**Effects of confined jet impingement on the thermal performance of a convergent heat sink**

**Mohsen Mashhadi Keshtiban1, Mohammad Zabetian Targhi2\*, Mohamad Mahdi Heyhat3, Sajad Bigham4**

|  |  |
| --- | --- |
| 1 Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran;  | Mohsen.mashhadik@modares.ac.ir |
| 2 Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran;  | zabetian@modares.ac.ir |
| 3 Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran;  | mmheyhat@modares.ac.ir |
| 4 Faculty of Mechanical Engineering, Michigan Technological University, Michigan, United States | sbigham@mtu.edu |
|  |  |

**Abstract**

Employing electro-cooling heat sinks has been growing in the last years, and numerous research has been done to enhance these heat sinks. Jet impingement in the heat sink significantly increases the heat transfer mechanism, and its improvement on the local heat transfer coefficient at the center of the jet is notable. Turbulent jet increases heat transfer by eliminating hydraulic and thermal boundary layers and leads to higher heat dissipation. The effect of the convergent heat sink with the combination of the jet has not been investigated, and this research tries to understand its effect on heat sink performance. $k-ε Realizable$ turbulence model is used for simulations which is used in several articles. It was found that the convergent heat sink has better performance. The heat transfer coefficient decreased in the convergent heat sink slightly, while the pressure drops enhanced notably. For Re=15000, the heat transfer coefficient is decreased by 10%, and the pressure drop is enhanced by 33% for the convergent channel. By reporting heat transfer coefficient and pressure drop, it is shown that for reaching the same heat transfer coefficient, the convergent heat sink exposes smaller pressure drops in the system and shows better performance.

**Keywords:** micro-channel, heat sink, jet impingement, heat transfer, fluid flow

استفاده از چاه­های حرارتی با استفاده از سیال مایع در سالیان اخیر بسیار مورد استقبال قرار گرفته است و تلاش­های بسیاری به منظور بهبود آن­ها صورت گرفته است. استفاده از جت برخوردی در چاه­های حرارتی باعث افزایش انتقال حرارت در آن شده و ضریب انتقال حرارت محلی در ناحیه جت را افزایش می­دهد. پدیده جت برخوردی از نوع جریان مغشوش بوده و با از بین بردن لایه­های مرزی حرارتی و سیالاتی به بهبود انتقال حرارت کمک می­کند. تحقیقات زیادی به منظور مطالعه رفتار جت صورت گرفته است ولی تاکنون بررسی اثرات شیب در سطح زیرین بررسی نشده است. تغییرات شیب در سطح زیرین چاه حرارتی در میکروکانال­ها باعث تغییرات هیدرودینامیکی و انتقال حرارتی می­شود. در این مقاله از مدل توربولانسی $k-ε Realizable$ استفاده شده است که در بسیاری از مقالات پیشین کاربرد داشته است. نتایج حاصل نشان از بهبود عملکرد چاه حرارتی با افزایش شیب چاه حرارتی و همگرا کردن آن است. با همگرا کردن چاه حرارتی شعاعی شاهد کاهش انتقال حرارت در سیستم هستیم ولی این در حالی است که مقدار افت فشار به صورت محسوسی کاهش پیدا کرده است. با همگرا کردن کانال، مقدار ضریب انتقال حرارت کاهش 10 درصدی و مقدار افت فشار بهبود 33 درصدی را نشان میدهد. با بررسی مقادیر ضریب انتقال حرارت و افت فشار به صورت همزمان مشاده می­شود که همگرا کردن چاه حرارتی باعث بهبود عملکرد شده و برای رسیدن به مقادیر انتقال حرارت یکسان، مقدار افت فشار کمتری به سیستم تحمیل می­شود.

**1. Introduction**

The microelectronics sector has grown during the last several decades. This increase is the result of advancements in microelectronics manufacturing and packaging methods, as well as a strong desire to enhance microelectronics performance through device scaling [1]. Moore's law states that the segment density and performance of integrated circuits will double about every 1.5 years. However, current systems need more power and operating frequencies, resulting in enormous heat fluxes [2]. Higher temperatures (over 85 °) compromise the component operation and shorten their lifespan. Mini and microchannels are intended to address these problems and have been presented to the market using jet impingement, pin fins, and ribs [3].

