Journal of Thermal Engineering
Yildiz Technical University Press, Istanbul, Turkey
Vol. 2, Special Issue 4, No. 4, pp. 826-836, July, 2016.

http://eds.yildiz.edu.tr/journal-of-thermal-engineering/Articles
Manuscript Received July 23, 2015; Accepted September 18, 2015

This paper was recommended for publication in revised form by Regional Editor Kwok-wing Chau

THERMAL DESIGN METHODOLOGY AND PREDICTION OF HEAT SINK PERFORMANCE

*Dongmei Zhou
Department of Mechanical Engineering
California State University, Sacramento
Sacramento, California, USA

Timothy Rau
Department of Mechanical Engineering
California State University, Sacramento
Sacramento, California, USA

Keywords: Heat sink; Natural convection performance; Thermal design methodology; Hand calculation; CFD simulation; Mock-up testing
** Corresponding author: Phone: +1 916 278 7504*
E-mail address: zhoud@ecs.csus.edu

ABSTRACT

The ability to identify heat sources and predict their temperatures across a variety of operating conditions is the key in the design of a reliable electronic system such as a computer, a server, or any other system containing a printed circuit board assembly. The goal of this paper is to investigate a heat sink to come up with a framework to correlate hand calculations and numerical simulations with experimentally obtained results. The thermal design methodology is discussed and a natural convection study of a heat sink is used to demonstrate this thermal design process. Multiple fin heights of a heat sink at different ambient temperatures are analyzed by hand calculations, numerical simulations using ANSYS Icepak 14.0, and a mock-up testing using a copper slug with a Minco heater attached to simulate a processor on a printed circuit board assembly. These three methods can help check upon each other's accuracy and credibility. Results from the experiment demonstrate how one can improve the accuracy of results by using known correlated data to future investigations. This allows for improved optimization studies and helps reduce design cycle time.

INTRODUCTION

When electric current goes through a resistance heat is generated. The failure rate of electronic equipment increases rapidly with temperature rise in electronic equipment components. Therefore, thermal control on temperature management has become increasingly important in the design and operation of electronic equipment [1]. Thermal management strategies ultimately determine the cost, reliability, and performance of electronic equipment and system. The fundamentals of thermal management stem from basic heat transfer modes (conduction, convection and radiation) and phase change. For electronic equipment at low heat flux (<100W/cm^2), air cooling is widely used. In this method, heat is removed either by natural convection over a finned heat sink or by forced convection which incorporates the use of fans. A heat sink with embedded heat pipes or using looped Thermosiphon can greatly improve thermal performance when compared to a typical aluminum or copper base heat sink [2, 3]. For high flux applications, liquid cooling is considered [4, 5]. Liquids with high specific heats and thermal conductivities can be used as the coolant to remove more heat than air cooling. Liquid cooling systems can be classified as direct cooling and indirect cooling systems. In direct cooling systems such as immersing cooling, the electronic equipment is cooled directly by being immersed into a dielectric liquid with boiling. In indirect cooling systems, a liquid-cooled heat exchanger such as a cold plate is placed in the heat source to extract heat and then dissipate to other places. For more compact and smaller electronic equipment with extremely high heat flux, advanced cooling solutions have been developed and it includes jet impingement, spray cooling, thermoelectric cooling, refrigeration cycle cooled system, and mini-channel and microchannel heat sinks [6, 7]. A number of researchers have studied microchannel heat sinks for chip cooling applications [8-11]. This method has shown the capacity and capability to achieve very high level of cooling performance (> 790W/cm^2).

The task of thermal design is providing an effective path to safely remove this internally generated heat from electronic components to the surroundings. The purpose of this paper is to investigate the heat sink with fins and develop a methodology in thermal design for the chip on the printed circuit board assemblies (PCA's). The methodology representing an end user experience intends to lead to a reliable operating system for electronic equipment.

The research begins with the system definition for a better cooling solution and design. The system definition covers the major components that typically need thermal design attention although the focus is on the cooling of the chip. It also covers the environmental factors that affect the ability to cool the components. Fans and other methods of cooling is described along with some preferred design practices on how to structure or layout one's system or sub-system. After an understanding of the system definition is gained, methods for analysis and performance prediction are discussed, including important hand calculations, numerical simulations using the computational fluid dynamics (CFD) approach, and a mock-up testing. Finally, an example problem is performed by following the methodology established. The example problem is a heat sink investigation that is defined, designed, tested, and analyzed. Comparing results from three different methods allows the methodology to be evaluated for accuracy. Results and assumptions that return less than desired results can be modified for improvement. It is hoped that the methodology described in the paper is beneficial to the applications to the larger scaled projects as well as to the investigations with different cooling technologies.

THERMAL SYSTEM DEFINITION

Heat Sources

The first step in selection and design of a cooling system begins by identifying the major power consumers that need thermal attention as well as the amount of heat generated which determines what kind of cooling methods is going to be chosen. Typical power-consuming components are central processing unit (CPU), power converters, logic controllers, memory, storage, and input/output (I/O) cards. The importance of maintaining thermal limits of these components is to allow for system reliability and flawless performance. Since these power consumers are semi-conductors and silicon-based, they suffer from leakage current at higher temperatures, causing more power to be consumed than originally intended.

The central processing unit (CPU) is typically the hottest component on a printed circuit board assembly (PCA) and thus it needs the most attention to maintain its operating requirements. The central processing unit (CPU) is mainly made of transistors and it may contain capacitors as well. As the transistor is turned on, current flows through the collector junction and the power in the form of heat is dissipated, while the charging and discharging of capacitance of the chip through capacitors consume power to generate heat. The CPU itself tends to have an ideal working temperature limiting to a maximum junction temperature. Excess heat, if not removed, remains on the chip, and thus an increase in junction temperature can cause electro-migration or oxide to break down, leading to crashes and CPU failure.

