INTERNATIONAL RESEARCH JOURNAL OF MULTIDISCIPLINARY STUDIES & SPPP's, Karmayogi Engineering College, Pandharpur Organize National Conference Special Issue March 2016

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

NUMERICAL ANALYSIS OF A HEAT SINK WITH DIMPLES

[1] S. V. Jadhav,[2] P. M. Pawar,
[1, 2] SVERI.s College of Engineering, Pandharpur

Abstract

In this paper, a numerical simulation is carried out to analyse the heat transfer performance of copper channel. COMSOLMULTIPHYSICS commercial software is used for the analysis. The material chosen for the channel is copper because of its good thermal properties. A channel with dimples on its base is chosen for the analysis. The heat transfer of the microchannel is found to increase with the use of dimples.

Introduction

The previous century brought the miniaturization of electronic components. Overheating of these devices necessitates the need of advanced cooling technologies. The main reason for using these technologies is to eliminate as fast as possible the maximum heat quantity from these systems in order to ensure an increased reliability and functional stability (Kim & Kim, 2007). We know that Using CPU's at high temperatures can cause system to crashes in the short term and in the long term cause to reduce the life of your CPU. In extreme cases the CPU could be burn out or melt onto the motherboard. The dissipation of this large amount of heat cannot be achieved by conventional cooling methods. Thus, the modern electronic component cooling needs proper technology to ensure proper performance.

In electronic systems, a heat sink is a passive heat exchanger that cools a device by dissipating heat into the surrounding medium. In computers, heat sinks are used to cool central processing units or graphics processors. Heat sinks are used with high-power semiconductor devices such as power transistors and optoelectronics such as lasers and light emitting diodes (LEDs), where the heat dissipation ability of the basic device is insufficient for its cooling.

A heat sink transfers the thermal energy from a higher temperature surface of the electronic component to a lower temperature fluid medium. The fluid medium is frequently air, but can also be water or in the case of heat exchangers, refrigerants and oil. The most common heat sink materials are aluminum alloys. Copper has excellent heat sink properties in terms of its thermal conductivity and corrosion resistance.

A heat sink is designed to maximize its surface area in contact with the cooling medium surrounding it. Fluid velocity, choice of material, protrusion design and surface treatment are factors that affect the performance of a heat sink. Heat sink attachment methods and thermal interface materials also affect the die temperature of the integrated circuit. The use of thermal adhesive or thermal grease can also improve the heat sink's performance by filling air gaps between the heat sink and the device.

In designing or selecting an appropriate heat sink that satisfies the required thermal and geometric criteria, one needs to examine various parameters that affect not only the heat-sink performance itself, but also the overall performance of the system. Option of choosing a particular type of heat sink depends largely on the thermal budget allowed for the heat sink and external conditions surrounding the heat sink. In any type of heat sink, one of the most important

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

external parameters in air cooling is the flow condition which can be classified as natural, low flow mixed, and high flow forced convection.

A list of design constraints for a heat sink may include parameters, such as

1) Induced approach flow velocity
2) Available pressure drop
3) Cross sectional geometry of incoming flow
4) Amount of required heat dissipation
5) Maximum heat sink temperature
6) Ambient fluid temperature
7) Maximum size of the heat sink
8) Orientation with respect to the gravity
9) Appearance and cost

Given a set of design constraints, one needs to determine the maximum possible performance of a heat sink within the envelope of constraints. The parameters, over which a designer has a control for optimization, typically include,

1) Fin height
2) Fin length
3) Fin thickness/spacing
4) Number/density of Fins
5) Fin shape/profile

Descriptions of projected cooling performance of some of the recent heat sink cooling technologies for CPU cooling (Pautsch, 2005) are mentioned in the Table No.1below. Some additional methods other than those mentioned in Table No. 1 proposed by (Banton & Blanchet, 2004) are as follows.

- Spray cooling with mercury inside
- Hollow core liquid cooled electronic modules
- Microchannel cooling;
- Refrigeration, air chillers;
- Heat pipes;
- Thermo-electric coolers (TEC).

