Numerical Investigation of Convective Heat Transfer In Microchannel Heat Sink With Corrugated Channels

Ganesh C[1], Kulandaivel D[2], Ramesh K[3]
[1, 2] Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[3]Professor, Dept of Mechanical Engineering
[1, 2, 3] Government College of Technology, Coimbatore -641 013, Tamil Nadu, India

Abstract- *For this thesis, a microchannel heat sink with corrugated channels was selected. The study looked at how the heat transfer performance of microchannel heat sink was affected by geometrical features related to the channel zone, such as flow disruption and corrugated channel. Concave, convex, convergent-divergent, and hybrid concave-convex channel designs were among the several that were introduced for effective cooling. It has been demonstrated that a number of interesting mechanisms, including the induction of flow mixing and the disruption of the boundary layer, enhance the effectiveness of this process of passive augmentation, or flow disruption. Reynolds number 50, 500, 2000 were used to assume the flow regime. The channel with concave cavity and convex ribs, one of the flow disruption techniques, was found to have a significant effect on the thermal performance of microchannel. Comparing a conduit with circular cavities to one without any interruptions, the pressure drop in the former was correspondingly smaller. Many techniques to obstruct flow were suggested, including expansion-contraction cavities, hybrid techniques between concave and convex ribs, and circular cavities. Concave-convex hybrid techniques are more advantageous than interference-free single-channel techniques. Better fluid mixing and a larger heat transfer area are examples of these qualities. The interrupted-wall channel with expansion-contraction that demonstrated a minimal pressure drop is the most important characteristic of flow disturbance techniques.*

Keywords- Microchannel heat sink(MCHS), Corrugated channel, concave cavities, Flow disruption, Hybrid methods.

I. INTRODUCTION

For a wide variety of purposes, humans have employed heat as an energy source. As manufacturing processes and industrial needs have grown, people have discovered and produced a wide range of gadgets to meet these needs. Modern applications use a broad range of heat exchanger types to control heat in various scenarios. The increasing demand for regulated environments is leading to a greater specialization and diversity among exchangers. Heat exchangers have application in the petrochemical, end-user, and chemical industries. Researchers' efforts to increase their understanding of heat transfer in a variety of applications have been spurred by the expansion of the energy sector. [1, 2]

Heat has been utilized by humans for a wide range of purposes. Humanity has discovered and produced a vast range of devices in response to growing industrial needs and expanding manufacturing processes. Many types of heat exchangers are utilized in current applications to handle heat in various scenarios. Exchangers are become more varied and specialized as a result of the increasing demand for controlled environments. The petrochemical, end-user, and chemical industries all use heat exchangers. The energy industry's rapid expansion prompted scientists to carry out investigations aimed at deepening their understanding of heat transport in various contexts. [1,2]

Because the channels of traditional straight Microchannel heat sinks are so tiny, the majority of the flow occurs in the laminar flow regime, keeping the flow from transitioning into the turbulent regime. Furthermore, in the conventional straight MCHS, the continual expansion of the thermal boundary layer causes hotter fluid to accumulate at the channel wall and cooler fluid to accumulate throughout the channel core, lowering the heat transmission in MCHS. To maximize the thermal performance of conventional straight rectangular MCHS, most of the early research concentrated on altering the channel aspect ratio, channel length, and wall thickness. [3-6]

For the purpose of improving performance, some researchers changed the cross-sectional shape of microchannel to something like a circle, triangle, or trapezoidal shape. [7-12]

The characteristics of the serpentine flow path are advantageous for microfluidic cooling applications, including wavy, zigzag, and convergent-divergent MCHS. Sui et al. [13] statistically examined the laminar flow and HT in 3D wavy MCHS using a rectangular cross-section. The dynamic technique is used to determine the fluid mixing properties (Poincare section). The results demonstrated the creation of a secondary flow (Dean Vortices) when liquid passed through wavy MCHS. Chaotic advection (CA) is created when the location and quantity of vortices vary along the direction of flow, facilitating convective fluid mixing. Wavy MCHS enhances its heat transmission capabilities as a result.

