HEFAT2012
9th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
16 – 18 July 2012
Malta

THE STUDY ON DESIGN OF FLOW PATH AND EFFECT OF HEAT REMOVAL FOR LIQUID-COOLED THERMAL MODULE

Ching-Song Jwo[1*], Ming-Wei Lin[2], Chun-Yu Lu[3]
[1*] Professor, Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology
[2] Doctoral Program, Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology
[3] Master Program, Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology
E-mail: frankjwo@ntut.edu.tw

ABSTRACT

Semiconductor components have their power enhanced and their effectiveness improved gradually, leading to an increasing demand of heat removal in them. Therefore, an increase in heat removal volume of heat sink is currently an important issue. It is known that water cooling system can solve the problem effectively. The paper, mainly focusing on the liquid-cooled thermal module used for heat generation of microchip in water cooling system, alters the geometric change in its flow path, and explores its influence on heat removal effectiveness of thermal module. Through a software for calculation of fluid mechanics, and verification by practical experiments, the paper analyzes three kinds of thermal modules so as to explore the change and influence of heat removal performance. Using the software FLUENT for calculation of fluid mechanics, the paper calculates the heat sink of three different modules, explores the thermal conduction problem under different CPU wattages, and observes the internal flow path change and temperature field distribution. After that, through calculation of fluid mechanics, the paper predicts the flow path change and temperature field distribution under different flow velocities. When simulation analysis is made, it can be found that heat resistance value of thermal module does not have obvious change with the rise of wattage, but contrarily decreases with the increase of flow velocity. Finally, the paper induces that snake-shaped cylindrical composite flow path can achieve the best heat removal effect; snake-shaped flow path is the second best one; and cylindrical flow path is the worst. The heat resistance values of these three flow paths are 0.08℃/W, 0.1℃/W and 0.15℃/W respectively.

INTRODUCTION

In recent years, the development and design of computer technology tend to take thinning, small sizing, high frequency and high performance as the core objectives. Relatively, the heat removal problem of electronic products is getting more and more serious. Among all accessories and parts of computer, the core technical part is central processing unit (CPU). Just like the heart of computer, CPU is responsible for the main calculation and commands of computer. In order to achieve the goal of transistor growth as mentioned in Moore's Law, the development of CPU has to give consideration to cost. Therefore, new process technology is urged to evolve continuously and shorten the line width. As a result, without the need to enlarge the volume of silicon crystal, more transistors can be put in it. Therefore, the higher the grade of CPU, the higher its operation clock, the more the number of transistors, and relatively, the higher the density of heat generation.

The heat removal tools that are commonly used today include sink of natural convection, fan-attached sink of compulsory convection, water cooling system, the phase-changed heat pipe and vapor chamber, thermoelectric cooling chip …etc. However, the currently marketed computerized heat removal tools are mainly air cooled. With the improvement of efficacy and increase of power in these tools, the demand of heat removal volume is getting great. Therefore, increasing the heat removal volume of heat sink and accelerating the rotational velocity of fan can really ensure improved heat removal effect of system, but the derived noise and heat accumulation problems are getting serious. As for water cooling heat exchanger, it has the advantages of great removal of heat and low noise, so that heat removal problem can be solved. In fact water cooling system has several advantages: water can be obtained easily; and water's greater heat removal volume than air can bring the heat produced from heat source to other places for removal, making the use of space more flexible, and creating no such problems as noise from fan and vibration from machine.

The research of the paper mainly aims to explore the part of thermal module in water cooling system. Using the numerical simulation analysis software FLUENT, the paper analyzes three kinds of thermal modules, namely snake-shaped, cylindrical, and snake-shaped cylindrical composite thermal modules, and changes different import flow velocities, temperatures and CPU heat generation in order to analyze their advantages, disadvantages and heat removal effectiveness, and acquire optimal design of products.

INTRODUCTION OF LIQUID-COOLED HEAT REMOVAL SYSTEM

The principle of water-cooled thermal system is that water-cooled motor delivers working fluid to cool water inlet, from which water flows to the thermal module above the chip of central processor. Through thermal conduction of thermal module and heat convection effect of working fluid, the heat produced from the chips of central processor is removed, and heat is carried to heat exchanger. Through heat exchanger, heat is exhausted to air, and finally flows back to water tank and motor, and then forms a close-loop circulation system. In the experiment of the paper, water cooling system is equipped with flow meter, thermocouple wire and temperature logger. Through water cooling motor, the size of flow velocity can be controlled, and the influence of flow velocity on heat removal effect can be explored. Thermocouple wire can be placed between thermal module and the chip of central processor, and temperature logger can be connected to record the temperature change of the chip of central processor at all times.

