A FLOW NETWORK ANALYSIS OF A LIQUID COOLING SYSTEM THAT INCORPORATES MICROCHANNEL HEAT SINKS

Amir Radmehr and Suhas V. Patankar
Innovative Research, Inc.
3025 Harbor Lane North, Suite 300
Plymouth, MN 55447
Phone: (763) 519-0105
Fax: (763) 519-0239
Email: radmehr@inres.com

ABSTRACT

The objective of this study is to show the applicability of Flow Network Modeling (FNM) in analyzing liquid cooling systems that incorporate microchannel heat sinks. The study is divided in two parts. In the first part, an analytical model of a microchannel heat sink is proposed and its validity is established by comparison with a detailed CFD analysis. In the second part of the study, a liquid cooled system that accommodates three microchannel heat sinks is analyzed using Flow Network Modeling (FNM). In the FNM method, the pressure drop and heat transfer coefficient of the components in the system are calculated using analytical or empirical correlations. The goal of the analysis is to balance the liquid flow passing through each heat sink such that the chips case temperature remains below recommended values. This study illustrates the use of the FNM technique for analyzing liquid cooled systems and microchannel heat sinks.

KEY WORDS: Microchannel, Heat sink, Liquid cooling, Flow Network Modeling, FNM, CFD, Thermal design

NOMENCLATURE

A_f	Fin base area
B_w	Heat sink base height
C_p	Fluid thermal capacity
C_w	Channel width
D_h	Hydraulic diameter
F_w	Fin width
F_h	Fin height
f	Friction factor
H	Height of the heat sink
HSAR1	Heat sink with aspect ratio of one
HSAR1	Heat sink with aspect ratio of two
h	Heat transfer coefficient
k_s	Silicon thermal conductivity
L	Length of the heat sink
$\dot{m}$	Mass flow rate
P	Pressure
Pr	Prandtl number
q”	Heat flux
q	Heat flow
Re_D	Reynolds number based on D_h
$\overline{T}_s$	Channel wall average temperature
$\overline{T}_b$	Heat sink average base temperature
W	Width of the heat sink
V	Average velocity

Greek symbols

ρ	Water density
η	Fin efficiency

INTRODUCTION

The heat density of the new-generation chips is predicted to exceed 100 W/cm^2 in the near future [1]. As a result, it will be very difficult for the traditional air cooled heat sinks to keep the junction temperature below acceptable values. Another concern is the relatively large size of the air cooled heat sinks which defies the market demand for more compact electronics devices. Microchannel liquid cooled heat sinks are attractive alternatives for the traditional air cooled heat sinks [2]. The flow of coolant in these heat sinks is usually laminar due to the small hydraulic diameter of the microchannels [2, 3]. Also, the flow inside the microchannels is hydrodynamically and thermally fully developed [3]. Therefore, the flow and thermal analysis of individual microchannel heat sinks is rather simple. However, the heat sinks are usually parts of a cooling system where the flow in the rest of the system may be turbulent. The combination of laminar and turbulent flow makes the thermal analysis of the entire system complex.

This study is divided into two parts. First, an analytical model of a microchannel heat sink is constructed and its validity is verified by a detailed calculation using Computational Fluid Dynamics (CFD). In the second part of the paper, a typical liquid cooled system utilizing microchannel heat sinks is analyzed. The goal of the study is to balance the coolant flow through the heat sinks based on their heat loads. For this part of the study, the technique of Flow Network Modeling (FNM) is used. FNM is a valuable technique for quick prediction of system-wide flow and temperature distribution in thermal design of electronics equipments. Ellison [4] presented a traditional use of flow network analysis in the design of electronics cooling systems. A generalization of this technique has been proposed by Belady et al. [5]. This paper presents an application of FNM to a liquid cooling system.

THE PHYSICAL SYSTEM

In this study, microchannel heat sinks with two different aspect ratios of the channel are used. We call these heat sinks HSAR1and HSAR2, with aspect ratio of 1 and 2 respectively. The schematic of the heat sinks under consideration and their dimensions are shown in figures 1 and 2. The heat flux is supplied at the bottom of the heat sink. It is assumed that the top surface of the heat sink is not participating in the heat transfer. In other words, the entire heat flow is transferred to the coolant through the channel base and vertical walls.

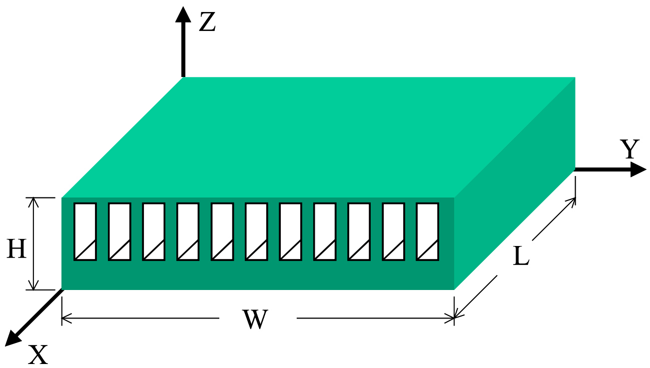

Figure 1. The schematic and overall dimensions of the microchannel heat sink (The drawing is not to scale)