Jet impingement has many industrial applications, including cooling for electronic devices and gas turbine blades[4]. Numerous studies on single-phase jet impingement have shown better heat transfer coefficients than parallel-flow heat sinks; consequently, it has become a frequent method for enhancing heat transfer [5]. The influence of geometrical characteristics, such as jet configuration, jet array layout, jet-to-target distance, nozzle diameter, nozzle design, and jet-fin combination, has been studied in several studies, including both confined and unconfined types [6].

The mechanics of the jet's heat transfer boost the rate of heat transfer at the jet's center [7]. Multiple jet inlets would help to keep the base temperature low and improve the system's overall thermal performance[8]. Using optimized jet arrays could aid heat sinks in obtaining higher performance levels and dispersing a larger quantity of heat flux. Geometric parameters control the characterization of jet fluid. For instance, jet-to-target distance and nozzle diameter may result in a lower heat transfer coefficient or a more noticeable pressure drop, dependent on the design [9], [10].

Radial convergence channels have not been investigated in previous articles, and this study tries to study the impact of jet impact on the radial convergent heat sink. The fluid flow and heat transfer characteristics of a radial convergent heat sink under jet impingement are examined. The heat transfer coefficient, pressure drop, and base temperature are reported. This study uses the $k-ε Realizable$ turbulence model for jet investigations, as it has been employed in several previous jet-related studies [11].

**2. Simulation approach**

Configuration of the heat sink with jet impingement and the radial convergent base is presented in figure (1). As can be seen, the jet inlet is at the center of the heat sink, and fluid flow inters from the inner part of a radial heat sink and exits from the outlet at the outer part of the heat sink on the top. Figure (2) shows the boundary conditions and the geometric parameters in the heat sink, and table (1) presents the value of simulation parameters in this investigation. Water and aluminum are set for fluid and solid domain, and conjugated heat transfer is considered for simulation of both cases. 22.5° slice of the heat sink and rotating periodic boundary is used for simulations to lower the high computational costs and have good mesh quality.





Figure 1. The geometry of the radial heat sink, (a) total heat sink, (b) cross-section, and the geometric parameters.



Figure 2. A 22.5° slice of the heat sink and Boundary conditions

Table 1. geometric parameter values

|  |  |
| --- | --- |
| Parameters | Value |
| Rin | 1.5 mm |
| W | 3 mm |
| Hb | 1 mm |
| Hin | 3 mm |
| Hout/Hin | 0, 0.5 |
| Rb | 15 mm |
| Re | 5000,1000,15000,2000,25000 |
| q’’ | 40 W/cm2 |

The Reynolds-Averaged Navier-Stokes (RANS) approach and the control volume-based finite-difference scheme are used in numerical simulations of thermal and turbulent flow fields. Equations in the simulation are presented in several articles[12].

For this simulation, the following assumptions are made:

• water is a Newtonian, incompressible fluid;

• The flow is in a steady state;

• Non-slip boundary conditions at the walls;

• Constant thermo-physical properties of solid and fluid;

• Negligible radiation heat transfer;

• Negligible gravitational forces, viscous dissipation, and other body forces;

• The outlet flow pressure is assumed to be atmospheric pressure.

Average heat transfer coefficient, pressure drops, and base temperature are reported for each case in five Re numbers. Re number is defined based on the jet inlet diameter in the simulations. Re number and average heat transfer coefficient are presented below:

|  |  |
| --- | --- |
|  | (1) |
|  | (2) |

Ansys fluent 2020 is employed for the simulations. The second-order differencing method is utilized for energy and momentum equations, and the Semi Implicit Method for Pressure Linked Equation (SIMPLE) technique for pressure and velocity variables coupling.