NUMERICAL ANALYSIS

The study uses commercial-use software package FLUENT6.3.26 to carry out simulative calculation and analysis. The flow fields that can be calculated and analyzed by FLUENT are laminar flow, turbulent flow, steady flow field, transient flow field, concentration diffuse flow field, Newton flow field, non-Newton flow field or porous media flow field. The calculation ways of convection item and diffusion item include upwind differencing, QUICK (Quadratic Upstream Interpolation of Convective Kinematics) and power law. The handling ways of turbulent flow field are K-ε, Realizable K-ε and RNG K-ε. The solutions of flow field and pressure include the checking calculation ways of SIMPLE、SIMPLEC and PISO.

The operation process of simulative calculation by FLUENT can be divided into three parts for handling. The first part takes GAMBIT2.4.6 for drawing simulative physical model, establishing grids and setting boundary conditions. The second part imports the files already set in the first part, to FLUENT, and then sets the calculation way of flow field, convergence range and iteration number. The final part carries out data analysis and drawing of simulative results.

BASIC SUPPOSITIONS

The study is a simplified numerical analysis, so that the following basic suppositions are made:

(1) Suppose the working fluids are at 30^{o}C, 25^{o}C and 20^{o}C.
(2) Suppose the flow velocities of working fluid inlets are 0.5m/s and 1m/s.
(3) Suppose the heating wattages of chips are 90W, 120W, 150W and 200W.
(4) The relative velocities of solid surfaces to fluids are all supposed to be zero, implying to non-sliding condition.
(5) Fluid is Newton fluid, and cannot compress fluid.
(6) Flow field is steady laminar flow field.
(7) Radiation effects are neglected.
(8) Heat transfer coefficient is a constant (k = constant).

SETTING OF PARAMETERS AND FLOW PATH SIMULATION DIAGRAMS

In the numerical analysis software of the paper, the constituting components established by model include flow paths in thermal module and CPU heat source chips. Figure 1, Figure 2 and Figure 3 show different kinds of thermal module after numerical simulation by FLUENT. In order to achieve accuracy of simulation, it has to cope with the given component dimensions and specifications of actual objects as well as material parameters, as shown in Tables 1 and 2.

Table 1 Specifications of thermal module components

Name	Dimensions	Material	Unit
Heat sink	L86×W63×H12	Copper	mm
Heat generation chip	L32×W32×H1	Copper	mm
Water inlet and outlet	ϕ6	Copper	mm
Cylinder	D4×H9	Copper	mm
Flow path	L68 ×W44 ×H9	Copper	mm
Upper edge cover	L86×W63×H10	Acrylic	mm

Table 2 Material parameters

Name	Specific heat	Thermal conductivity	Density
Copper	381 J/kg・k	387.6W/m・K	8978 kg/m^3
Acrylic	1300 J/kg・k	0.08W/m・K	1165 kg/m^3
Water	4182 J/kg k	0.6W/mK	998.2kg/m^3

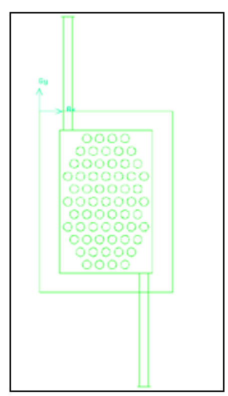

Figure 1 Simulation diagram of cylindrical flow path

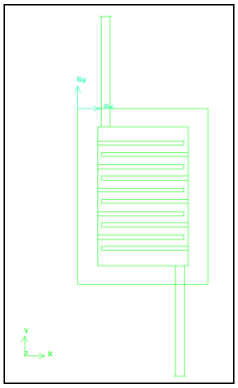

Figure 2 Simulation diagram of snake-shaped flow path

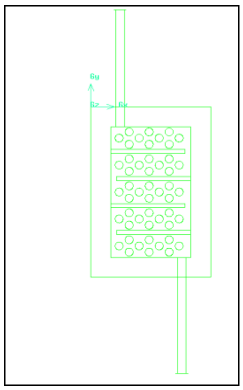

Figure 3 Simulation of snake-shaped cylindrical composite flow path

GRID SHAPE

After geometric model is constructed, pre-treatment has to be made by establishing a grid system. It is known that software package uses the calculation way of fluid mechanics developed from limited control volume. The configuration and quantity of grids have certain extent of influence on the correctness of solutions. Too many grids would increase the capacity of memory and calculation time, but too less grids would reduce the correctness of solutions.

The simulated grids are constructed by GAMBIT2.4. The simulation targets of the study are cylindrical flow path, snake-shaped flow path, and snake-shaped cylindrical composite flow path, which have complicated directions and angles. Therefore, tetrahedral grids are selected as they can better accommodate geometric shapes than hexahedral grids. First of all, the laying and distribution of routes are adjusted. Then, triangular grids are laid, and they would automatically develop to be the volume grid of TGrid. After completion, the grid quality should be inspected, where the equi angle skew should not be greater than 0.97; otherwise convergence would not be performed. The grid diagrams are shown in Figure 4, Figure 5 and Figure 6.