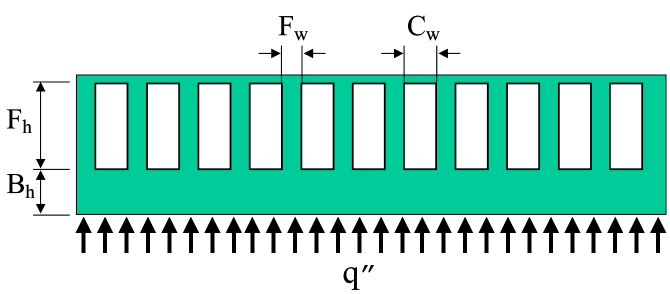

Figure 2. Dimensions of the microchannel heat sink (The drawing is not to scale)

Table 1 shows the geometrical quantities for the two heat sinks. The heat flux is supplied to the entire bottom area of the heat sink which is 4 cm^2. The total number of channels is 100 and the total length of the channels is 20 mm. Since it is assumed that the total heat will be transfer to the coolant from the base and the vertical walls of the channel, the thickness of the top section of the heat sink is irrelevant. The heat sink material is silicon with thermal conductivity of 150 W/mK. The cooling liquid is water and the water inlet temperature is 20 degree C. A uniform heat flux of 400,000 W/m^2 is applied at the bottom of the heat sink, which generates a total heat flow of 160 W. The analysis is done for various mass flow rates of water ranging from 5 to 30 milligram/s. Consequently, the inlet velocity of the water varies from 0.4 to 2 m/s for HSAR1 and 0.1 to 1 m/s for HSAR2. For all cases the channel Reynolds number is small (below 240) and the flow inside microchannels is laminar.

Table 1. Geometrical quantities for HSAR1 and HSAR2

	HSAR1	HSAR2
W (cm)	2	2
L (cm)	2	2
H (μm)	350	350
C_w (μm)	120	120
F_w (μm)	80	80
B_h (μm)	200	200
F_h (μm)	120	240
No. of Channels	100	100

THE CFD APPROACH

The CFD calculation reported here is conducted using a commercial general purpose CFD software package COMPACT™ [6], which solves the three-dimensional form of mass, momentum, and energy equations. The problem under consideration is a conjugate heat transfer problem which involves heat transfer in both solid and liquid. The governing equations are discretized on a Cartesian grid using the finite-volume method described by Patankar [7].

The CFD calculation of the entire heat sink with 100 channels is very intensive and time consuming. Since all the channels are geometrically identical and receive the same flow rate and heat flux the calculation domain can be restricted to only one channel. Note that the end effects can be ignored since the heat transfer from the side walls of the heat sink only affects a few channels in each side of the heat sink. Also, due to the symmetry in the flow and heat transfer across a channel the computational domain can be limited to half of a channel and half of the fin adjacent to the channel. Figure 3 shows the computational domain and the boundary conditions. Table 2 summarizes the boundary conditions.

The number of control volumes used for all calculations presented in this paper is 100 × 20 × 40 in the x, y, and z directions respectively. To assure that the results obtained are grid independent the number of control volumes was increased to 200 × 40 × 80. The changes in the overall pressure drop across the heat sink and the average heat sink base temperature were less than one percent. Therefore, the results presented here are considered to be grid independent.

Table 2. Flow and thermal boundary conditions

Boundary	**Flow B.C.**	**Thermal B.C.**
Front inlet	Inlet	Adiabatic
Front Solid	Wall	Adiabatic
Back Outlet	Outlet	Adiabatic
Back Solid	Wall	Adiabatic
Left	Symmetry	Symmetry
Right	Symmetry	Symmetry
Bottom	Wall	Const. Heat Flux
Top	Wall	Adiabatic

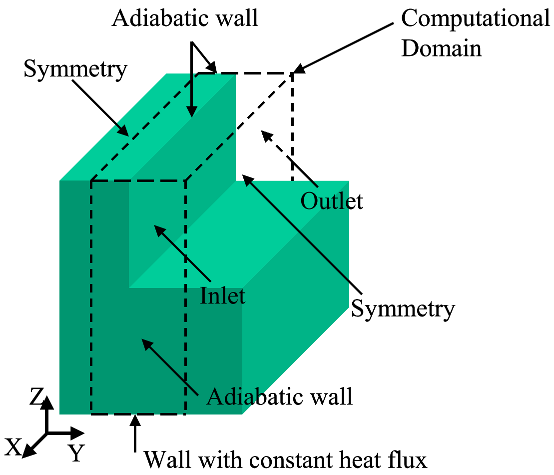

Figure 3. The computational domain and boundary conditions

THE ANALYTICAL APPROACH

The Reynolds number of flow inside the microchannels is less than 240 and the flow is laminar. The flow is assumed to be hydrodynamically and thermally fully developed throughout the channel. The hydrodynamic and thermal entry lengths can be calculated from the following expressions:

$$\frac{x_{fd,h}}{D_h} \approx 0.05\,\mathrm{Re}_D \tag{1}$$

$$\frac{x_{fd,t}}{D_h} \approx 0.05\,\mathrm{Re}_D\,\mathrm{Pr} \tag{2}$$

where D_h is the hydraulic diameter, Re_D is the Reynolds number based on the hydraulic diameter, and Pr is the Prandtl number. The hydrodynamic entry length for the microchannels under consideration is very small and can be neglected. However, the neglect of the thermal entry length may introduce some error especially at the large flow rates. This will later be examined with reference to the CFD results.