To overcome overheating, the central processing unit (CPU) needs to be equipped with cooling systems to help regulate temperature within the unit. There are several methods to keep the processor cool. Air-cooling, heat sink and/or a combination of heat sink and airflow are currently the prevalent methods of cooling the CPU assembly and other heat source components. Air-cooling normally incorporates the use of fans that pump air to effectively carry the heat away from the device. The purpose to use a finned heat sink is to extend the surface of the material that is in contact with air such that heat is quickly removed from the device. Figure 1 illustrates the path of heat flow through a device with a heat sink, where, T_j is the maximum junction temperature of the device, T_c is the case temperature of the device, T_s is the heat sink temperature, and T_a is the ambient air temperature. Maximum junction temperature T_j values that are allowable can range from about 115°C in typical microelectronics applications to as low as 75°C for some special applications. Since the heat sink temperature T_s depends on the location of measurement, it usually represents the maximum temperature of a heat sink at the location closest to the device. For the purpose of selecting a heat sink, the heat sink should have lowest thermal resistance, low pressure drop and maximum surface heat transfer coefficient. The heat sink thermal performance is determined by the air velocity, fin design, and the choice of fin material.

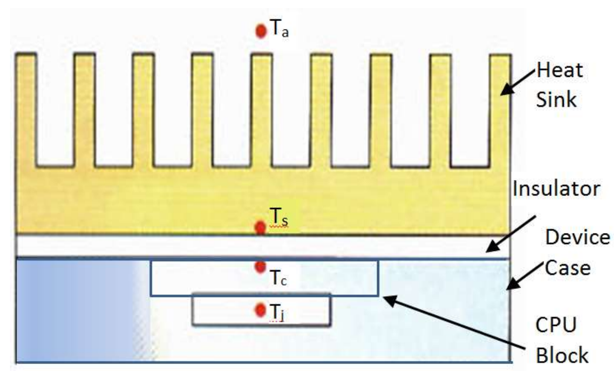

Figure 1 Illustration of path of heat flow

Thermal Environment

The second step in selection and design of a cooling system is to identify the environment conditions that the system will be exposed to. Some of the important factors to understand are the ambient temperatures that the system will function in, altitudes that it will operate at, and relative humidity or wet-bulb conditions that it will be exposed to. The range of these environmental conditions defines the design in terms of robustness or what engineering design decisions need to be made to achieve them.

The ambient temperature range affects how the electronic equipment is cooled. If the system is to function in a high ambient environment, there may not be a large temperature difference between the ambient temperature and the maximum temperature of a component. Different cooling technologies rather than air-cooling may need to be explored. However, a low ambient temperature could also cause concern. Some components may need to be insulated to maintain the design intent.

Altitude becomes important to thermal design if air is the cooling fluid. Essentially, air becomes less dense as altitude increases and thus its convective capability is reduced. The electronics will therefore experience greater component temperature rise at higher altitudes even when the operation is at constant power outputs. To account for this change in air density, extra margin needs to be designed into the cooling system or the product needs to have operating specifications stating where and how it will be operated. Since designing margin can be expensive, many server specifications state an inlet ambient temperature maximum that adjusts with altitude. For instance, for every 304.8m (1000 ft.) above the sea level that the server operates at, the maximum inlet ambient temperature is reduced by 1ºC. A second correction formula, as shown in Eq. (1), uses multipliers to correct the temperature. The multiplier value depends on the altitude and how the component is cooled and they are stated in Table 1. This formula is not as conservative as the first correction method but it is a simple correction calculation. The second formula is used in the paper as follows:

$$T(z) - T_a = \left(T_{SL} - T_{SL,a}\right) * Multiplier \qquad (1)$$

where, z is the altitude (ft.), $(T(z) - T_a)$ (°C) is the surface or air temperature minus the ambient temperature at altitude, z, $(T_{SL} - T_{SL,a})$ (°C) is the surface or air temperature minus ambient temperature at sea level, and Multiplier (no unit) is the multiplier that is determined from Table 1.

Table 1 Altitude multiplier at different cooling methods [12]

Altitude	Multiplier		
feet	Fan-Cooled (General)	Fan-Cooled (High Power)	Naturally Cooled
0	1.00	1.00	1.00
5000	1.20	1.16	1.10
10,000	1.45	1.35	1.21
15,000	1.77	1.58	1.33
20,000	2.18	1.86	1.48

Humidity does not have a major effect on thermal performance because the changes in air thermal conductivity with humidity at different temperatures are small [13]. However, if pressure changes within the system create condensation, it needs to pay attention to the effect of humidity. This is because condensation has the potential to accumulate within a system and it may cause the system to fail. It is important to specify an operating region recommended by ASHRAE to prevent this from occurring [14].

Cooling Methods

The cooling methods applied to electronic equipment vary widely and its selection depends on heat amount generated, environmental conditions, reliability requirements for particular application, and cost. For low-cost electronic equipment, it is common to use inexpensive cooling method: heat sink combined with natural or forced convection with air as the cooling medium. Heat sinks come in a variety of shapes and sizes [15-19]. They are typically made of high conductivity material to dissipate heat more efficiently. As system constraints make the cooling solution more difficult to achieve, more complicated or expensive heat sinks are used and continue to be developed. Some examples of these technologies are heat pipes, liquid cooling, phase change cooling, heat exchangers, and thermoelectric coolers. Most of these technologies described above take advantage of liquid cooling in one way or another. A thermoelectric cooler, however, works on the principle of the Peltier effect that creates a temperature difference when a voltage is applied to the free ends of two dissimilar materials. Some of these advanced cooling solutions have cost challenges when one is trying to implement them. The heat sink that meets system and reliability requirements for the least cost usually is the chosen solution.