Heat transfer and flow in sinusoidal MCHS with rectangular cross sections are studied experimentally [14]. Each MC consists of ten wavy units with a wavelength of 2.5 mm, a depth of 404 µm, an average width of 205 µm, and wavy amplitudes of 138 µm and 259 µm. Deionized water is a working fluid with a variable Reynolds number ranging from 300 to 800. The Nusselt numbers of wavy and straight microchannels are compared when their cross sections and footprint lengths are equal. The wavy microchannel exhibits much superior heat transfer performance than the straight ones, with an enhanced heat transfer efficiency factor of up to 211%.Furthermore, it is discovered that, with a maximum increase in the PD penalty factor of 76%, the Pressure drop (PD) penalty of wavy microchannels is much lower than that of the heat transfer efficiency. The presence of chaotic advection was confirmed by numerical simulation using particle tracking and Poincare sections, which greatly enhanced fluid mixing and heat transfer. Meanwhile, the dean vortice patterns observed in the wavy microchannel cross section along the flow direction showed a significant variation. Sui et al. [44] examined the fully developed flow and heat transfer in a periodic wavy channel with a rectangular cross-section, where the Reynolds numbers span equivalent to constant laminar to transitional flow regimes.It was observed that dean vortices, or a symmetric secondary flow, formed as the liquid flow passed through the bends. The flow changed from a steady state to a single-frequency periodic one when the Reynolds number increased even higher. The flow in this stage is described by complex dean vortices with a temporal and spatial evolution pattern along the flow direction.

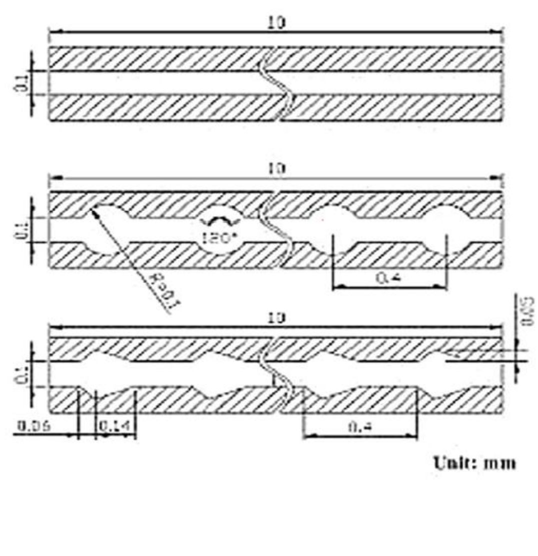
Mohammed et al.'s quantitative investigation of laminar forced convection in wavy MCHS with a rectangular cross-section was conducted [45]. The investigation includes a large range of Reynolds number (100–1000) and wavy amplitude (125–500). The heat sink consists of 25 wavy microchannels with a hydraulic diameter of 339 µm and a total length of 10 mm. The heat transmission performance of a wavy microchannel is found to increase with increasing wavy amplitude, with the exception of 500 µm. Additionally, the pressure drop and friction factor increase in proportion to the wave amplitude.Abed et al. [48] examined fluid flow and heat transfer in wavy MCHS with square cross-section by varying the dean number between 0.07 and 106.5 for two Prandtl number values (137 and 1038). A working fluid with glycerin and water in various weight percent ratios (30:70 and 10:90) is needed to get the necessary Prandtl values. The Nusselt number and friction factor increase when the dean number increases because to the better flow mixing. Additionally, there is an increase in thermal efficiency, with Prandtl numbers of 137 and 1038, respectively, producing ratios of 85% and 53%.

II. NUMERICAL MODEL AND METHOD

The use of micro-channel heat sinks in effective electronic cooling systems is growing in popularity. The effectiveness and functionality of these systems were the subject of numerous studies. Relationships between pressure drops and heat transfer were presented by many authors.