The pressure drop across channels can be obtained from Darcy equation:

$$\Delta P = f\,\frac{L}{D_h}\,\frac{\rho V^2}{2} \tag{3}$$

where f is the friction factor, L is the length of the channel, ρ is density, and V is the average fluid velocity. The value of f can be obtained analytically by solving the momentum equation inside the channel. For rectangular channels with aspect ratio of one and two the value of fRe_D is 57 and 62 respectively [8].

For the heat transfer analysis, the flow inside a rectangular channel with constant heat flux is considered. Figure 4 shows the calculation domain for one of the microchannels under consideration. The bottom of the channel receives a uniform heat flux. Two vertical planes pass the middle of the channel walls and are adiabatic. The top of the channel is adiabatic too. The goal of the analysis is to calculate the average wall temperature of the heated surface at the bottom of the heat sink (the heat sink base temperature). Heat spreads in the solid through conduction and enters to the water flow through convection from the inner walls of the channel.

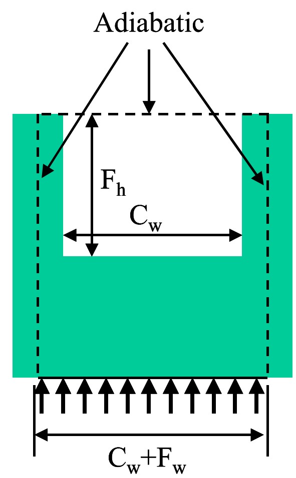

Figure 4. Calculation domain for one microchannel

Due to the high thermal conductivity of the silicon it can be assumed that the temperature of the inner wall of the channel is uniform at each longitudinal cross section (the CFD results presented in the next part support this assumption). The problem can be considered as a classical conduction problem in a fin as shown in figure 5.

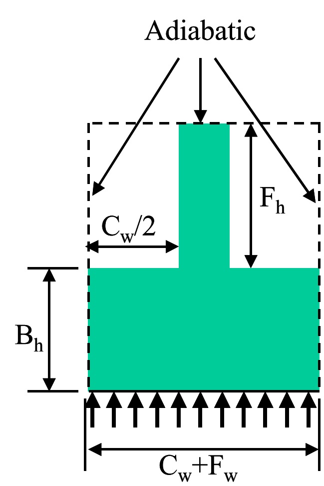

Figure 5. Another calculation domain

Then, the effective heat transfer perimeter at the inner wall of the channel is:

$$HT_{Perimeter} = C_w + 2\eta F_h \tag{4}$$

where η is the fin efficiency and can be calculated as follows:

$$\eta = \frac{\tanh(mF_h)}{mF_h} \quad (5)$$

$$m = \sqrt{\frac{2hL}{k_s A_f}} \quad (6)$$

h is the heat transfer coefficient, $2L$ is the fin heated perimeter which is twice the length of the channel, k_s is silicon thermal conductivity, and A_f is the fin base area. The following relationship exists between the heat flux at the external wall of the channel and the heat flux at the internal walls of the channel:

$$q''_{ext.}(C_w + F_w) = q''_{\text{int.}}(C_w + 2\eta F_h) \quad (7)$$

Therefore, the heat flux at the channel internal wall is:

$$q''_{\text{int.}} = \frac{q''_{ext.}(C_w + F_w)}{C_w + 2\eta F_h} \quad (8)$$

The channel internal surface temperature at any cross section along the channel (in the x direction) can be calculated from the energy balance across the channel as follows:

$$T_s(x) = Ti + \frac{q''_{ext.}(C_w + F_w)}{\dot{m}C_p} x + \frac{q''_{\text{int.}}}{h} \quad (9)$$

Ti is the water inlet temperature, $\dot{m}$ is the water mass flow rate. The second term in the above equation is the rise in the average water temperature at location x. The third term is the difference between the water and the channel internal surface temperature at location x. The average temperature of the internal wall of the channel can be obtained by integrating the above equation along the length on the channel.

$$\overline{T}_s = Ti + \frac{q''_{ext.}(C_w + F_w)}{2\dot{m}C_p} L + \frac{q''_{\text{int.}}}{h} \quad (10)$$

The average base temperature of the heat sink can be calculated using a one dimensional conduction equation between the base of the heat sink and the internal surface of the channel.

$$\overline{T}_b = \overline{T}_s + q''_{ext.} \frac{B_h}{k_s} \quad (11)$$

The heat transfer coefficient is obtained analytically for the fully developed laminar flow inside rectangular channels. The Nusselt number for the laminar flow inside rectangular channels with constant heat flux is 3.61 and 4.12 for channel aspect ratio of one and two respectively.