System Layout

In order to maintain power-consuming components at intended temperatures one must understand the system layout as well, i.e., how the power-consuming components are positioned as the air flow passes over. System layout is a key factor in thermal system design because it helps determine potential difficulties in cooling some components. An example of this can be described with the help of Figure 2. This figure displays a flow network. It can be seen that air enters the enclosure and passes the storage (a bank of hard drives) and then it splits across the central processing unit (CPU) and memory. The air from the memory leaves the enclosure, but the air from the CPU's continues across another logic controller and then the input/output (I/O) cards before leaving the enclosure. Even though the system may operate with a cool inlet air temperature, the air flow will be heated by storage, CPU's, and a logic controller before it ever begins to cool the I/O cards. This preheated air will not be able to cool the I/O cards to an acceptable temperature. This is why it is important to understand the airflow path for a system and how the cooling of one component could affect each other.

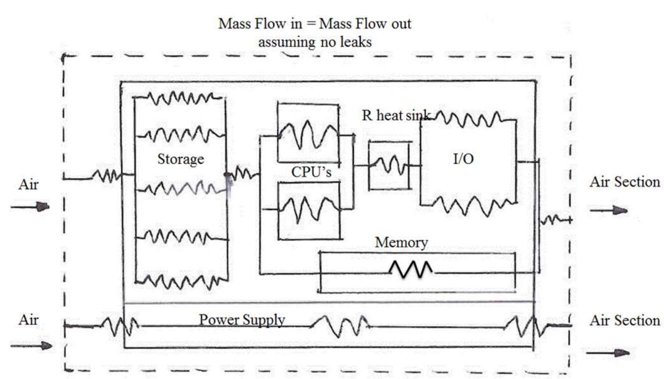

Figure 2 Example of a first order flow network

THERMAL ANALYSIS APPROACH

Thermal analysis for the printed circuit board can be accomplished by using initial hand calculations, computational fluid dynamics, the testing of a mock-up system and/or combined all three. Each approach has its own advantage in terms of ease and speed to collect the data and when they should be performed in the design cycle.

The first approach is initial hand calculation which is an early analysis for quick simple results. The primary use of these calculations is for one-dimensional analysis of thermal and flow networks. The formulas used in this approach are often simplified. Assumptions for these problems usually are uniform heat sources, constant flow, and uniform heat spreading. The purpose of these calculations is for early feasibility studies in a very simple and quick way even though it may limit the accuracy of the results. In spite of simplicity, hand calculation can be accurate within 10-20% of the experimental results.

Computational fluid dynamics (CFD) is a numerical simulation approach used for three-dimensional analysis of components or systems. Since these calculations are performed by a computer, partial differential equations based on the conservation of momentum, mass, and energy for fluid mechanics and heat transfer are calculated. Depending on the complexity of the model being analyzed, numerical simulations may take minutes to days. The advantage of using numerical simulation is its speed in the calculations. It also allows one to visualize temperature and velocity gradients within the component. If the boundary conditions are applied correctly with a good mesh, the CFD simulations can provide accurate results. However, the computational results are only as good as the models created in the CFD simulations, which still contains assumptions for simplification. Many of the assumptions made are coming from the limitations of the software, for instance, the software cannot handle complex geometries. Although the software is getting better to be able to handle complex geometries these days, a simple numerical model with a low grid count for saving calculation time can still lead to inaccurate numerical results. Therefore, it is important to validate the theoretical and computational results using empirical and/or experimental data.

The testing of a mock-up is the analysis that should be performed whenever it is allowed since it helps achieve a successful final system design. A mock-up is a simplified construction of a system. It can be constructed by using salvaged predecessor components, heaters, simplified sheet-metal, and even cardboard. The purpose of mock-up testing is to get preliminary real world results to validate hand calculations and CFD simulations before spending money and time to get prototypes made. It also allows for quicker real world feasibility studies to avoid potential costly fixes and revisions on prototype.

EXAMPLE PROBLEM STATEMENT

A demonstration of the design methodology is implemented in the testing of a heat sink. An extruded aluminum crosscut heat sink with a fin height of 23mm, made by Delta Electronics, is chosen for this study. The heat sink dimension and sizes are shown in Table 2 with illustration in Figure 3. The heat sink is analyzed with hand calculations and CFD simulations. A mock-up is then built and tested to validate hand calculation and CFD simulation. Heat sink is tested first at an ambient temperature of 10ºC for three different power levels of 5, 10, and 15 Watts, respectively. Three potential use orientations of flat, vertical, and horizontal, as seen in Figure 3, is further tested to investigate the effects of gravity and orientation dependency. To investigate the effect of altitude, the heat sink is tested at a simulated elevation of sea level, 5,000, 10,000, and 15000 feet, respectively, by using a Russells Technical Products RHD-64 altitude chamber. All of these tests are accomplished with no airflow so that the natural convection performance of the heat sink is checked. Humidity is not monitored in the testing.

Results of the initial mock-up testing are used to validate hand calculation and CFD simulation with heat sink at different power levels, different orientations, and different altitudes. Initial assumptions and predictions made in the hand calculation and CFD simulation are evaluated and adjusted for subsequent iteration calculation. Improved assumptions are further used to predict two other heat sinks with changes in the fin height from 23mm to 19.75mm and 16.45mm, respectively. All three heat sinks with different heights (23mm, 19.75mm, and 16.45mm) are then tested at a higher ambient temperature of 30ºC rather than 10ºC subject to different power levels of 5, 10, and 15 Watts. These tests are performed at the same sea level. The purpose is to check how well one can get accurate results by using known correlated data for future investigations.