Researchers can now create more suitable designs because to improvements in CFD software. Pressure drops, heat transfer rates, design mass-flow rates, and fluid dynamic forces like lift and drag can all be calculated using CFD. It makes use of the finite element approach of discretization, which is used in structural and thermal analysis. The division of the mathematical model into non-overlapping fundamental geometric components is the main concept in the physical interpretation of the finite element method.

ANSYS design modeler is used to develop the channel geometry. For the current project, four distinct designs have been considered. The uninterrupted microchannel heat sink is represented by Model 1. When fluid is moving through the channel, there is no interruption to the flow. In Fig.1, reference geometry is displayed. According to the reference study by Ihsan Ali Ghani et al., the inlet's diameter is assumed to be 0.1 mm, and the channel's length is assumed to be 10 mm.



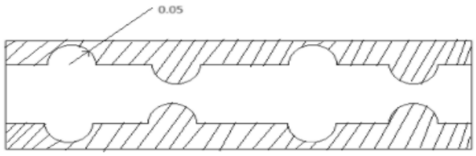


Fig.1 The schematic of four configurations of the Microchannels.

Interfaces were defined as surfaces in the 2D model. The model's mesh was created using the ANSYS Workbench Mesher program. In order to capture precise thermal and flow boundary layer, 10.8 million quadrilateral mesh elements are taken into consideration, with the first element height of 3 μm determined using Y+ value. four distinct configurations' modeled geometry are displayed in Fig. 2(a-d).

Fig 2a Modeled geometry1.

Fig 2b Modeled geometry2.

Fig 2c Modeled geometry3.

Fig 2d Modeled geometry4.

Reynolds number 50, 500, 2000 were used to assume the flow regime.The flow and heat transfer are modelled using the continuity, momentum and energy formulations represented by equations (1)– (4) [18].

$$\nabla.U = 0 \qquad \ldots\ldots\ldots\ldots(1)$$
$$U.\nabla(\rho U) = -\nabla p + \nabla.(\mu.\nabla U) \qquad \ldots\ldots\ldots\ldots..(2)$$
$$U.\nabla(\rho)C_p T_f = \nabla.(k_f \nabla T_f) \qquad \ldots\ldots\ldots\ldots..(3)$$
$$\nabla.(k_s \nabla T_s) = 0 \qquad \ldots\ldots\ldots\ldots..(4)$$

The material properties in the preceding equations are represented by the letters ρ, k, μ, and Cp, and the velocity and temperature are denoted by U and T, respectively. The governing equations for the solid and liquid domains were solved in this work using the finite volume method and the Ansys program. The geometry was constructed using the ANSYS DESIGN Modeler. The SIMPLE algorithm was used to replicate the pressure, temperature, and velocity computations.Additionally, the consistency requires that the residual for the continuity, momentum, and energy equations be smaller than 10-6 and 10-9, respectively, in order to obtain the appropriate convergence of the Navier-Stokes equations. To make sure that the mesh size has no effect on the solution, a mesh independence analysis is carried out. A steady state depending on pressure was simulated by the mesh geometry. The -y direction was loaded with gravitational forces. The k-epsilon, enhance wall treatment, constant wall temperature, and curvature effect were selected as the viscous model.

Based on the reference study, 373K was selected as the wall temperature. The references provided the flow Reynolds numbers of 50, 500, and 2000. Water is the fluid, and at 300K, its density (ρ=1000 kg/m3) and viscosity (μ=0.001 kg/m.s) are both constant.