RESULTS

The CFD solution includes the calculation of the pressure and velocity fields in the rectangular microchannels. The calculated values of fRe_D agree with the analytical values of 57 and 62 (for the two aspect ratio) to four significant figures. The more important outcome of the CFD calculation is the temperature distribution. Figure 6 and 7 show the temperature distribution inside the microchannel and silicon substrate for the HSAR1 and the water mass flow rate of 5 milligram/s. The temperature distribution for other cases is similar to the one shown below.

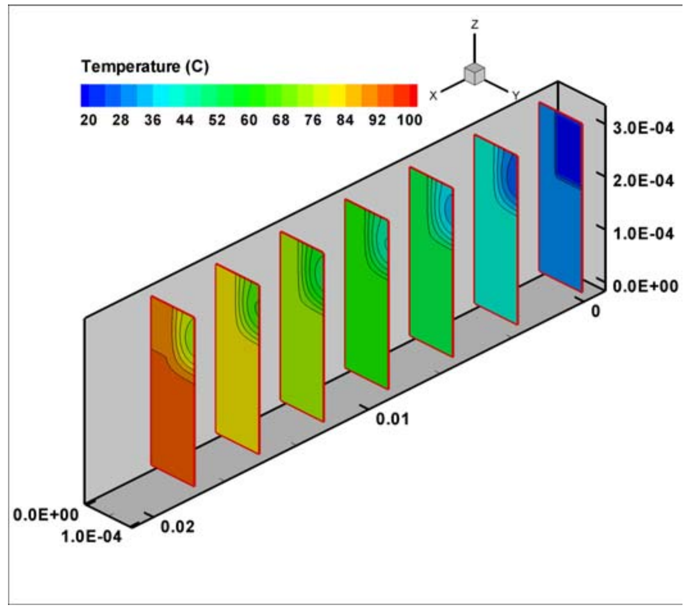

Figure 6. Temperature distribution inside HSAR1 at yz planes.

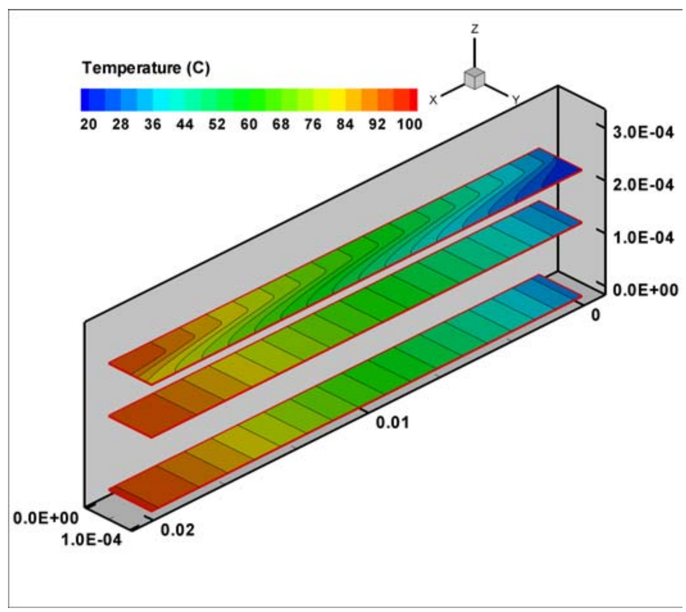

Figure 7. Temperature distribution inside HSAR1 at xy planes

The solid and fluid temperature increases along the x axis due to the heating. At each cross section of the channel the solid stays almost at the same temperature while the isotherms in the fluid are very close. The temperature change in the solid from base of the heat sink to the inner walls of the channel is less than 0.5 degree C. Also, it can be seen that the channel inner wall temperature is uniform at each cross section which supports the assumption made for the analytical solution at the previous section.

The results for the average heat sink base temperature are shown in figures 8 and 9. The base temperature changes inversely with flow rate as equation (10) suggests and reaches an asymptote at high mass flow rates. Here, the agreement between the results obtained from the two approaches is very good. The discrepancy between the results at the higher mass flow rates is due to neglecting the thermally entry length in the analytical approach. At the highest flow rate, this discrepancy can be seen to be about 10%.

Figures 8 and 9 also reveal the effect of the different aspect ratios of the two heat sinks. The base temperature of HSAR2 is about 3 degree C below the one for HSAR1. HSAR2 provides more heat transfer area which results in lower wall temperature although the Nusselt number is higher for HSAR1. In other words, the internal heat flux shown in equation (8) is smaller for HSAR2. Therefore, HSAR2 is more effective in cooling the chips.