Table 2 Dimensions of heat sink with a fin height of 23mm

Heat Sink dimensions in mm					
Row	Column	Short wall	Long wall	Fin height	Area mm^2
6	8	1.5mm	3.75mm	23mm	11590

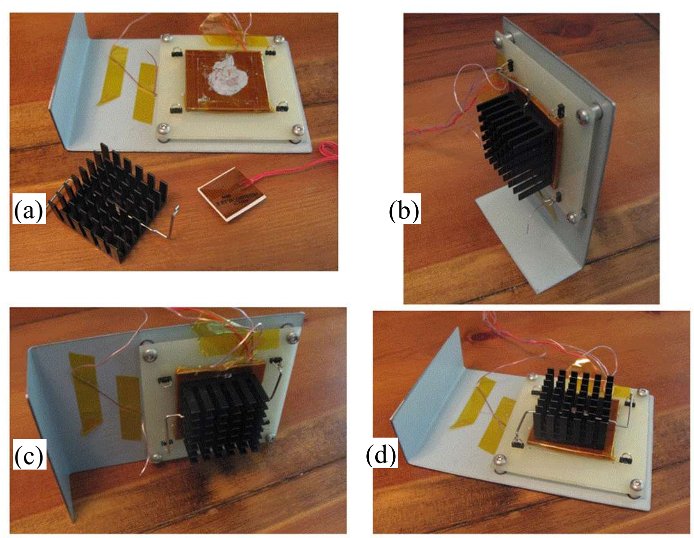

Figure 3 Illustration of orientation: (a) original; (b) vertical orientation; (c) horizontal orientation; (d) flat orientation

RESULTS AND DISCUSSION

Initial Hand Calculations

By assuming uniform heat source, constant flow, and uniform heat spreading, the initial calculation used for the heat sink is power formula for convection, the equation (2),

$$Q = hA(T_s - T_a) \tag{2}$$

This formula is used throughout the initial hand calculations and subsequent iterations to predict resulting heat sink temperature, T_S. By knowing the power of heat source Q, heat sink surface area A, and an ambient temperature T_a, there is a second unknown, h, the heat transfer coefficient besides the unknown T_S. The heat transfer coefficient is used to estimate heat sink temperature T_s for the first-time hand calculation and later it is correlated with the experimental results to improve subsequent calculations and predictions.

There are numerous Nusselt number correlations available in reputed texts such as Incropera and DeWitt [20] to estimate the heat transfer coefficient h. If one has no idea of what value the heat transfer coefficient should be, Figure 4 can be used to make an initial prediction. By using the heat sink surface area A = 0.01159 m^2 and the test condition of total power dissipation rate Q = 5 Watts, one can calculate an approximate heat transfer flux of 431 watts/m^2 (i.e., Q/A = 5 Watts/0.01159 m^2). If a temperature difference (T_S - T_a) of 40ºC is estimated, Figure 4 reads an estimated value h =10 W/m^2-K for heat transfer coefficient. With known convective heat transfer coefficient h, the heat sink temperature can be calculated. For example, by using Eq. (2), plugging into numbers for heat dissipation rate Q, surface area A, heat transfer coefficient h, and ambient temperature T_a =10°C one can solve to get the heat sink temperature T_S=53.13°C.

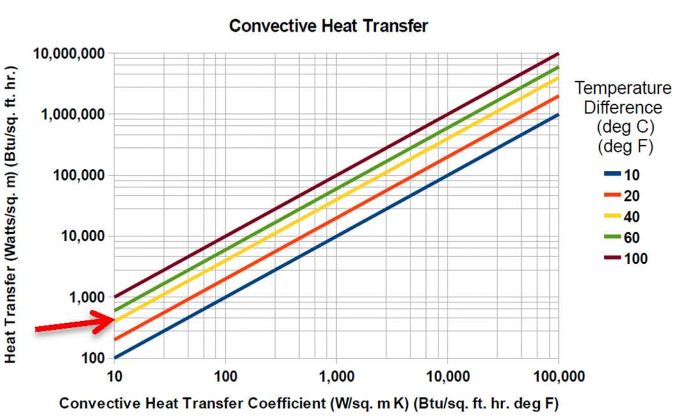

Figure 4 Convective heat transfer chart [21]

Similarly, by utilizing the convection power formula Eq. (2) and the estimated convective heat transfer coefficient, the calculated heat sink temperature results can be seen in Table 3, the column of "Hand Calculation". These data will be compared with the first-time CFD results and then with mock-up testing results.

Table 3 Initial heat sink temperature results

Ambient Temperature T_a = 10ºC; Orientation: Flat			
Power Q (Watts)	Heat Sink Temperature T_s (ºC)		
	Hand Calculation	CFD Simulation	Mock-up Testing
5	53.13	54.5	46.8
10	76.27	84.6	72.6
15	119.4	109	93.5

Initial CFD Simulation

By using the system testing conditions and the heat sink dimensions, a preliminary computation fluid dynamics (CFD) model is created using ANSYS ICEPAK 14.0. This software is used for all numerical simulations in the paper. Since there is no real world testing data to correlate CFD input with, variables for air and materials are chosen from the ANSYS library of materials included in the software. Later, a CFD validation study is performed for accuracy of the model with respect to these input variables when the experimental mock-up data can be correlated. The flow and heat transfer process are governed by the conservation laws as follows:

Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = 0 \tag{3}$$

Conservation of momentum

$$\frac{\partial}{\partial t}\left(\rho \vec{V}\right) - \nabla \cdot (\rho \vec{V}\vec{V}) = \rho \vec{g} - \nabla P + \nabla \cdot \tau \tag{4}$$

Conservation of energy

$$\frac{\partial}{\partial t}(\rho e) + \nabla \cdot \left(\rho e \vec{V}\right) = -P\nabla \cdot \vec{V} + \nabla \cdot (k \nabla T) + \dot{q} \tag{5}$$

Equation of state

$$P = \rho RT \tag{6}$$

where, ρ is the density, V is the velocity, g is the body force, P is the pressure, τ is the shear stress, e is the total energy, k is the thermal conductivity, T is the temperature, and $\dot{q}$ is the heat generation rate.