Microchannel heat sink cooler is an important approach to meet the requirements of microelectronics package cooling systems. A lot of investigations about micro-channels have been undertaken by many researchers in the past years. Increasing trend of higher packaging density in the electronic industry and the high-pressure drop problem limits the performance of traditional silicon heat sink making it necessary to search for heat transfer enhancement methods. Replacing the silicon fins with other conductive metals and introducing fins of different shapes and sizes into the cooling channel to enhance the thermal exchange rate between cooling liquid and substrate is one way to overcome this problem. Channels are generally etched and covered by a suitable cover on the top which ultimately form a complete structure which is responsible for removing the heat by conjugative heat transfer method. The basic principle of micro-channel heat sink is that bottom is in contact with the heat source and fluid enters through the entrance of the heat sink to take away heat thereby cooling the substrate.

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

However even today, the concept of micro channels is still not established well enough to be considered accurately predictable. There have been many inconsistencies in the micro channel studies published so far. The behavior of heat transfer mechanisms in micro channels is generally case-specific, resulting in disagreement in the prediction of similar macro channel systems by different researchers. Thus still a lot of work needs to be done in this area.

A lot of the published work is related to introducing fins in the channel, thereby increasing the area of heat transfer. In this study instead of increasing the heat transfer area flow disturbance by introducing dimples in the channel is investigated for their performance.

Description of the model

In this work, Computational Fluid Dynamics analysis is carried out for heat transfer analysis of a microchannel with dimples on its base. Here commercial software, COMSOL MULTIPHYSICS is used for numerically solving the governing equations. Selecting elemental volume as a representative part of a physical problem and applying suitable boundary conditions to it helps in reasonably predicting the behavior of the real process. In this work, water used as a coolant flowing through a copper heat sink. Since all the channels are geometrically identical and receive the same flow rate, a single channel with fluid in it is considered as a computational domain instead of the complete heat sink in this analysis. Channels considered for the simulations are of dimensions 1000 µm square and of length of 5 mm with 100 µm thick walls and nine equidistant dimples placed axially along the axis of the channel.

It is assumed that the entire heat is transferred to the coolant through the channel base and vertical wall. Thus the heat transferred to the elemental volume is a conjugate heat transfer problem, which combines conductive heat transfer in solids and convective heat transfer in liquids. Therefore the conjugate heat transfer module of the software is used for the simulation.

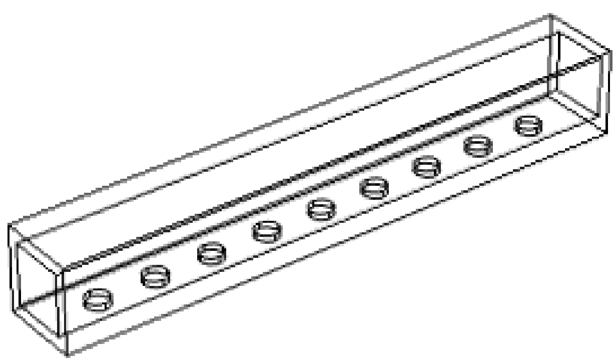


Fig. (1) Computational domain for the analysis

To simplify the analysis following assumptions are considered.
1) Flow is Steady, laminar and incompressible.
2) Water enters the heat transfer area with uniform velocity.
3) The thermo physical properties of both the fluid and the solid are temperature independent.

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

4) Heat is applied at a constant temperate to the channel substrate.
5) No radiation heat transfer.
The following governing equations are used to describe the fluid flow and heat transfer.
Continuity equation
$\nabla U = 0$ (1)
Conservation of momentum
$\rho\, U \cdot \nabla U + \Delta P - \mu \nabla\, 2\, T = 0$ (2)
Conservation of energy
$U. \nabla T = 0\ for\ solids$ (3)
$\rho Cp\ U\ .\ \nabla T - kf\, (\nabla\, 2\, T) = 0\ (for\ fluid)$ (4)

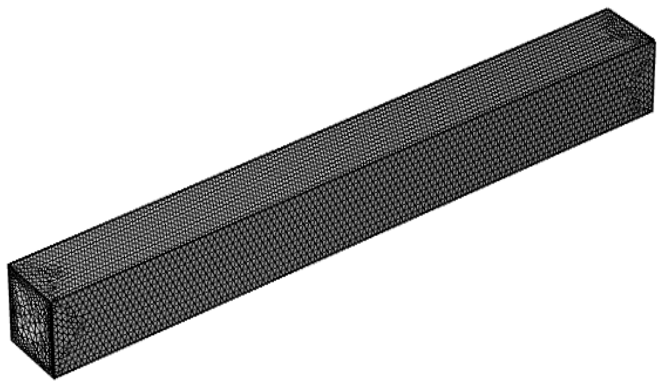


Fig. No. (2) Meshed water domain used in analysis

Boundary	**Fluid B C**	**Thermal B C**
Front inlet	Inlet	Adiabatic
Front solid	Wall	Adiabatic
Back outlet	Outlet	Adiabatic
Back solid	Wall	Adiabatic
Left	Symmetry	Symmetry
Bottom	Wall	Constant temp.
Right	Symmetry	Symmetry
Top	Wall	Adiabatic