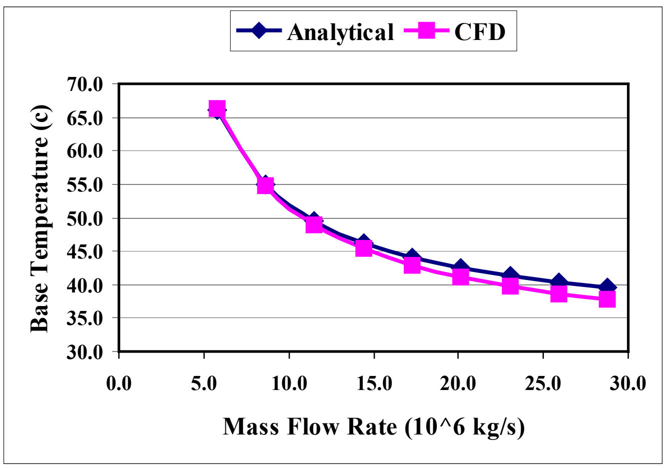

Figure 8. Average base temperature for HSAR1

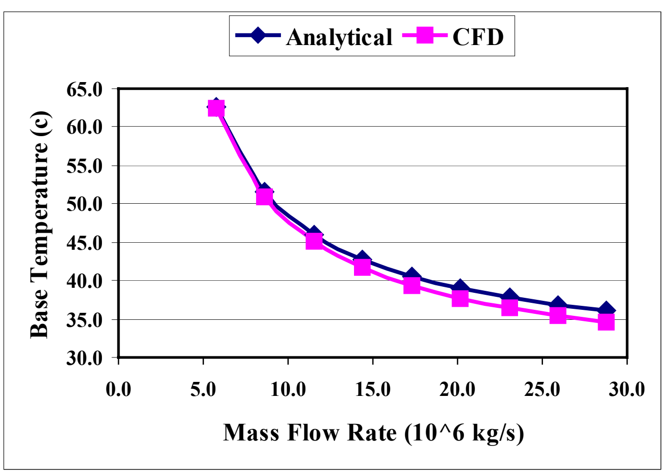

Figure 9. Average base temperature for HSAR2

The thermal resistance of a heat sink is defined as the ratio of difference between average base temperature and inlet fluid temperature to the total heat flow.

$$R_t = \frac{\overline{T}_b - T_i}{q} \qquad (12)$$

Figures 10 and 11 show the comparison between the thermal resistance calculated by CFD and analytical approaches for HSAR1 and HSAR2 respectively.

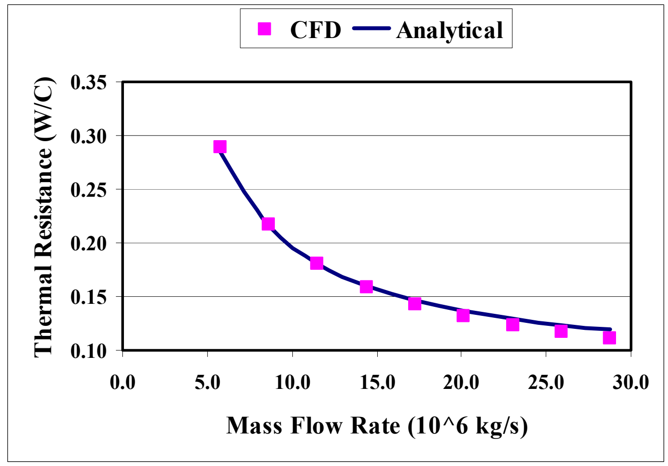

Figure 10. Thermal resistance for HSAR1

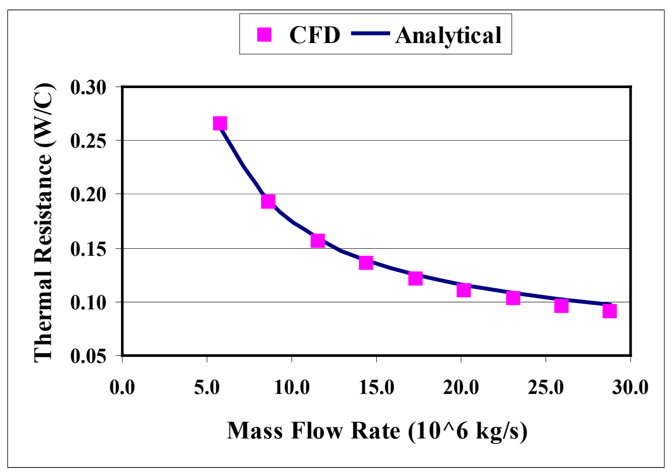

Figure 11. Thermal resistance for HSAR2

A LIQUID COOLED SYSTEM UTILIZING MICROCHANNEL HEAT SINKS

In this section a typical thermal design problem for a liquid cooled system is presented. The essential components in a liquid cooled system are pump, heat sink or cold plate, heat exchanger, and piping. The Pump circulates the coolant (water in this case) through the system. The heat sink or cold plate removes heat from the high heat load chips. The heat exchanger transfers the heat out of the system, and the piping system provides connections between the components.