Region	**Boundary condition**	**Expression**
Inlet	Velocity-inlet	u = 0.5 m/s, T = 300K u = 5 m/s, T = 300K u = 20 m/s, T = 300K
Outlet	Pressure-outlet	P = 0
Wall	Constant wall temperature	T = 373K

III. NUMERICAL RESULTS AND DISCUSSION

Laminar forced convection with different channel surfaces has been numerically explored in microchannel heat sinks, covering Reynolds numbers in the range of 50, 500, and 2000. A variety of numerical simulations using a fluid (water) on the channel side and the same inlet temperature are run on the ANSYS. Water is the working fluid on the channel surface in each and every instance. The first design considers a continuous heat sink with ribs that have a hydraulic diameter of 0.1 mm and a total length of 10 mm. The heat sink in the second design is made up of four 0.05 mm-diameter circular cavities known as corrugated channels.The channel surface of the third design saw the introduction of expansion and contraction, while maintaining the same inlet diameter of 0.1 mm and length of 10 mm. With the previously indicated diameter and length, a hybrid approach combining concave and convex cavities was established in the fourth design. At a low Reynolds number, Re = 50, the initial three designs exhibited identical behavior. However, in comparison to the other three designs, the fourth one performed better in terms of heat transfer. Fig. 3(a-d) displays the temperature profile and the velocity profile. Fig. 7 displays the temperature versus outlet length curve.

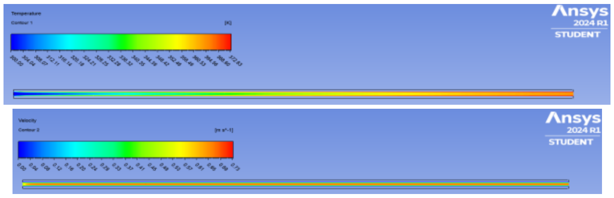


Fig. 3a. Temperature and velocity profile for Model1.

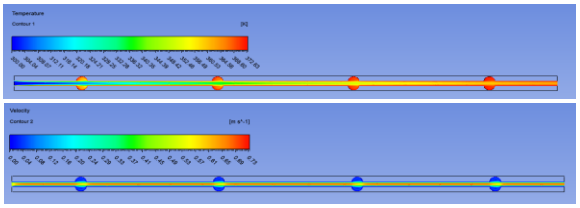


Fig. 3b. Temperature and velocity profile for Model2.

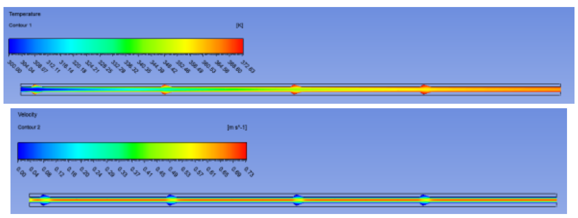


Fig. 3c. Temperature and velocity profile for Model3.

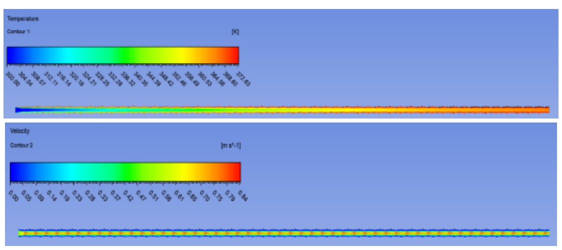


Fig. 3d. Temperature and velocity profile for Model4.

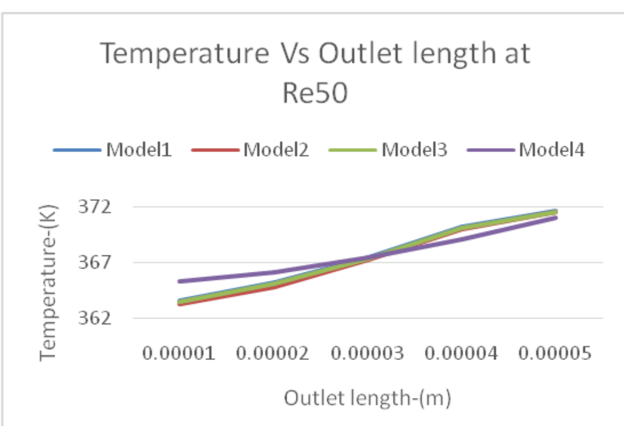


Fig. 4. Temperature vs Outlet length graph at Re50.