The mesh generated for the model is formed by more than one lack tetrahedral elements and ten thousand triangular elements with a 0.13 Minimum Element Quality. The above mentioned equations are subjected to following boundary conditions. The computational domain used in the simulation is as shown in fig. (1) and the meshed model in Fig. No.(2)

1) Water enters the copper channel with a uniform inlet velocity of 1 m\s
2) Water enters the copper channel with Temperature of 273K.
3) The coolant flow is assumed to be fully developed at outlet

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

4) No slip boundary condition is considered at inner wall solid boundaries.
5) Pressure of 1atm is considered to be acting at channel outlet.

Simulation results and its analysis

Coolant with a minimum temperature enters into the channel from its one side and leaves the channel with maximum temperature. The base of the channel is maintained at a constant temperature and therefore heat is supplied to the coolant continuously from the channel base and side walls. Therefore, the fluid picks up the heat as it flows from inlet to outlet in the channel, and its temperature continuously rises from its minimum entrance value to its maximum.

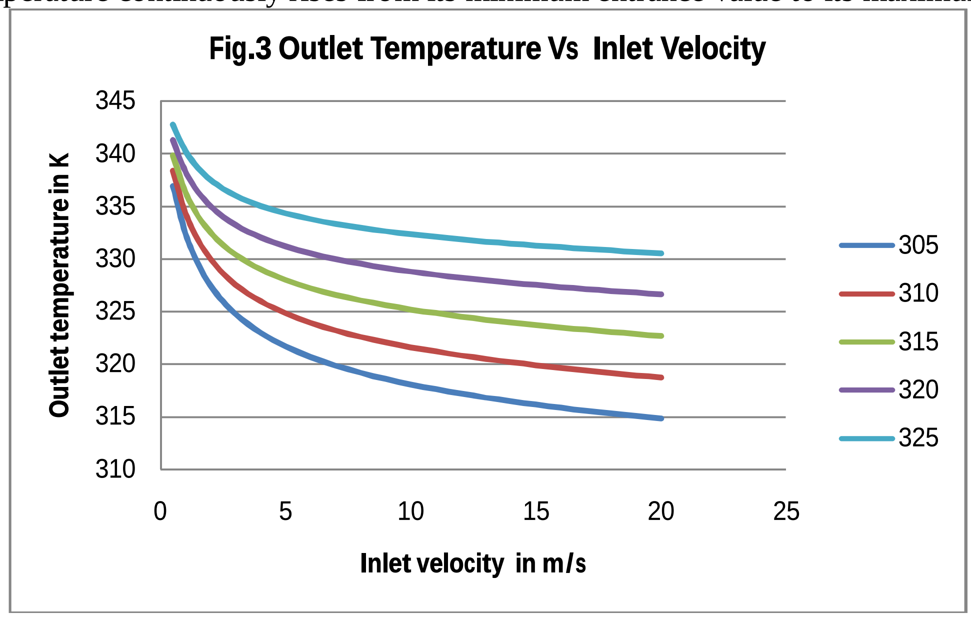


Fig. 3 shows that as the flow velocity increases, the outlet temperature of the coolant is found to be decreasing. This is due to the fact that as the velocity increases, the duration of contact between the fluid and the wall decreases. This decreases contact duration leads in decreases heat transfer and therefore decrease in temperature and obliviously the outlet temperature increases with increase in inlet temperature.