**2.1. Model Validation**

Tang et al. experimentally investigated the jet impingement on a flat plane and 45° cone in a heat sink [13]. This experimental article is considered for the model validation, and the numerical and experimental average Nu are presented in table (2). As can be seen, the numerical and experimental error is less than 3.5%, and the numerical model has good agreement with the experimental investigation.

Table 2. Results and errors for experiment [13] and numerical studies

|  |  |  |  |
| --- | --- | --- | --- |
| Geometry | Numerical Nu | Experimental Nu | Error % |
| 0 | 1219.97 | 1238 | 1.45 |
| 45 | 1303.33 | 1347 | 3.24 |

*2.2. Grid Independency*

The mesh pattern tried to accurately represent the flow structure while optimizing the number of cells and minimizing the simulation's runtime.

The realizable k- turbulence model is evaluated on four grid sizes, and as shown in table (3), the mesh independency is evaluated based on the numerical results of the average heat transfer coefficient. The greater mesh element number of 4.3 million presents errors of less than 2%, and as a result, it is chosen for all simulations.

Table 3. mesh element numbers and the related error for average heat transfer coefficient

|  |  |
| --- | --- |
| Error in average heat transfercoefficient [W/m2 K] | Number of elements |
| 21% | 1,501,684 |
| 15% | 2,231,548 |
| 1.5% | 4,321,986 |
| - | 8,854,734 |

 **3. Results and discussion**

Jet impingement heat transfer shows higher thermal performance rather than conventional micro-channel heat sinks. Here two heat sinks with different height ratios are simulated and compared with each other. The aim of this study is to understand the effect of convergence in such a heat sink on the pressure drop and heat transfer coefficient.

Figure (3) shows the heat transfer coefficient of heat sinks in five Re numbers. As it can be seen, the flat heat sink has a higher heat transfer coefficient, and by increasing Re, the heat transfer coefficient rises for both cases. This increment is higher for flat heat sink and has a much-enhanced heat transfer coefficient at higher Re numbers.



Figure 3. Heat transfer coefficient for both cases in different Re numbers

Figure (4) shows the temperature on the heat sinks based on the radial direction, and it shows that the height ratio of 0 has a lower base temperature compared to the height ratio of 0.5. Due to the larger stagnation zone in the height ratio of 0.5, the base temperature at the center of the jet is lower than in the other case. Stagnation zones for both cases are presented in figure (5), and it is clear that the flat case has a smaller stagnation zone. Also, as it is shown in figure (4), the temperature of the base does not exceed the maximum allowable temperatures for electronic devices.



Figure 4. Base temperature of the heat sink on the radial direction



Figure 5. velocity contours for the height ratio of 0 (a) and height ratio of 0.5 (b)

Figure (5) presents the pressure drop in both cases, and it shows that by increasing the height ratio, the pressure drop decreases, which is an improvement. The amount of exposed pressure is at the allowable range and can be provided by conventional pumps.



Figure 6. Pressure drop for both cases in different Re numbers.

As the heat transfer coefficient and pressure drop decrease by increasing the height ratio, another parameter is needed for choosing between them. Figure (7) presents the pressure drop and heat transfer coefficient, and as can be seen, the height ratio of 0.5 shows better performance and for reaching the same heat transfer coefficient, expose smaller pressure drops.



Figure 7. Pressure drops and heat transfer coefficient for all the cases.

**Conclusions**

This article studied the effect of a convergent channel in a radial heat sink. The findings of this investigation are as follows:

* The convergent channel has a lower heat transfer coefficient compared to the flat one.
* The flat heat sink has a higher pressure drop compared to the convergent channel.
* The base temperature at the center of the jet is lower in the Flat type due to the smaller stagnation zone.
* In the same pressure drop in both systems, convergent channels have a higher heat transfer coefficient and performance.