Figure 12 shows the physical system which utilizes three HSAR1. The flow and thermal characteristics of the HSAR1 have been established through the analytical approach described earlier. The heat sinks are arranged in a parallel manner through a cooling manifold. The heat loads for heat sinks 1, 2, and 3 are 70, 120, and 200 Watts respectively. The average case temperature of the chips which is the same as the base temperature of the heat sinks should stay below 40 degree C to ensure proper performance of the chips. Four main system level design issues can be identified for this problem.

1) Sizing the pipes to minimize the pressure drop in the system
2) Selecting the pump to provide the needed total flow rate
3) Selecting the heat exchanger to maintain the proper coolant temperature
4) Balancing the flow through heat sinks to ensure the proper cooling of the chips

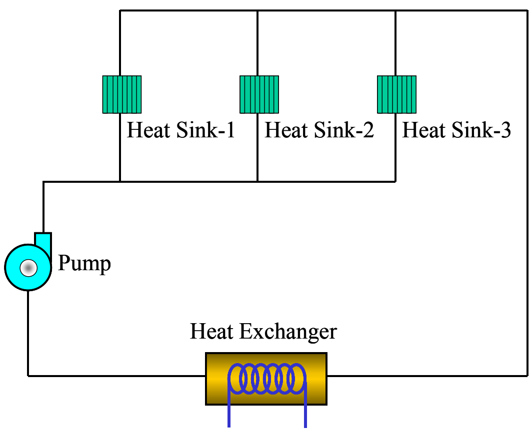

Figure 12. A liquid cooled system

The combination of available solutions for each issue creates many design choices. A fast and efficient technique is needed to find the best design from the possible options within the design time constraint. The technique of Flow Network Modeling (FNM) explained in the next section is the technique of choice since it can evaluate many design options accurately in a very short time.

Analyzing all the design options is beyond the scope of this paper. The goal of this study is limited to balance the coolant flow through heat sinks to keep the heat sinks base temperature below 40 degree C. An effective heat exchanger is selected which maintains the water exit temperature (the inlet temperature to the heat sinks) at about 20 degree C. The diameter of the piping is 3 mm throughout the system.

THE FNM TECHNIQUE

A brief description of the FNM technique is provided in this section for completeness. More details of this methodology are described at [4] and [5].

FNM is a generalized methodology involving representation of a flow system as a network of components and flow paths for the purpose of predicting system-wide distribution of flow rates and temperatures. Practical electronics cooling systems can be represented as a network of components such as pipes, ducts, heat sinks, screens, filters, cold plates, pumps, heat exchanger, etc. Each component in the flow network is represented by empirical correlations that relate pressure drop and heat transfer rate to the corresponding flow rate. The flow and thermal performance of the system is predicted by imposition of the conservation of mass, momentum, and energy over the flow network. The set of discrete equations for the network is solved to calculate velocity, pressure, temperature, and the related quantities throughout the system.

Overall flow characteristics of standard components such as pipes, ducts, screens, and bends can be obtained from handbooks by Idelchik [9] and Blevins [10]. For nonstandard components supplier data, CFD analysis, or test data can be used to get the flow characteristics. Empirical correlations are also used for heat transfer coefficients needed in determining the heat losses or surface temperatures. Because of the use of overall component characteristics, FNM-based analysis is very quick in terms of model definition and computational time. Further, use of empirical characteristics assures that predictions of the system performance obtained from FNM analysis are accurate over wide range of operating conditions. The strength of FNM is its ability to analyze system-wide interaction of the individual components in a rapid and accurate manner.

In the present study, a commercially available program MacroFlow [11] that incorporates the FNM technique is used for analysis of the flow and temperature distribution within the system.

THE FNM MODEL FOR THE SYSTEM

The liquid cooled system is represented as a network of components through which water flows. The flow and thermal characteristics of standard components such as pipes, bends, and Tee junctions, are available from handbooks. The characteristics of the microchannel heat sinks are determined by the analytical model presented earlier in this paper. The characteristics of components such as pump and heat exchanger are available through the manufacturers of those components. FNM considers the interaction of the various flow impedances and the pump characteristics curve to determine the total flow rate and the flow rates through individual components. Also, temperature distribution in the system is calculated by solving a set of algebraic equations which represents the energy conservation for individual components.

Figure 13 shows the flow network model of the system. The pump circulates the water through the closed loop. Water enters the inlet side of the manifold, splits through the three heat sinks, and leaves from the exit side of the manifold. This type of manifold is called an S-shape manifold since it looks like the letter S. Water picks up heat from the heat sinks and leaves the manifold at a higher temperature. The water then enters the heat exchanger, transfers the heat to the ambient, leaves heat exchanger at a lower temperature, and enters the pump.