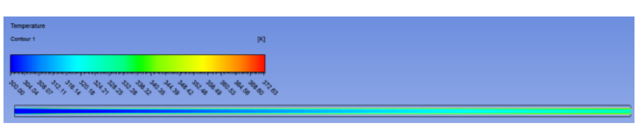


Fig. 5a. Temperature profile for Model1.

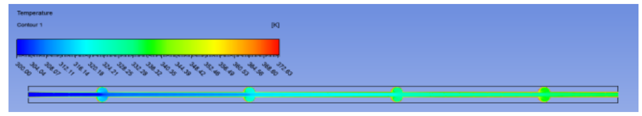


Fig. 5b. Temperature profile for Model2.

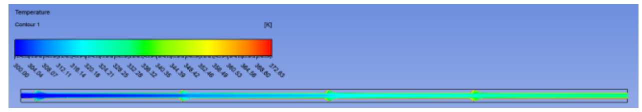


Fig. 5c. Temperature profile for Model3.

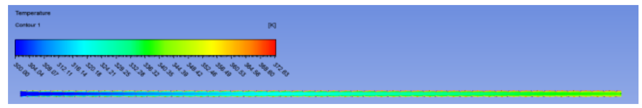
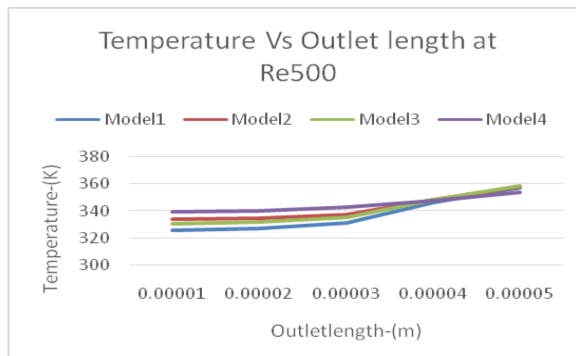


Fig. 5d. Temperature profile for Model4.



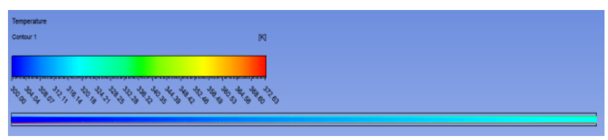


Fig. 7a. Temperature profile for Model1.

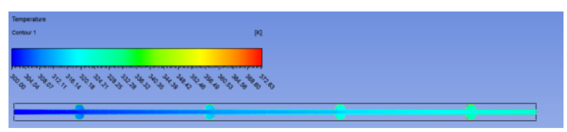


Fig. 7b. Temperature profile for Model2.

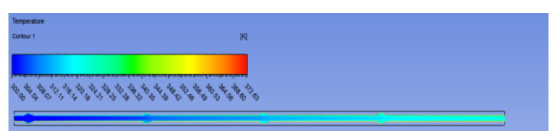


Fig. 7c. Temperature profile for Model3.

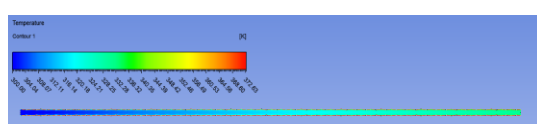


Fig. 7d. Temperature profile for Model4.

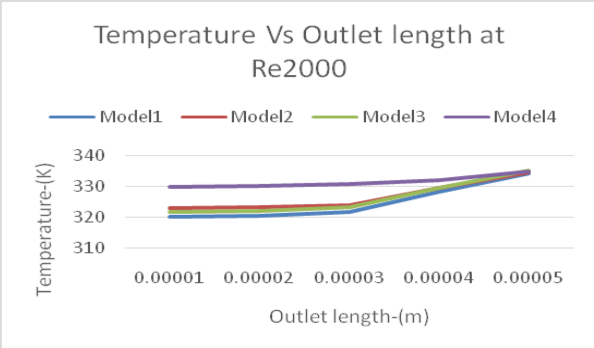


Fig. 8. Temperature vs Outlet length graph at Re 2000.