**List of Symbols**

|  |  |
| --- | --- |
| Din | Inlet diameter, mm |
| Rin | Inlet radius, mm  |
| W | Outlet width, mm |
| Hb | Base height, mm |
| Hin | Channel height, mm |
| Rb | Channel radius, mm |
| Tave | Average temperature at heat sink base |
| Tin | Fluid temperature at inlet |
| Re | Reynolds |
| q" | Heat flux W/cm2 |
| h | Heat transfer coefficient W/m2.K |
| $$∆P$$ | Pressure drops, Pa |

**References**

[1] L. Hussain *et al.*, "Heat Transfer Augmentation through Different Jet Impingement Techniques: A State-of-the-Art Review," *Energies*, vol. 14, no. 20, p. 6458, Oct. 2021, doi: 10.3390/en14206458.

[2] S. Chandra and O. Prakash, "Heat Transfer in Microchannel Heat Sink: Review," *Int. Conf. Recent Adv. Mech. Eng.*, no. October 2016, 2016, [Online]. Available: https://www.sciencedirect.com/science/article/pii/S0735193300001329.

[3] Y. Alihosseini, M. Zabetian Targhi, M. M. Heyhat, and N. Ghorbani, "Effect of a micro heat sink geometric design on thermo-hydraulic performance: A review," *Appl. Therm. Eng.*, vol. 170, p. 114974, Apr. 2020, doi: 10.1016/j.applthermaleng.2020.114974.

[4] F. P. Incropera and S. Ramadhyani, "Single-Phase, Liquid Jet Impingement Cooling of High-Performance Chips," *Cool. Electron. Syst.*, pp. 457–506, 1994, doi: 10.1007/978-94-011-1090-7\_21.

[5] V. S. Devahdhanush and I. Mudawar, "Critical Heat Flux of Confined Round Single Jet and Jet Array Impingement Boiling," *Int. J. Heat Mass Transf.*, vol. 169, p. 120857, Apr. 2021, doi: 10.1016/j.ijheatmasstransfer.2020.120857.

[6] B. W. Webb and C. F. Ma, "Single-Phase Liquid Jet Impingement Heat Transfer," *Adv. Heat Transf.*, vol. 26, no. C, pp. 105–217, 1995, doi: 10.1016/S0065-2717(08)70296-X.

[7] S. V. Garimella, "HEAT TRANSFER AND FLOW FIELDS IN CONFINED JET IMPINGEMENT," *Annu. Rev. Heat Transf.*, vol. 11, no. 11, pp. 413–494, 2000, doi: 10.1615/AnnualRevHeatTransfer.v11.90.

[8] V. Radmard *et al.*, "Multi-objective optimization of a chip-attached micro pin fin liquid cooling system," *Appl. Therm. Eng.*, vol. 195, no. December 2020, p. 117187, 2021, doi: 10.1016/j.applthermaleng.2021.117187.

[9] M. Froissart, P. Ziółkowski, W. Dudda, and J. Badur, "Heat exchange enhancement of jet impingement cooling with the novel humped-cone heat sink," *Case Stud. Therm. Eng.*, vol. 28, no. August, 2021, doi: 10.1016/j.csite.2021.101445.

[10] C. Nuntadusit, K. Wongcharee, and V. Chuwattanakul, "An Impingement Cooling Using Swirling Jets Induced by Helical Rod Swirl Generators," 2016, doi: 10.1515/tjj-2016-0043.

[11] P. Naphon and S. Klangchart, "Effects of outlet port positions on the jet impingement heat transfer characteristics in the mini- fi n heat sink ☆," *Int. Commun. Heat Mass Transf.*, vol. 38, no. 10, pp. 1400–1405, 2011, doi: 10.1016/j.icheatmasstransfer.2011.08.017.

[12] S. H. Seyedein, M. Hasan, and A. S. Mujumdar, "Modelling of a single confined turbulent slot jet impingement using various k - ϵ turbulence models," *Appl. Math. model.*, vol. 18, no. 10, pp. 526–537, 1994, doi: 10.1016/0307-904X(94)90138-4.

[13] Z. Tang, Q. Liu, H. Li, and X. Min, "Numerical Simulation of Heat Transfer Characteristics of Jet Impingement with a Novel Single Cone Heat Sink Highlights :," *Appl. Therm. Eng.*, 2017, doi: 10.1016/j.applthermaleng.2017.08.099.