INTERNATIONAL RESEARCH JOURNAL OF MULTIDISCIPLINARY STUDIES & SPPP's, Karmayogi Engineering College, Pandharpur Organize National Conference Special Issue March 2016

Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

Fig.4 Press drop Vs Inlet Velocity

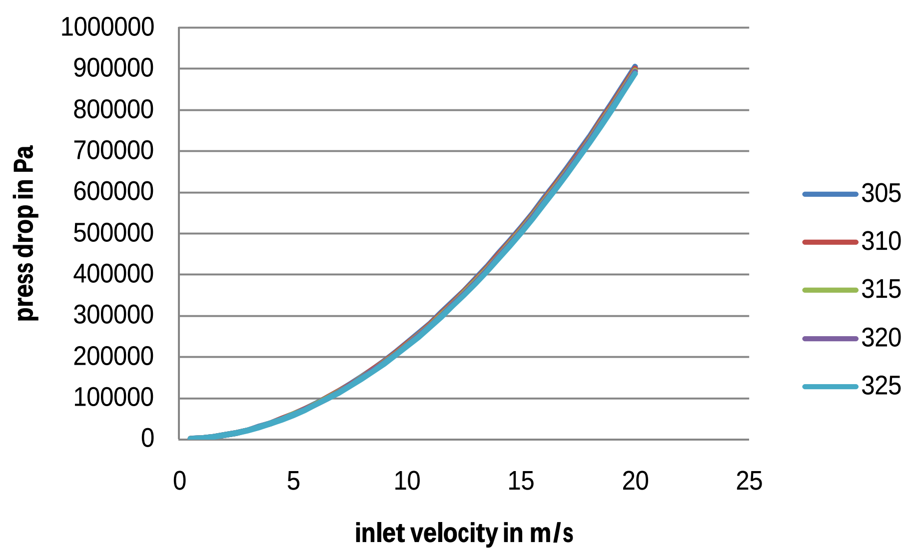
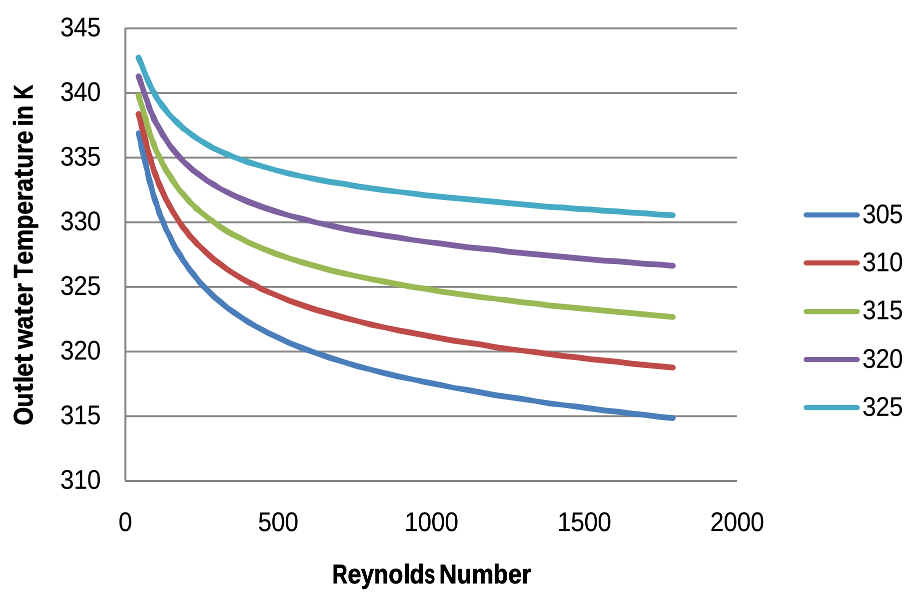


Fig.5 Water Outlet temp. Vs Reynolds Number



Pressure drop in the channel versus inlet velocity is plotted in Fig.4. Pressure drop is found to increase in the channel with increase in the velocity. Also it is observed that inlet temperature change has no effect on pressure drop. The effect of Reynolds number on the coolant outlet temperature is as shown in Fig.5 similar to increase in velocity, Reynolds number also shows decrease in temperature with increase in Reynolds number and with increase in inlet coolant temperature outlet temperature also increases.