The numerical investigation starts with a minimum element in a gap of three and an iteration number of 50. A screen capture of one of the CFD simulations is presented in Figure 5 to show a temperature field. As expected, the sink bottom is the hottest and temperature gradually decreases toward the tip end. The CFD calculated heat sink temperature is presented in Table 3, the column of "CFD Simulation". By comparison, it shows some differences in heat sink temperature between hand calculation and CFD simulation. The comparison is not complete until the mock-up testing results are available.

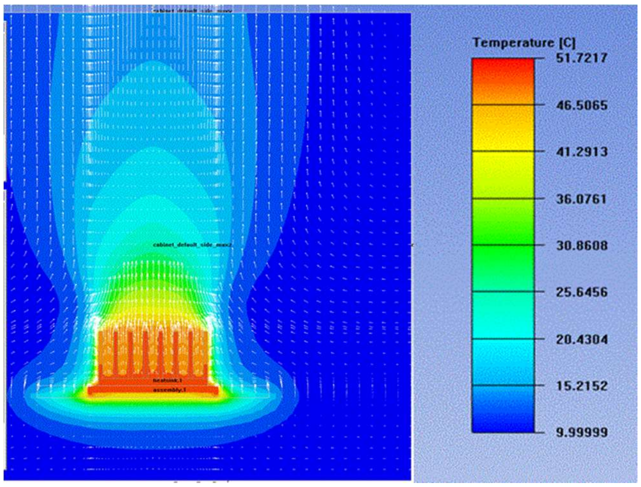

Figure 5 CFD results of heat sink temperature contour

Mock-up Testing

A simplified mock-up is built to test the heat sinks in three different orientations: Flat, Vertical and Horizontal as defined in Figure 3. A copper slug, as shown in Figure 6**,** is machined with a 13.7 mm by 13.7 mm raised pedestal to simulate the contact area of a small application-specific integrated circuit (ASIC) that has Shin-Etsu 7783 as the thermal interface material. A Kapton Minco heater is attached to the base of the copper slug that is able to provide the adjustable wattage up to 15 Watts. A piece of sheet-metal is bent with four standoffs press-fitted to allow the FR4 board, which is similar to the printed circuit board assembly (PCA) material, to be attached by screws. The FR4 has heat sink anchors to be glued to it such that the simulated application-specific integrated circuit heater is held to stay in place.

Two different chambers are used for environmental control in testing, a Russells Technical Products RHD-64 altitude chamber and a RTP RD-64 temperature/humidity chamber. Agilent VEE 6.0 is used in automating the testing and data collection. An Agilent Data Acquisition Unit 34970A is used to collect the thermocouple readings. One of these setups can be seen with a screen capture of the program in Figure 7. The results from the first-time mock-up testing are shown in Table 3, the column of "Mock-up Testing". Table 3 shows some differences in heat sink temperature using these three methods, indicating that improvements need to be made for prediction of future studies and correlation of the results.

By using the data collected at three different altitudes 5000ft, 10000, and 15000ft, respectively, it allows the altitude multiplier to be validated with mock-up results. Table 4 shows the comparison of heat sink temperature between hand calculation and test after altitude multiplier is employed. It indicates that the multiplier appears to be conservative, especially as the altitude increases there is a larger temperature difference (7.4 ºC, 10.3 ºC, and 13.5) between calculated and actual testing.

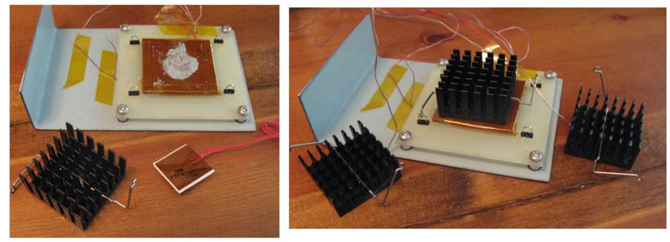

Figure 6 Illustration of built mock-up

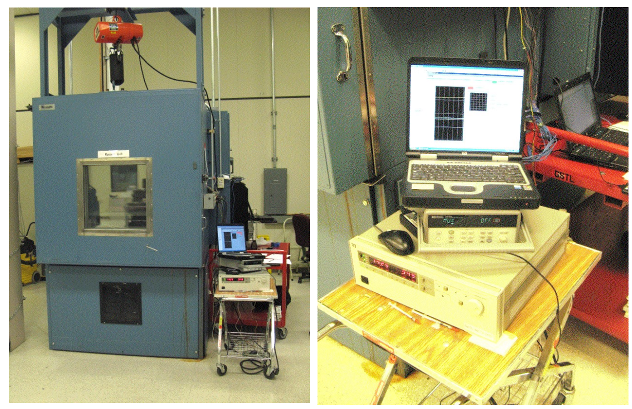

Figure 7 Mock-up with screen capture of Agilent VEE

Hand Calculation Iteration

The initial hand calculations are off the mock-up values from 3.7 ºC to 25.9 ºC as seen in Table 3. Most of these differences can be attributed to the estimate on convective heat transfer coefficient, h. Now that there are tested data to correlate to, one can reevaluate the convective coefficient and use this new convective heat transfer coefficient value for the first iteration of hand calculations. The new heat transfer coefficients at a flat orientation are shown in Table 5 and they are then used for calculation of heat sink temperature at two other orientations, vertical and horizontal.

