IJARES/June 2016/Volume-4/Issue-6/2420-2425 **ISSN 2347- 9337**

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH IN ENGINEERING AND SCIENCE

COMPUTATIONAL EFFORT ON HEAT TRANSFER IN MICRO-CHANNEL HEAT SINK

Dasari Uday Kumar[1], R.Srikanth[2]

[1]M.Tech Student, Dept of Mechanical, Nishita college of Engineering and Technology,
Hyderabad, T.S, India

[2]Assistant Professor, Dept of Mechanical, Nishita college of Engineering and Technology,
Hyderabad, T.S, India

ABSTRACT:

For many years, studies were conducted to understand fluid flow as well as heat transfer features in silicon-based micro channel heat sinks considered for electronic cooling applications. These non-circular channels as well as silicon based micro-channel heat sinks merge features of high material compatibility, high surface area for each unit volume ratios as well as huge potential heat transfer performance by extremely complicated fabrication process. These benefits make silicon based micro-channel heat sinks particularly attractive for extensive commercial applications. Our work focus on electronic chips cooling by forced convection of water within silicon based single micro-channel heat sinks by commercial computational fluid dynamics software FLUENT.

Keywords: Silicon based micro-channel heat sinks, Electronic chips, Commercial computational fluid dynamics, FLUENT, Electronic cooling.

1. INTRODUCTION:

In the recent times, developments concerned with Very-large-scale integration technology and Micro-electro mechanical systems demand fabrication of electronic chips on single silicon wafers for which micro channels are imbedded by means of silicon based micro systems. Thus understanding of heat as well as fluid flow phenomenon through micro-channels is the most important thrust area of electronic packaging engineers [1]. The thermal energy that is developed during relentless process of electronics chips is to be dissipated by means of incorporation of resourceful heat sinks on chips. It was observed that chip failures are caused mainly because of temperature rise within the circuits due to accumulation of heat. Thus, micro-channel embedded chips are promising solution for ultra-compact electronics gadgets. The trend in the direction of high packaging densities considerably has gained momentum with development of integrated circuit. In numerous instances trend towards higher circuit density was accompanied by means of improved power dissipation for each circuit to offer reductions within circuit delay. Thermal management plays an important role for the entire types of electronics products. Increased heat fluxes at the entire levels of packaging from chip to system pose a most important cooling challenge. Among several new methods for thermal management of high heat fluxes that are found in microelectronic devices, micro-channels are the most valuable at heat removal. The option of integrating micro-channels directly in to the substances of heat generating makes them mainly attractive. The two significant objectives within electronics cooling, minimization of maximum substrate temperature as well as reduction of substrate temperature gradients are achieved by means of micro-channels [2]. Several studies made in the recent times, have undertaken to learn the basics of micro-channel flow in addition to comparison of flow as well as heat transfer features of micro-channels by conventional channels. Studies that are made on micro-channel flows in the past years are divided in to different topics for instance temperature, heat transfer in micro-channels, Nusselt number, heat flux, studies of single phase as well as two-phase flows in micro-channels, gas flow in micro-channels, analytical studies made on micro-channel flows and designing as well as testing of micro-channel heat sinks for electronics cooling. FLUENT, is a computational program of fluid dynamics that predicts temperature distribution in micro-channel heat exchanger.

2. METHODOLOGY:

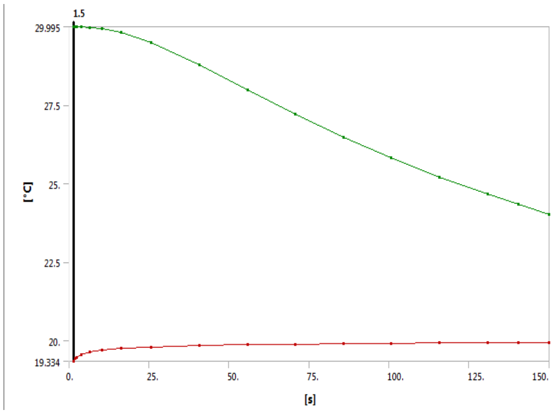
Micro-channel liquid cooling method has emerged as well-organized thermal management technique meant for microelectronics as necessity of cooling is moving beyond limit of conventional method of air cooling. The micro-heat sink modelled includes a 10 mm long silicon substrate containing a width of 57 µm as well as a depth of 180 µm has been analyzed. This model with completely developed laminar flow analyzes heat transfer within a micro-channel heat sink for various pressure drops. The numerical representation is on the basis of three dimensional conjugate heat transfers. In reality, it is tricky to attain an adiabatic boundary at inlet as well as outlet of heat sink as believed in numerical model; an important portion of heat loss is transferred towards ambient environment, particularly for low fluid flow conditions. One disadvantage of micro-channel heat ink is comparatively advanced temperature rise all along micro-channels when compared to that for conventional heat sink designs. The huge quantity of heat generated by means of semiconductor chips is performed from package by means of a comparatively small quantity of coolant, thus coolant exists at a comparatively high temperature. This objectionable high temperature gradient is significant consideration in designing of electronic cooling method. The highest temperature is found at channel outlet in low pressure drop as well as high heat flux [3]. As boundary layer thickness is minute at entrance portion, heat transfer coefficient, Nusselt number as well as heat flux is enormously high in that section but these perimeters regularly reduce all along the flow direction because of increasing boundary layer thickness. While the heat flux is supplied at channel bottom wall however side wall show additional heat than top and bottom wall because of conduction.