It is recognized that the model 2 and model 4 provide the best thermal performance at Reynolds numbers of 500 and lower, respectively. Higher Reynolds numbers improved the thermal performance of Model 4. Enhancing the quantity of flow disruption promotors would lead to better thermal performance. The impact of corrugated channels on the thermal performance of microchannel heat sinks was investigated in the three graphs above.

IV. CONCLUSION

Present article investigated numerically the effects of introducing flow disruption promotors in the channel surface on the heattransfer performance of microchannel heat sink. The reference geometry of the microchannel was simulated. The simulationresults were in agreement with reference geometry and literature. The temperature distribution and flow rate of the geometry were obtained. With regards to flow disruption technique, the superior heat transfer performance of channel with concave cavity in acceptable level of pressure drop is emphasized at low Reynolds number. Meanwhile expansion-contraction channel revealed higher heat transfer performance at high Reynolds number. But one of the main drawbacks of introducing flow disrupting techniques is the increase in pressure drop. Pressure drop penalty is justified by the thermal performance of the microchannel with expansion-contraction channel. Although heat transfer is superior in the concave cavity and convex rib channel but it is associated with high pressure drop. The most important characteristic within flow disturbance techniques is found in interrupted-wall channel which manifested low Pressure drop. It is established to be the most desirable feature in microchannel heat sink applications due to its role in reducing the pump power and liquid leakage risk.

REFERENCES

[1] S.B. Abubakar, N.A. Che Sidik, Numerical prediction of laminar nanofluid flow in rectangular microchannel heat sink, J. Adv. Res. Fluid Mech. Therm. Sci. 7 (2015) 29–38.

[2] M.M. Rahman, F. Gui, Experimental measurements of fluid flow and heat transfer in microchannel cooling passages in a chip substrate, in: The ASME International Electronics Packaging Conference, Binghamton, NY, USA, 1993,pp. 685–692.

[3] G. Gamrat, M. Favre-Marinet, D. Asendrych, Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular microchannels, Int. J. Heat Mass Transfer 48 (2005) 2943–2954

[4] I.A. Ghani, The significant effect of secondary flow in wavy microchannel for augmentation of heat transfer, J. Adv. Res. Fluid Mech. Therm. Sci. 25 (2016) 1–18.

[5] W. Owhaib, B. Palm, Experimental investigation of single-phase convectiveheat transfer in circularmicrochannels, Exp. Therm. Fluid Sci. 28 (2004) 105–110.

[6] J.P. McHale, S.V. Garimella, Heat transfer in trapezoidal microchannels of various aspect ratios, Int. J. Heat Mass Transfer 53 (2010) 365–375.

[7] Y. Chen, C. Zhang, M. Shi, J. Wu, Three-dimensional numerical simulation ofheat and fluid flow innoncircular microchannel heat sinks, Int. Commun. Heat Mass Transfer 36 (2009) 917–920.

[8] S.B. Abubakar, N.A. Che Sidik, A.S. Ahmad, The use of Fe3O4-H2O4 nanofluid forheat transferenhancement in rectangular microchannel heatsink, J. Adv. Res. Mater. Sci. 23 (2016) 15–24.

[9] H. Mohammed, P. Gunnasegaran, N. Shuaib, The impact of various nanofluidtypes on triangularmicrochannels heat sink cooling performance, Int. Commun. Heat Mass Transfer 38 (2011) 767–773.

[10] A.A. Razali, A. Sadikin, CFD simulation study on pressure drop and velocity across single flow microchannel heat sink, J. Adv. Res. Des. 8 (2015) 12–21.

[11] Y. Sui, C.J. Teo, P.S. Lee, Y.T. Chew, C. Shu, Fluid flow and heat transfer in wavy microchannels, Int. J. Heat Mass Transfer 53 (2010) 2760–2772.

[12] Y. Sui, P.S. Lee, C.J. Teo, An experimental study of flow friction and heattransfer in wavy microchannelswith rectangular cross section, Int. J. Therm. Sci. 50 (2011) 2473–2482.