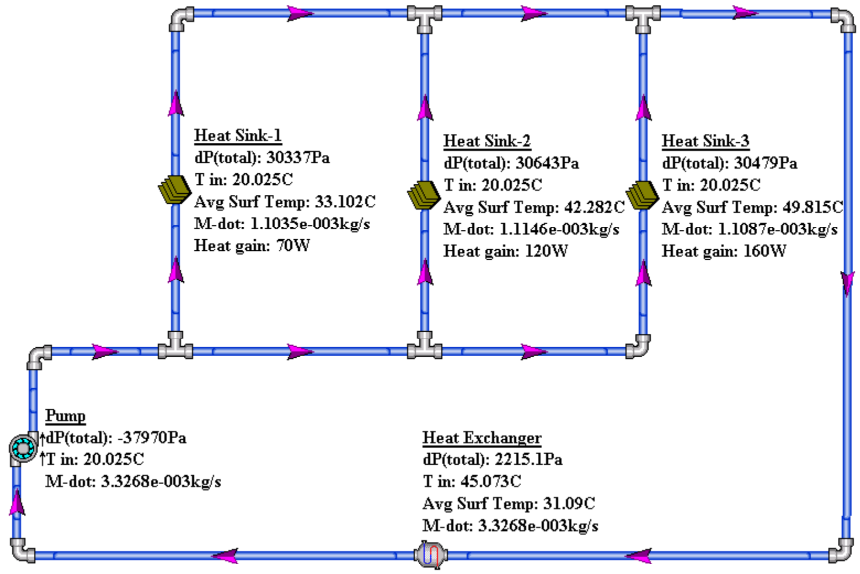

Figure 13. FNM model of the original liquid cooled system

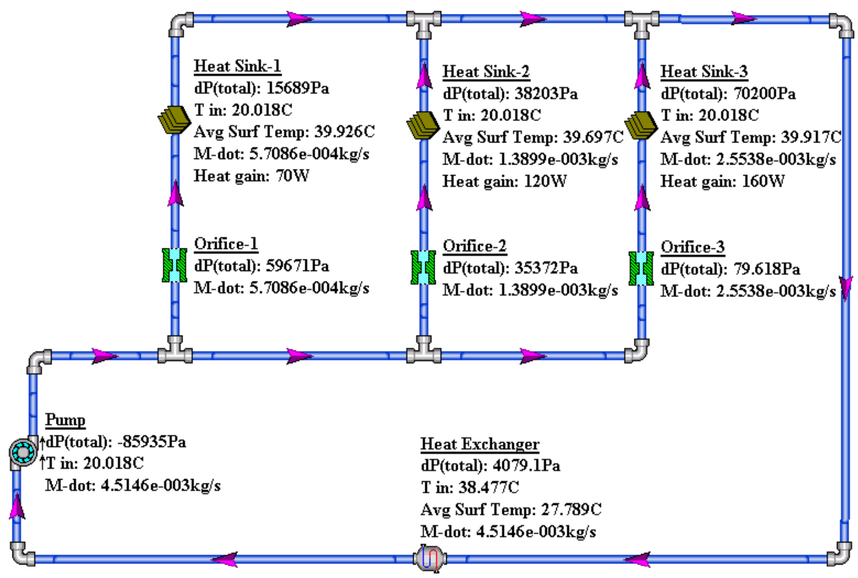

Figure 14. FNM model of the modified liquid cooled system

BALANCING THE FLOW THROUGH THE HEAT SINKS

The original design of the system was modeled using FNM technique to calculate the flow and temperature distribution in the system. The calculated, flow rate, pressure drop, water temperature, and heat sink base temperature are shown in figure 13. The total flow rate through the system is 3.3×10^{3} kg/s. The three heat sinks receive almost the same flow rate which is one third of the total flow rate. The small differences in the heat sinks flow rates are due to the losses in the manifold. The base temperature of heat sink-1 is 33.1 degree C which is below the 40 C design requirement. However, the base temperatures for heat sink-2 and heat sink-3 are 42.3 C and 49.8 C respectively. To maintain the base temperature of all the heat sinks below the design criteria, the flow rates should be balanced based on the chips heat load. An effective way to balance the flow rate is to use micro orifices in each branch of the cooling manifold and adjust their openings to balance the flow rate. This is a trial and error procedure and needs many trials to achieve the desire flow distribution. Adding the orifices will increase the total resistance of the system. As a result, a more powerful pump is needed to circulate the water in the system.

Figure 14 shows the modified design of the cooling system. The flow through the heat sinks is very well balanced. The base temperature of all heat sinks is almost the same and less than 40 C. That means the flow rate through each heat sink is proportional to the heat generated at the chips. The orifice in series with heat sink-3 is completely open and creates very little pressure drop. The orifice in series with heat sink-2 is partially open and creates a pressure drop comparable to the heat sink-2 pressure drop. The orifice in series with heat sink-1 is very restricted and creates a pressure drop about four times the pressure drop in the heat sink-1. Note that the total pressure drop in each branch of the manifold is the same since they are parallel branches and have the same inlet and exit pressures. However, each branch has a different K-factor (loss factor). Branch-1 has a high K-factor and takes small amount of flow rate to produce the overall branch pressure drop. On the other hand, branch-3 has the smallest K-factor and takes the maximum flow rate (more than four times of the flow rate in branch-1) to create the same overall pressure drop. Compared to the original design, the total flow rate through the system has increased to 4.5×10^{3} kg/s since a more powerful pump is added to the system. Although it has not been shown here, using a more powerful pump without the orifices cannot bring the base temperature of heat sink-3 below 40 C due to the flow maldistribution in the manifold. Figure 15 and 16 show the mass flow rate and base temperature of the three heat sinks for the original and modified designs.

