie, A.S. Goldanlou, Molecular dynamic simulation of Argon boiling flow inside smooth and rough microchannels by considering the effects of cubic barriers, J. Mol. Liq. 312 (2020), 113130.

[82] A.S. Goldanlou, M. Zarringhalam, N. Shirani, A. Alizadeh, D. Toghraie, S. Rostami, Investigation the effects of an external driving force and cone shape of roughness on the phase change behavior of Argon fluid within a microchannel by molecular dynamic simulation, J. Mol. Liq. 313 (2020), 113503.

[83] Y. Che, Y. Qi, S. Pan, S. Niu, H. Zhang, Microscopic mechanism of alternating oscillations in a microchannel pulse tube based on molecular dynamics, J. Mol. Liq. 313 (2020), 113486.

[84] Y. Che, Y. Qi, S. Niu, Y. Liu, H. Zhang, Simulation on alternating oscillation flow in microchannel pulse tube coupled with active piston using non-equilibrium molecular dynamics, Chem. Phys. Lett. 759 (2020), 137965.

[85] R.B. Dehkordi, D. Toghraie, M. Hashemian, F. Aghadavoudi, M. Akbari, Molecular dynamics simulation of ferro-nanofluid flow in a microchannel in the presence of external electric field: Effects of Fe3O4 nanoparticles, Int. Commun. Heat Mass Transf. 116 (2020) 104653, https://doi.org/10.1016/j.icheatmasstransfer.2020.104653.

[86] A. Arjmandfard, D. Toghraie, B. Mehmandoust, M. Hashemian, A. Karimipour, The study of atomic porosity effect on water/Fe nanofluid flow in a microchannel with a molecular dynamics method, J. Mol. Liq. 317 (2020), 114291.

[87] A. Mosavi, M. Zarringhalam, D. Toghraie, A. Rahmani, A. Karimipour, Boiling of Argon flow in a microchannel by considering the spherical geometry for roughness barriers using molecular dynamics simulation, J. Mol. Liq. 321 (2021), 114462.

[88] K.G. Dehkordi, A. Karimipour, M. Afrand, D. Toghraie, A.H.M. Isfahani, Molecular dynamics simulation concerning nanofluid boiling phenomenon affected by the external electric field: Effects of number of nanoparticles through Pt, Fe, and Au microchannels, J. Mol. Liq. 324 (2021), 114775.

[89] G.-M. Guo, Q. Luo, L. Zhu, Y.-X. Bian, Flow characteristics of supersonic gas passing through a circular micro-channel under different inflow conditions, Chinese Physics B 28 (6) (2019) 064702, https://doi.org/10.1088/1674-1056/28/6/064702.

[90] A. Rath, N. Singh, A. Agrawal, A perturbation-based solution of Burnett equations for gaseous flow in a long microchannel, J. Fluid Mech 844 (2018) 1038–1050.

[91] N. Shah, A. Gavasane, A. Agrawal, U. Bhandarkar, Comparison of Various Pressure Based Boundary Conditions for Three-Dimensional Subsonic DSMC Simulation, J. Fluids Eng 140 (2018) 3.

[92] A. Taassob, R. Kamali, A. Bordbar, Investigation of rarefied gas flow through bended microchannels, Vacuum 151 (2018) 197–204.

[93] V. Shariati, M.H. Ahmadian, E. Roohi, Direct Simulation Monte Carlo investigation of fluid characteristics and gas transport in porous microchannels, Sci. Rep. 9 (1) (2019), https://doi.org/10.1038/s41598-019-52707-3.

[94] A. Ebrahimi, V. Shahabi, E. Roohi, Pressure-Driven Nitrogen Flow in Divergent Microchannels with Isothermal Walls, Appl. Sci 11 (2021) 18.

[95] M.S. Mozaffari, E. Roohi, On the Thermally-Driven Gas Flow through Divergent Micro/Nanochannels, Int J Mod Phys C 28 (2017) 12.

[96] A. Gavasane, A. Agrawal, A.M. Pradeep, U. Bhandarkar, Simulation of a temperature drop for the flow of rarefied gases in microchannels, Numer. Heat. Tr. 71 (10) (2017) 1066–1079.

[97] T.R. Teschner, L. Koenoezsy, K.W. Jenkins, Progress in particle-based multiscale and hybrid methods for flow applications, Microfluid. Nanofluid. 20 (2016) 68.

[98] S.M. Javadpour, E.A.J. Abadi, O.A. Akbari, M. Goharimanesh, Optimization of geometry and nano-fluid properties on microchannel performance using Taguchi method and genetic algorithm, Int. Commun. Heat Mass Transf. 119 (2020), 104952.