[13] Y. Sui, C. Teo, P. Lee, Direct numerical simulation of fluid flow and heattransfer in periodic wavychannels with rectangular cross-sections, Int. J. Heat Mass Transfer 55 (2012) 73–88.

[14] H. Mohammed, P. Gunnasegaran, N. Shuaib, Numerical simulation of heattransfer enhancement inwavy microchannel heat sink, Int. Commun. Heat Mass Transfer 38 (2011) 63–68.

[15] W.M. Abed, R.D. Whalley, D.J.C. Dennis, R.J. Poole, Numerical andexperimental investigation of heattransfer and fluid flow characteristics ina micro-scale serpentine channel, Int. J. Heat Mass Transfer 88(2015) 790–802.

[16] N. Burgess, P. Ligrani, Effects of Dimple Depth on Channel Nusselt Numbers and Friction Factors, 2005.

[17] R. Kempers, J. Colenbrander, W. Tan, R. Chen, A.J. Robinson, Experimental characterization of a hybrid impinging microjet-microchannel heat sink fabricated using high-volume metal additive manufacturing, Int. J. Thermofluids 5–6 (2020), 100029.

[18] M.Ebrahim Qomi, G.A. Sheikhzadeh, A. Fattahi, Heat transfer enhancement in a microchannel using a pulsating MHD hybrid nanofluid flow, Energy Sources, A Recover. Util. Environ. Eff. 00 (00) (2020) 1–16.

[19] M.I. Hasan, H. Mohammed, G.A. Abid, Study of the axial heat conduction in parallel flow microchannel heat exchanger, J. King Saud Univ. - Eng. Sci. 26 (2) (2014) 122–131.

[20] S.H. Menon, R. Rao A, J. Mathew, J. J, Derivation of Navier–Stokes equation in rotational frame for engineering flow analysis, Int. J. Thermofluids 11 (2021), 100096.

[21] Z. Alhajaj, A.M. Bayomy, M.Z. Saghir, M.M. Rahman, Flow of nanofluid and hybrid fluid in porous channels: experimental and numerical approach, Int. J. Thermofluids 1–2 (2020).

[22] K. Egab, A. Okab, H.S. Dywan, S.K. Oudah, Enhancing a solar panel cooling system using an air heat sink with different fin configurations, in: IOP Conference Series: Materials Science and Engineering 671, 2020.

[23] F. Alnaimat, Heat transfer analysis of air-mist evaporative cooling in heat sink, Int. J. Thermofluids 14 (January) (2022), 100145.

[24] Ghiaasiaan, S. (2008). Two-Phase Flow, Boiling and Condensation. Cambridge: Cambridge University Press

[25] Husain, A. (2016). A Review on Advances in Design and Development Of Heat Exchangers. International Journal of Research and Scientific Innovation (IJRSI), Volume III(Issue XI), 60.

[26] Jafar M. Hassan, & Thamer A. Mohamed. (2014). Modeling the Uniformity of Manifold with Various Configurations. (M. S. Mansour, Dü.) Journal of Fluids, 8. doi:10.1155/2014/325259

[27] Kakaç, S., Hongtan, L., & Anchasa, P. (2012). Heat Exchangers Selection, Rating and Thermal Design. Boca Raton: CRC Press.

[28] Kandlikar, S. G., & William J. Grande. (2003). Evolution of Microchannel Flow Passages-Thermohydraulic Performance and Fabrication Technology. Heat Transfer Engineering, 24(1), 3–17. doi:10.1080/01457630390116077

[29] Kandlikar, S., Garimella, S., Li, D., Colin, S., & King, M. (2006). Heat Transfer and Fluid Flow in Minichannels and Microchannels. Waltham: Elsevier Ltd.

[30] Kim, S., Eunsoo Choi, & Young I. Cho. (1995). The Effect Of Header Shapes on the Flow Distribution in a Manifold For Electronic Packaging Applications. International Communications in Heat and Mass Transfer, 329-341