Table 5 Corrected new heat transfer coefficient

T_a=10ºC; A =0.011592m^2; Orientation: Flat		
Power (Watts)	Tested T_s (ºC)	New h (W/m^2-K)
5	46.8	**11.72**
10	72.6	**14.55**
15	93.5	**16.22**

Table 4 Comparison of heat sink temperature between hand calculation and test using altitude multiplier

T_a =10ºC		Q=5Watts			Q=10Watts			Q=15Watts		
Multiplier	Altitude (feet)	T_s ºC Hand	T_s ºC Test	Delta to Test ΔT ºC	T_s ºC Hand	T_s ºC Test	Delta to Test ΔT ºC	T_s ºC Hand	T_s ºC Test	Delta to Test ΔT ºC
1	0	45.80	45.8	**0.0**	69.50	69.5	**0.0**	92.10	92.1	**0.4**
1.1	5000	50.38	46.8	**3.6**	76.45	74.5	**2.0**	101.31	96.7	**4.6**
1.21	10000	55.42	49.9	**5.5**	84.10	78.8	**5.3**	111.44	102	**9.4**
1.33	15000	60.91	52.6	**8.3**	92.44	82.1	**10.3**	122.49	109	**13.5**

CFD Simulation Iteration

The initial CFD simulation results had temperature differences from 1°C to 10°C from the mock-up values as seen in Table 3. To improve the correlation of the CFD data with testing ones, some constant are changed to improve the results. The first thing attempted is to get the data to correlate better across the different altitudes. To do this, specific air densities are calculated for the different altitudes at the temperatures tested. These density values instead of the standard air ones in the software library are provided in the simulation and it results in a significant improvement in terms of CFD simulation accuracy.

With mock-up testing data in hand, the first CFD validation study performed is to check whether enough iteration in the simulation is provided. By checking simulations at 40, 60, and 80 iterations, respectively, a resulting temperature change is 0.7°C between 40 and 60 iterations and 0.1°C between 60 and 80 iterations. Since the change in the simulation results is very small between 60 and 80 iterations, it is decided that 60 iterations are chosen for the rest of the simulations.

The second CFD validation study performed is to check the gridding. By continuing to use the Mesher-HD in ICEPAK, the minimum elements in the gap are adjusted at 1, 3, and 5, respectively, resulting in a corresponding mesh count of 130000, 31700, and 588000 cells respectively. Resulting heat sink temperature are observed to vary from 1.3°C between 130000 and 317000 mesh cells and 0.2°C between 317000 and 528000 mesh cells. Thus, it is decided to perform the rest of the simulations with 317000 grid cells. The importance of these validation and verification studies is to demonstrate credibility in the numerical simulation model and accuracy in the numerical simulation results.

After the validation study of numerical simulations is completed with variables to be adjusted and changed to improve correlation between the CFD calculation and mock-up testing results, rest of the simulations are conducted as an iteration process for heat sink at different orientations of flat, vertical and horizontal and different altitudes of 5,000ft, 10,000ft and 15,000ft subjecting to three different power dissipation levels of 5, 10, and 15 watts, respectively. A summary of the results are shown in Table 6. Compared to the initial CFD simulation results, new results show a much improved correlation and accuracy, especially at high altitude for the orientations of flat and horizontal. There are still outliers at higher altitudes (15,000 feet) for vertical orientation of heat sink. This might be caused by some airflow in the test chamber.

Overall, the results have improved and now the learning's from these studies can be applied to predict the heat sink temperature at two other different heat sink heights (19.97mm and 16.45mm) with three different orientations (flat, vertical and horizontal). The heat sink is subject to a different ambient temperature of T_a = 30°C than T_a =10°C.

Final Investigation

The final test configurations are for all three heat sinks at a new ambient temperature of 30°C rather than 10°C. This is the 2rd iteration for the 23mm heat sink but first-time calculation at 30°C ambient temperature, and a first pass for the heat sink at two other different heights of 19.75mm and 16.45mm. Results for the heat sink temperature can be viewed in Table 7, Table 8, and Table 9, respectively.

Table 7 shows that, results for the heat sink at height of 23mm are within 7.6°C for all three orientations and all three power levels when hand calculation and CFD simulation results are compared to testing ones. Results for power levels at each orientation (flat, vertical and horizontal) are within 3°C of each other except for one reading that is 5°C. This demonstrates consistent results that are correlating. It should note that the ambient temperature affects the natural heat transfer and the adjustment for heat transfer coefficient value should be sufficient when the ambient temperature is switched from 10°C to 30°C.

The heat sink at height of 19.75mm is only a first-time calculation for predicting results. Table 8 indicates that, the CFD results are fairly consistent with each other and are within 4°C of each other except for one outlier that is 6°C. Compared to test results, the CFD results are within 8.7°C for all orientations and all power levels. However, hand calculation could use some improvement since as much as 14.5°C off from mock-up results is observed. The hand calculation results even vary within an orientation, which signifies that there could be some improvement to the multiplier used for a better correlation for next iteration.