3. AN OVERVIEW OF PROPOSED SYTEM:

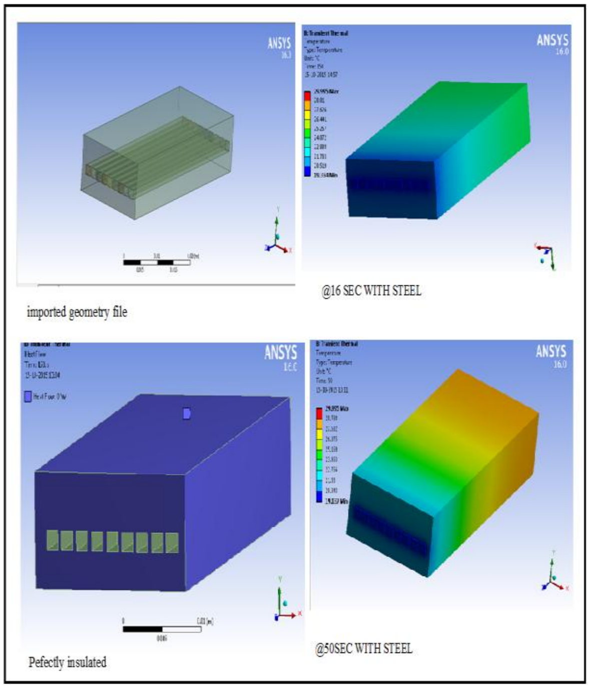
Our work address electronic chips cooling by means of forced convection of water in silicon based single micro-channel heat sinks by help of commercial CFD software FLUENT. The computational domain is discretized by means of non-uniform Meshs on flow face but consistent Mesh along flow. For single micro-channel, Mesh is generated by means of implementation of Fluent software, which is incorporated by means of Cooper method for 3-dimensional Mesh generation. The pressure, velocity as well as temperature contours at inlet and outlet are presented all along with difference of these fields in flow direction intended for visual comparisons. The output of single microchannel is in fine agreement with obtainable results for silicon substrate. The Nusselt number variation all along the flow direction is presented and compared in support of three various flow rates. The convection heat transfer co-efficient is moreover provided for three cases. Semi-Implicit Method for Pressure Linked Equations by second order upwind scheme is put into practice in support of laminar fully-developed flow. The continuity, momentum as well as energy equations are solved within a segregated manner due to its accuracy. As the fluid flows within a micro-channel, laminar flow is considered throughout simulation all along with energy equation. The single micro-channel solution is united rapidly by means of minimum number of iterations. The post processing output results are carried out by means of Excel. Three different pressure drops were considered throughout simulation [4]. As pressure difference is directly associated towards pumping power, optimization of power consumption is entertained by means of this simulation. The maximum pressure difference is considered for additional heat flux condition imposed on boundary and moreover to suppress temperature rise in sink. It is quite applicable to suppress temperature rise of water within microchannel above boiling point of water. The post processing results concerning single micro-channel is in fine agreement with the obtainable results.

4. RESULTS:

A model of three-dimensional is developed to examine flow as well as conjugate heat transfer in micro-channel based heat sink meant for electronic packaging applications. A sequence of numerical calculations were conducted by means of FLUENT and the results are presented to explain effects of temperature distribution, heat flux distribution in addition to average heat transfer coefficient in micro-channel heat sinks. The numerous Mesh generation method implemented in FLUENT is moreover put into practice for 2-D and 3-D domain. Both pre as well as post processing results of FLUENT and FLUENT are utilized effectively during simulation of present situation. The results are moreover in good agreement with obtainable results.