[99] M.A. Moradkhani, S.H. Hosseini, M. Valizadeh, A. Zendehboudi, G. Ahmadi, A general correlation for the frictional pressure drop during condensation in mini/micro and macro channels, Int. J. Heat Mass Trans. 163 (2020), 120475.

[100] S.H. Hosseini, M.A. Moradkhani, M. Valizadeh, A. Zendehboudi, M. Olazar, A general heat transfer correlation for flow condensation in single port mini and macro channels using genetic programming, Int. J. Refrig. 119 (2020) 376–389.

[101] R. Jafari, M. Kahya, S.N.B. Oliaei, H.Ö. Ünver, T.O. Özyurt, Modeling and analysis of surface roughness of microchannels produced by μ-WEDM using an ANN and Taguchi method, J Mech Sci Technol. 31 (11) (2017) 5447–5457.

[102] H.-N. Shi, T. Ma, W.-X. Chu, Q.-W. Wang, Optimization of inlet part of a microchannel ceramic heat exchanger using surrogate model coupled with genetic algorithm, Energ. Convers Manage. 149 (2017) 988–996.

[103] A.F. Al-Neama, Z. Khatir, N. Kapur, J. Summers, H.M. Thompson, An experimental and numerical investigation of chevron fin structures in serpentine minichannel heat sinks, Int. J. Heat Mass Transfer. 120 (2018) 1213–1228.

[104] R. McCann, K. Bagga, G. Duaux, A. Stalcup, M. Vázquez, D. Brabazon, Taguchi method modelling of Nd:YAG laser ablation of microchannels on cyclic olefin polymer film, Opt. Laser. Technol. 106 (2018) 265–271.

[105] D.T.W. Lin, C.H. Kang, S.C. Chen, Optimization of the Micro Channel Heat Sink by Combing Genetic Algorithm with the Finite Element Method, Inventions. 3 (2018) 32.

[106] R. Wu, X. Zhang, Y. Fan, R. Hu, X. Luo, A Bi-Layer compact thermal model for uniform chip temperature control with non-uniform heat sources by genetic-algorithm optimized microchannel cooling, Int. J. Therm. Sci. 136 (2019) 337–346.

[107] M. Yoshimura, K. Shimoyama, T. Misaka, S. Obayashi, Optimization of passive grooved micromixers based on genetic algorithm and graph theory, Microfluid Nanofluid. 23 (3) (2019), 30.1-30.21.

[108] X. Lin, S. Mo, L. Jia, Z. Yang, Y. Chen, Z. Cheng, Experimental study and Taguchi analysis on LED cooling by thermoelectric cooler integrated with microchannel heat sink, Appl. Energ. 242 (2019) 232–238.

[109] J.-F. Zhang, L. Jia, W.-W. Yang, J. Taler, P. Oclon, Numerical analysis and parametric optimization on flow and heat transfer of a microchannel with longitudinal vortex generators, Int. J. Therm. Sci. 141 (2019) 211–221.

[110] Y. Alperen, C. Sertac, Multi objective optimization of a micro-channel heat sink through genetic algorithm, Int. J. Heat Mass Transfer. 146 (2020), 118847.

[111] T.H. Wang, H.C. Wu, J.H. Meng, W.M. Yan, Optimization of a double-layered microchannel heat sink with semi-porous-ribs by multi-objective genetic algorithm, Int. J. Heat Mass Transfer. 149 (2020), 119217.

[112] V. Glazar, A. Trp, K. Lenic, Optimization of air-water microchannel heat exchanger using response surface methodology, Int. J. Heat Mass Transfer. 157 (2020), 119887.

[113] S. Bazkhane, I. Zahmatkesh, Taguchi–based sensitivity analysis of hydrodynamics and heat transfer of nanofluids in a microchannel heat sink (MCHS) having porous substrates, Int. Commun. Heat Mass Transf. 118 (2020), 104885.

[114] S. Park, H.S. Chuang, J.S. Kwon, Numerical study and Taguchi optimization of fluid mixing by a microheater-modulated alternating current electrothermal flow in a Y-shape microchannel, Sens. Actuators B Chem. 329 (2021), 129242.

[115] M.M. Tafarroj, O. Mahian, A. Kasaeian, K. Sakamatapan, A.S. Dalkilic, S. Wongwises, Artificial neural network modeling of nanofluid flow in a microchannel heat sink using experimental data, Int. Commun. Heat Mass Transf. 86 (2017) 25–31.

[116] A. López-Belchí, F. Illán-Gómez, J.M. Cano-Izquierdo, J.R. García-Cascales, GMDH ANN to optimise model development: Prediction of the pressure drop and the heat transfer coefficient during condensation within mini-channels, Appl. Therm. Eng. 144 (2018) 321–330.