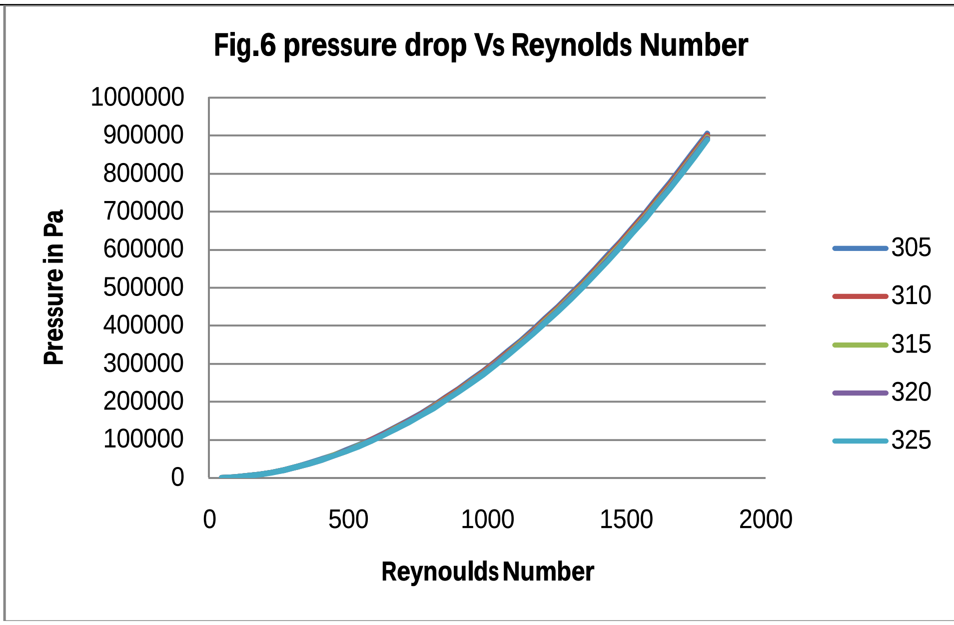
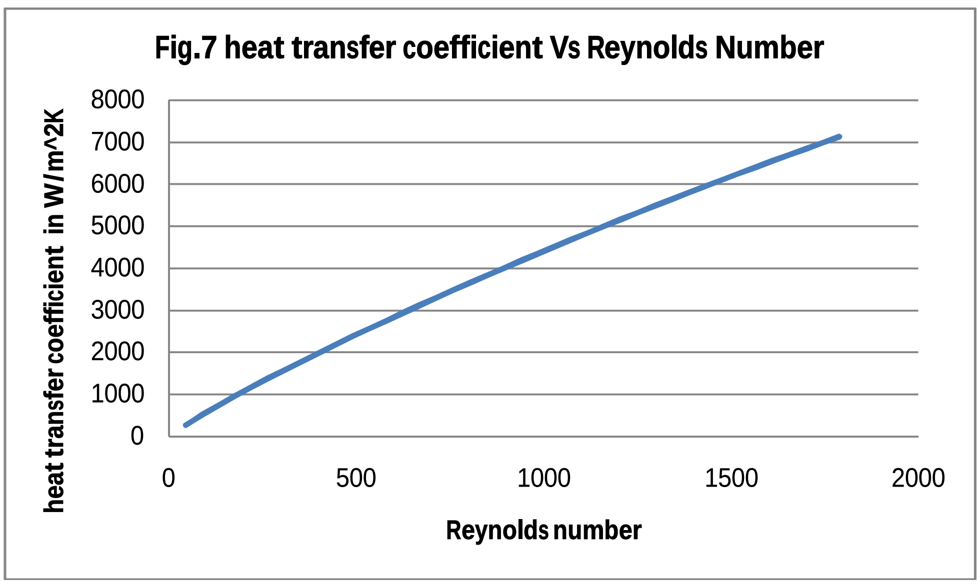


Fig. 6 shows increase in pressure drop with increase in Reynolds number. The inlet temperature does not affect the pressure. Similarly the heat transfer coefficient shows increase with increase in Reynolds Number as shown in Fig.7

Conclusions:

A copper channel with dimples on its base is chosen for this study. The equidistant nine dimples are placed along the axis of the channel. The numerical analysis shows an enhancement in the heat transfer performance with dimples. The increment in heat transfer in accompanied with a pressure drop penalty. Further it needs to analyse the optimized dimension of the dimples which will give better thermal performance with minimum pressure drop.



Vol. 2, Special Issue 1, March, 2016 | ISSN (Online): 2454-8499 | Impact Factor: 1.3599(GIF), 0.679(IIFS)

References:

[1] D. B. Tuckerman, et al.,“High performance heat sinking for VLSI,” IEEE, Electron Device Lett., 2 (5) (1981), 126–129.

[2] J. Xu et al, “Numerical simulations of interrupted and conventional microchannel heat sinks,” Int. J. Heat Mass Transfer, 51 (2008), 5906–5917.

[3] Suabsakul Gururatna,”Numerical simulation of Micro-channel heat Sink with Dimpled surfaces,” American journal of Applied Sciences, 9 (3) (2011), 399-404.