TITANIUM GRAPH

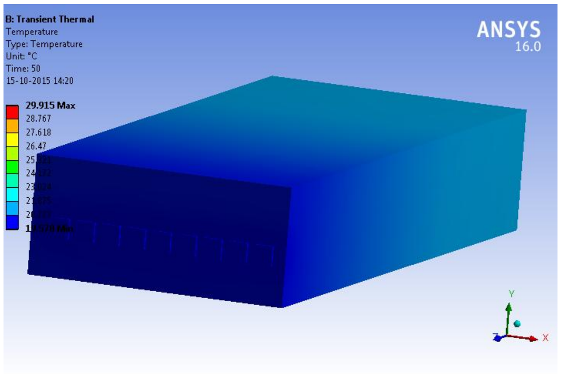


imported geometry file

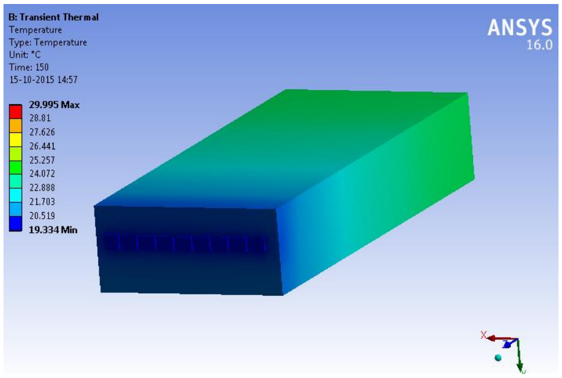
@16 SEC WITH STEEL

Pefectly insulated

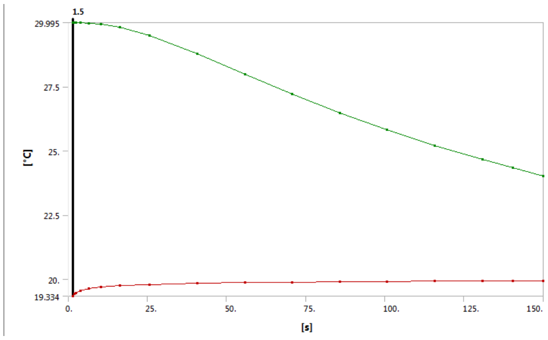
@50SEC WITH STEEL



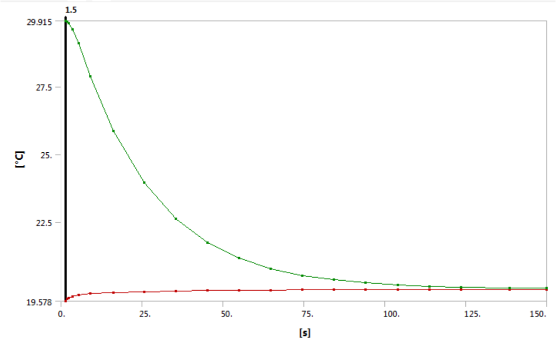
Aluminium @53 sec



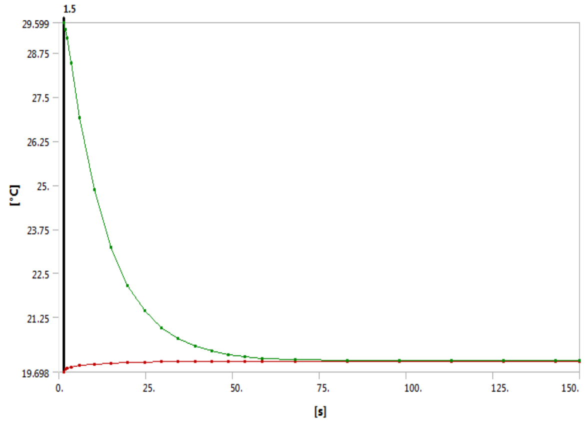
Titanium at 150sec



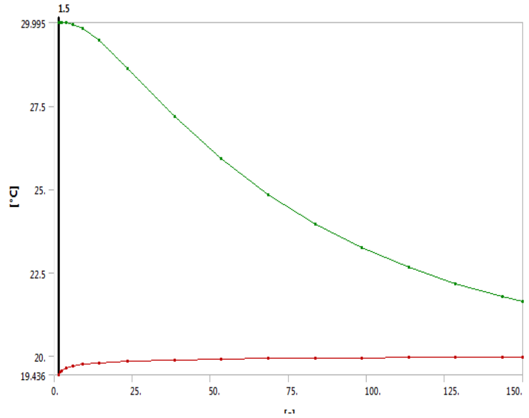
Titanium graph



Aluminium graph



COPPER GRAPH



Structural steel graph

5. CONCLUSION:

The analysis performed, provides basic understanding of combined flow as well as conjugate convection-conduction heat transfer in three-dimensional micro-channel heat sink. The results of analysis in addition to conclusions are considered as relatively general and applicable towards any three-dimensional conjugate heat transfer difficulty. The present analysis strongly specifies that forced convection water cooled micro-channel heat sink contains a better potential for application within thermal management of electronic packages. From study of single micro-channel heat fluid flow analysis, it was found that sometimes temperature of fluid is extremely high. The positioning of micro-channel for highest heat pumping capability on sink material is moreover an interest of study.

REFERENCES

[1] Y. Wang and K. Vafai, ―An experimental investigation of the thermal performance of an asymmetrical flat plate heat pipe, Int. J. Heat Mass Transfer‖, 43 (2000) 2657-2668

[2] D. Kim et al, ―Thermal optimization of micro channel heat sink with pin fin structures‖, Proceedings of the IMECE'03, IMECE2003- 42180 (2003).

[3] R.Mohan and Dr.P.Govindarajan. ―Thermal analysis of CPU with composite pin fin heat sinks ‖,International Journal of Engineering Science and Technology ,Vol. 2(9), 2010, 4051-4062.

[4] Konstantinos-Stefanos P. NIKAS* and Andreas D.Panagiotou.―Numerical Investigation of Conjugate Heat Transfer in a Computer Chassis‖ Columbia International Publishing Journal of Advanced Mechanical Engineering (2013) 1: 40-57 doi:10.7726/jame.2013.1004.