[117] R. Khosravi, S. Rabiei, M. Bahiraei, A.R. Teymourtash, Predicting entropy generation of a hybrid nanofluid containing graphene-platinum nanoparticles

through a microchannel liquid block using neural networks, Int. Commun. Heat Mass Transf. 109 (2019), 104351.

[118] P. Naphon, S. Wiriyasart, T. Arisariyawong, L. Nakharintr, ANN, numerical and experimental analysis on the jet impingement nanofluids flow and heat transfer characteristics in the micro-channel heat sink, Int. J. Heat Mass Trans. 131 (2019) 329–340.

[119] Y. Qiu, D. Garg, L. Zhou, C.R. Kharangate, S.M. Kim, I. Mudawar, An artificial neural network model to predict mini/micro-channels saturated flow boiling heat transfer based on universal consolidated data, Int. J. Heat Mass Trans. 149 (2020), 119211.

[120] X. Liang, Y.Q. Xie, R. Day, X.H. Meng, H.W. Wu, A data driven deep neural network model for predicting boiling heat transfer in helical coils under high gravity, Int. J. Heat and Mass Transfer. 166 (2021), 120743.

[121] L. Zhou, D. Garg, Y. Qiu, S.M. Kim, I. Mudawar, C.R. Kharangate, Machine learning algorithms to predict flow condensation heat transfer coefficient in mini/micro-channel utilizing universal data, Int. J. Heat Mass Trans. 162 (2020), 120351.

[122] S. Heshmatian, M. Bahiraei, Numerical investigation of entropy generation to predict irreversibilities in nanofluid flow within a microchannel: Effects of Brownian diffusion, shear rate and viscosity gradient, Chem. Eng. Sci. 172 (2017) 52–65.

[123] X. Xiang, Y.u. Fan, A. Fan, W. Liu, Cooling performance optimization of liquid alloys GaIny in microchannel heat sinks based on back-propagation artificial neural network, Appl. Therm. Eng. 127 (2017) 1143–1151.

[124] N. Giannetti, M.A. Redo, Sholahudin, J. Jeong, S. Yamaguchi, K. Saito, H. Kim, Prediction of two-phase flow distribution in microchannel heat exchangers using artificial neural network, Int. J. Refrig. 111 (2020) 53–62.

[125] T. Shen, Z. Zhu, L. Huang, J. Chang, Transient flow behavior in serpentine curved microchannel of inertial microfluidic devices, J. Micromech. Microeng. 30 (2020), 025005.

[126] K. Deb, An introduction to genetic algorithms, Sadhana (1999).

[127] E. Roohi, M. Darbandi, V. Mirjalili, Direct simulation Monte Carlo solution of subsonic flow through micro/nanoscale channels, J. Heat Transfer. 131 (9) (2009) 1–8.

[128] Y. Peng, M. Zarringhalam, A.A. Barzinjy, D. Toghraie, M. Afrand, Effects of surface roughness with the spherical shape on the fluid flow of argon atoms flowing into the microchannel, J. Mol. Liq. 297 (2020), 111650.

[129] A.Y. Rumyantsev, M.V. Petrenko, S.A. Poniaev, Y.A. Shustrov, V.P. Kochegarov, D.L. Schennikov, The influence of interatomic collisions on the angular distribution of Cs atomic flow from microchannel, J. Phys. Conf. Ser. 929 (2017) 012095, https://doi.org/10.1088/1742-6596/929/1/012095.

[130] E. Roohi, M. Darbandi, V. Mirjalili, DSMC Solution of Supersonic Scale to Choked Subsonic Flow in Micro to Nano Channels, Am. Soc. Mechan. Eng. 62282 (2008) 23–25.

[131] A.A. Mehrizi, F. Besharati, O. Jahanian, H.H. Afrouzi, Numerical investigation of conjugate heat transfer in a microchannel with a hydrophobic surface utilizing nanofluids under a magnetic field, Phys. Fluids. 35 (2021), 052002.

[132] T.-M. Liou, T.-C. Wei, C.-S. Wang, Investigation of nanofluids on heat transfer enhancement in a louvered microchannel with lattice Boltzmann method, J. Therm. Anal. Calorim. 135 (1) (2019) 751–762.

[133] C. Yuan, Z. Pan, H. Wu, Inertial migration of single particle in a square microchannel over wide ranges of Re and particle sizes, Microfluid Nanofluidics. 22 (2018) 102.

[134] A.A. Lalami, H.H. Afrouzi, A. Moshfegh, Investigation of MHD effect on nanofluid heat transfer in microchannels, J. Therm. Anal. Calorim. 136 (2019) 1959–1975.

[135] N. Shah, A. Agrawal, U. Bhandarkar, 3D study of temperature drop behavior of subsonic rarefied gas flow in microchannel, Numer. Heat. Tr. 73 (2018) 1–12.