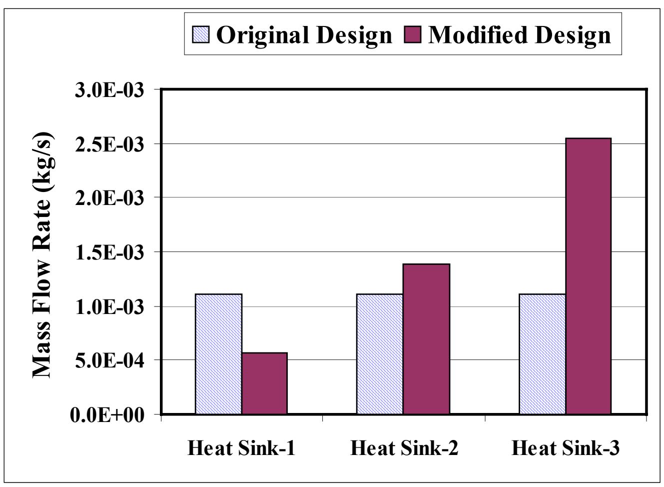

Figure 15. Mass flow rate through heat sinks

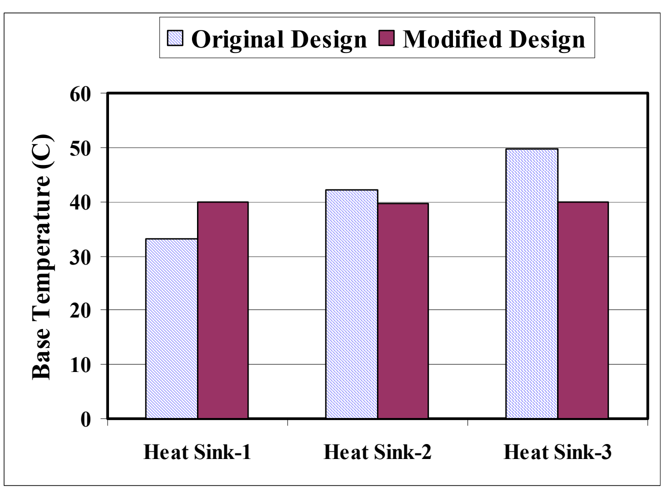

Figure 16. Heat sinks base temperature

CONCLUSION

An analytical model of microchannel heat sink is described. The problem is also solved by using the CFD technique. It is shown that the flow and heat transfer characteristics given by the analytical model agree very well with the CFD results.

In the second part of the paper, a typical liquid cooled system with three microchannel heat sinks has been analyzed using FNM technique. The goal of the analysis is to balance the flow through the heat sinks based on their heat loads. Orifices with different openings are used to balance the flow in the heat sinks. The modified design meets the design requirements. The FNM technique provides an effective way of obtaining system-wide results for flow and heat transfer.

References

[1] A. Bar-Cohen, "Thermal management of microelectronics in the 21st century," *Proceeding of the IEEE/CPMT Electronic Packaging Technology Conference*, 1997, pp. 29-33.

[2] A.G. Fedorov, R. Viskanta, "Three-dimensional conjugate heat transfer in the microchannel heat sinks for electronic packaging," *Int. J. Heat Mass Transfer*, No. 43, 2000, pp. 399-415.

[3] S.K. Lim, K.T.Ooi, T.N. Wong, K.C. Toh, K. Suzuki, "Experimental investigation of liquid flow in micro-tube," *Engineering advances at the dawn of the 21st century*, Proceeding of the Seminar on Integrated Engineering, 2000, pp. 487-493.

[4] G.N. Ellison, *Thermal Computations for Electronic Equipment*, Robert E. Krieger Publishing Co., Malibar, Florida, 1989.

[5] C. Belady, K.M. Kelkar, and S.V. Patankar, 1999, "Improving Productivity in Electronic Packaging with Flow Network Modeling (FNM)," *Electronics Cooling*, Vol. 5, No. 1, pp. 36-40.

[6] Compact User's Manual, Innovative Research, Inc., 3025 Harbor Lane N., Suite 300, Plymouth, MN 55447.

[7] S.V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere, 1980.

[8] F.P. Incropera., and D.P. DeWitt, *Introduction to Heat Transfer*, John Wiley & Sons, 1996.

[9] I.E. Idelchik, *Handbook of Hydraulic Resistance*, CRC Press, Florida, 1994.

[10] R.D. Blevins, *Fluid Dynamics Handbook*, Krieger Publishing Company, 1992.

[11] MacroFlow Users Manual, Innovative Research, Inc., Plymouth, MN 55447.