Table 6 CFD results of heat sink from first iteration and comparison with test data

T_a =10ºC	Q=5Watts		
Flat	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	45.8	46.8	**-1**
5000	46.8	49.3	**-2.5**
10000	49.9	52.6	**-2.7**
15000	52.6	56.4	**-3.8**

T_a =10ºC	Q=5Watts		
Vertical	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	46.8	47.2	**-0.4**
5000	48.6	51.3	**-2.7**
10000	50.8	56.2	**-5.4**
15000	53.7	61.9	**-8.2**

T_a =10ºC	Q=5Watts		
Horizontal	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	49.6	47.5	**2.1**
5000	50.3	50.2	**0.1**
10000	52.1	53.1	**-1**
15000	55.4	56.1	**-0.7**

T_a =10ºC	Q=10Watts		
Flat	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	69.5	69.3	**0.2**
5000	74.5	74.4	**0.1**
10000	78.8	79.9	**-1.1**
15000	82.1	85.4	**-3.3**

T_a =10ºC	Q=10Watts		
Vertical	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	72.6	70.9	**1.7**
5000	76.3	77.8	**-1.5**
10000	81.3	85.6	**-4.3**
15000	85.7	94.8	**-9.1**

T_a =10ºC	Q=10Watts		
Horizontal	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	73.3	72.3	**1**
5000	78.6	76.8	**1.8**
10000	84.4	81.6	**2.8**
15000	87	86.6	**0.4**

T_a =10ºC	Q=15Watts		
Flat	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	92.1	89.8	**2.3**
5000	96.7	96.1	**0.6**
10000	102	103.3	**-1.3**
15000	109	111.1	**-2.1**

T_a =10ºC	Q=15Watts		
Vertical	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	93.5	91.5	**2**
5000	100	100.4	**-0.4**
10000	107	110.8	**-3.8**
15000	111	123.2	**-12**

T_a =10ºC	Q=15Watts		
Horizontal	Test	CFD	
Altitude (feet)	T_s ºC	T_s ºC	Delta ΔT
0	97	93.5	**3.5**
5000	103	99.7	**3.3**
10000	110	106.2	**3.8**
15000	111	112.7	**-1.7**

Table 7 Results for heat sink at a height of 23mm

H = 23mm T_a = 30ºC	**Orientation Flat**					**Orientation Vertical**					**Orientation Horizontal**				
Power (Watts)	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test
	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**
5	59.3	65.8	**6.5**	66.9	**7.6**	64.6	67.6	**3**	69.2	**4.6**	61.3	67	**5.7**	67.9	**6.6**
10	83	89.3	**6.3**	89.1	**6.1**	94.7	92.1	**-2.6**	93.2	**-1.5**	85.5	90.9	**5.4**	91.1	**5.6**
15	103	109.5	**6.5**	109.9	**6.9**	110	113.4	**3.4**	114.9	**4.9**	105	111.3	**6.3**	112	**7.0**

Table 8 Results for heat sink at a height of 19.75mm

H = 19.75mm T_a = 30ºC	**Orientation Flat**					**Orientation Vertical**					**Orientation Horizontal**				
Power (Watts)	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test
	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**
5	63	70.5	**7.5**	72.9	**9.9**	67.5	70.1	**2.6**	75.6	**8.1**	64.9	69.9	**5**	74.1	**9.2**
10	91.1	96.6	**5.5**	98.8	**7.7**	98.5	96.5	**-2**	103.7	**5.2**	92.3	95.8	**3.5**	101.2	**8.9**
15	111	119.7	**8.7**	123	**12**	116	119.3	**3.3**	128.9	**12.9**	111	118.4	**7.4**	125.5	**14.5**

Table 9 Results for heat sink at a height of 16.45mm

H = 16.45mm T_a = 30ºC	**Orientation Flat**					**Orientation Vertical**					**Orientation Horizontal**				
Power (Watts)	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test	Test	CFD	Delta To Test	Hand	Delta To Test
	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**	T_s ºC	T_s ºC	**ΔT ºC**	T_s ºC	**ΔT ºC**
5	66.9	74.7	**7.8**	81.5	**14.6**	70.5	73	**2.5**	84.8	**14.3**	69.5	73	**3.5**	82.9	**13.4**
10	96.7	104	**7.3**	112.6	**15.9**	102	101.8	**-0.2**	118.4	**16.4**	97.6	102.1	**4.5**	115.4	**17.8**
15	114	129.6	**15.6**	141.7	**27.7**	122	126.8	**4.8**	148.7	**26.7**	116	127.5	**11.5**	144.7	**28.7**

The results for heat sink at height of 16.45mm as shown in Table 9 do not fare well. The CFD results are reasonably good except for some outliers at high power level. The hand calculations do not predict results well. They are off the mock-up values by a range from 13.4 °C to as much as 28.7°C. The reason for the difference is that the thermal performance of heat sinks heavily relies on free convection which in turn is affected by the fin height and heat sink orientation. The estimated heat transfer coefficient must take these factors into consideration. Considering this is a first-time calculation, results will be improved with iteration. This demonstrates the importance of iterating to improve accuracy and better correlation with theoretical and empirical data.

CONCLUSION

The methodology in this paper presents an approach to the thermal design for a finned heat sink. It demonstrates quick feasibility studies to a built mock-up with progressively improved accuracy in results by iterating and correlating data. Initial hand calculation results are as much as 26 ºC off the tested results but after one iteration they are improved to 3ºC and the results are 7 ºC off when correlated to a new ambient temperature. Similar results are seen using numerical simulation (CFD approach) that begins 15.5 ºC off the tested results but they are improved to 2 ºC in one iteration and 6.5 ºC when correlated to a new ambient temperature. Empirical data helps confirm the accuracy in the process and increases credibility to the theoretical results. This allows for improved optimization studies and helps reduce design cycle time. The proposed design methodology can be used for other thermal investigation with formulas to be changed to meet the design requirements, while the process still remains the same.

NOMENCLATURE

T	Temperature (°C or K)
Q	Heat transfer rate / Power (W)
A	Heat sink surface area (m^2)
h	Heat transfer (convection) coefficient ($W/(m^2 K)$)
H	Heat sink height (m)
PCA	Printed Circuit Board Assembly
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit

REFERENCES

[1] Bash, C.E., and Chandrakant D.P., "Thermal Management of Electronics, Systems, and Packages," San Jose: Hewlett Packard Laboratories, 2003.

[2] Mohammed, R.K.; Yi Xia; Sahan, R.A.; Pang, Y., "Performance improvements of air-cooled thermal tool with advanced technologies," Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM), 2012 28th Annual IEEE, pp.354-361, March18-22, 2012.

[3] Kang, S., "Advanced Cooling for Power Electronics." Presented by invitation at International Conference on Integrated Power Electronics Systems, CIPS 2012, March 6-8, 2012, Nuremberg, Germany.

[4] Chu, R.C., Simons, R.E., Ellsworth, M.J., Schmidt, R.R., Cozzolino, V., "Review of cooling technologies for computer products," Device and Materials Reliability, IEEE Transactions on, Vol.4, no.4, pp.568 -585, Dec. 2004.

[5] Yang, C.Y., Yeh, C.T., Wang, P.K., Liu, W.C. and Kung, E.Y.C., "An in-situ performance test of liquid cooling for a server computer system," Microsystems Packaging Assembly and Circuits Technology Conference (IMPACT), 2010 5th International, pp.1-4, Oct. 20-22, 2010. Taipei, Taiwan.

[6] Mahajan, R., Chiu, C., Chrysler, G., Dev, A.T. and Chandler, A.Z., "Cooling a microprocessor chip," Proceedings of the IEEE, Vol. 94, 2006, pp.1476–1486.

[7] Satish G.K., "Review and Projections of Integrated Cooling Systems for Three-Dimensional Integrated Circuits," Journal of Electronic Packaging, Transactions of the ASME, June 2014, Vol. 136 / 024001-11.

[8] Kandlikar, S.G., "History, Advances, and Challenges in Liquid Flow and Flow Boiling Heat Transfer in Microchannels: A Critical Review," ASME J. Heat Transfer, Vol.134, no.3, pp. 034001, 2012.

[9] Kandlikar, S.G., Colin, S., Peles, Y., Garimella, S., Pease, R.F., Brandner, J.J., and Tuckerman, D.B., "Heat Transfer in Microchannels - 2012 Status and Research Needs," ASME J. Heat Transfer, Vol.135, no.9, pp.091001, 2012.

[10] Zhao J., Akanova, A., Yin, S., "Optimization and comparison of double-layer and double side micro channel heat sinks with nanofluid for power electronic cooling"-ELSEVIER-Applied thermal engineering, Vo.65, pp.124-134, 2014.

[11] Wang, G.L., Yang, D.W., Wang, Y., Niu, D., Zhao, X.L. and Ding G.F.， " Heat Transfer and Friction Characteristics of the Microfluidic Heat Sink with Variously-Shaped Ribs for Chip Cooling"，Sensors, Vol.15, pp. 9547-9562, 2015.

[12] Altitude Multiplier image, Website: http://www.electronics-cooling.com/1999/09/adjusting-temperatures-for-high-altitude/

[13] Humid Air image, Website: http://www.electronics-cooling.com/2003/11/the-thermal-conductivity-of-moist-air/

[14] ASHRAE Allowable image, Website: http://nsidc.org/about/green-data-center/project.html

[15] Ahmadi, M., Mostafavi, G., and Bahrami, M., "Natural Convection from Rectangular Interrupted Fins," Int. J. Therm. Sci., Vol.82, pp. 62-71, 2014.

[16] Chen, H.T., Lai, S.T. and Haung, L.Y., "Investigation of Heat Transfer Characteristics in Plate-Fin Heat Sink," Appl. Therm. Eng., Vol. 50, no. 1, pp. 352-360, 2013.

[17] Huang, R.T., Sheu, W.J. and Wang, C.C., "Orientation Effect on Natural Convective Performance of Square Pin Fin heat sinks," Int. J. Heat Mass Transf., Vol. 51, no. 9 -10, pp. 2368–2376, May 2008.

[18] Rana, R.A. and Li, R., "Design of Heat Sink for Orientation-Independent Free Convection," CSME International Congress 2014, June 1-4, 2014, Toronto, Canada.

[19] Lee, S., "Optimum Design and Selection of Heat Sinks," IEEE Trans. components, Package. Manuf. Technol. Part A, Vol. 18, no. 4, pp. 812-817, 1995.

[20] Incropera, F.P., DeWitt, D.P., Bergman, T.L. and Lavine, A.S., "Fundamentals of Heat and Mass Transfer," 7th edition, 2011.

[21] Engineering Risk-Benefit Analysis. Prof. George Apostolakis. Spring 2007. MIT OpenCourseWare. http://ocw.mit.edu/courses/engineering-systems-division/esd-72-engineering-risk-benefit-analysis-spring-2007/lecture-notes/.

[22] Paterson, M.K., "The Effect of Data Center Temperature on Energy Efficiency", Thermal and Thermomechanical Phenomena in Electronic Systems, pp1167 – 1174, May 28-31, 2008, Orlando, FL, USA.

[23] O'Connor P.P., Kleyner A., "Practical Reliability Engineering,", Wiley, 5th Edition, 2011.

[24] Thuesen, G.J., and Fabrycky, W.J., "Engineering Economy." 9th ed. Englewood Cliff, NJ: Prentice-Hall, Inc., 2000. ISBN: 9780130281289.

[25] Engineering Risk-Benefit Analysis. Prof. George Apostolakis. Spring 2007. MIT OpenCourseWare. http://ocw.mit.edu/courses/engineering-systems-division/esd-72-engineering-risk-benefit-analysis-spring-2007/lecture-notes/.

[26] Convective Heat Transfer image, Website: http://www.engineeringtoolbox.com/convective-heat-transfer-d_430.html
[27] Different types of heat sinks, Website: http://www.pencom-web.com/manufacturing/fabrication.aspx
[28] Heat pipe heat sink, Website: http://www.pa-international.com/products/heatsinks-heat-pipes-china