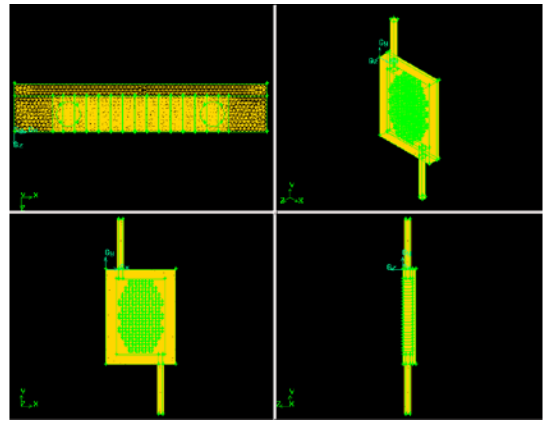

Figure 4 Grid distribution diagram of cylindrical flow path

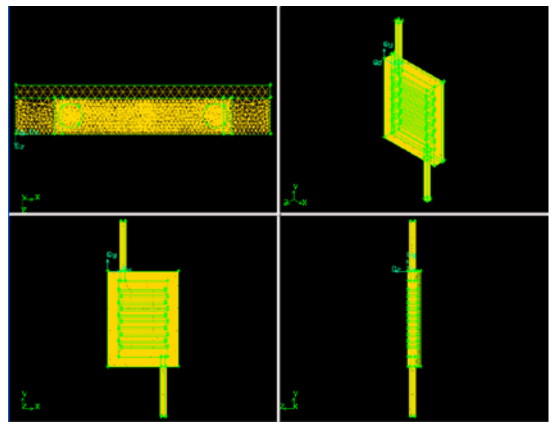

Figure 5 Grid distribution diagram of snake-shaped flow path

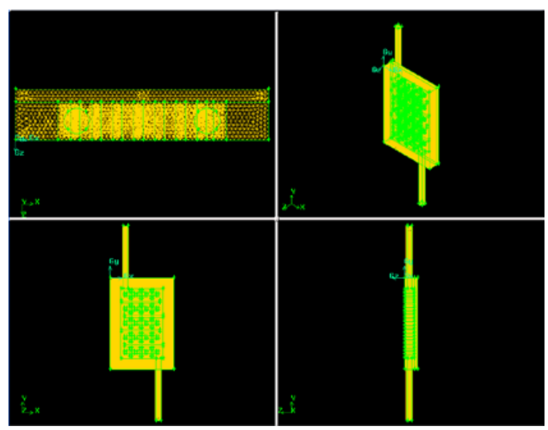

Figure 6 Grid distribution diagram of snake-shaped cylindrical composite flow path

RELAXATION FACTORS AND DECRETIZATION

Generally speaking, suitable low decretization coefficient is helpful to convergence. However, if decretization coefficient is fixed to be too small, convergence will be slow, and calculation time will be increased; but if it is fixed to be too great, divergence will occur easily. Therefore, it is very important to select a suitable low-relaxation coefficient. The relaxation factors in numerical simulation are the designed values set in FLUENT6.3.26 program, with numerical values shown in Table 3. The selection of discretization in the study is shown in

Table 4.

Table 3 Size of relaxation factors

Item	Size of relaxation factors
Pressure	0.4
Momentum	0.7
Viscosity	1
Density	1
Body Forces	1

Table 4 Selection of discretization

Item	Discretization
Pressure	Standard
Momentum	First Order Upwind
Pressure Velocity Coupling	SIMPLEC
Skewness Correction	0

RESULTS AND DISCUSSION

The study of the paper mainly explores the influence of geometric design of flow paths in different liquid-cooled thermal modules on the change of flow field and heat removal performance. The experimental parameters include heating wattages 90W, 120W, 150W and 200W, inlet flow velocities 0.5 m/s and 1 m/s, and temperatures 30℃, 25℃ and 20℃. Finally, a set of optimal parameters is taken to explore the heat removal performance of each sink. The paper is going to present analysis on the flow field, temperature field and pressure field of tangent plane by using numerical analysis software under the conditions that the various kinds of thermal modules are under the same heating wattage (200W), inlet flow velocity (1m/s) and inlet temperature (25°C), and the flow paths inside thermal modules are of half height. In addition, the paper will explore the advantages and disadvantages of different kinds of thermal modules, their heat removal effectiveness, and mutual influence of different field maps, and also compare the experimental values.

CYLINDRICAL FLOW PATH

Copper cylinders are placed in a thermal module. When working fluid impacts and flows to the copper cylinders, the flow field inside will become spiral flow field, and produce spiral flow phenomenon. It mainly suppresses separation of boundary layers, increases disturbance kinetic energy, and forces working fluid in flow path to disperse to wall surface. By doing so, better heat transfer effect can be obtained. As explored by the paper, the cylinder of simulated cylindrical flow path has two kinds of dimensions: diameter 3mm and diameter 4mm. As shown in Figures 7 and 8, through the difference in diameter of cylinder, the paper explores the distribution of flow fields and their heat removal effectiveness.

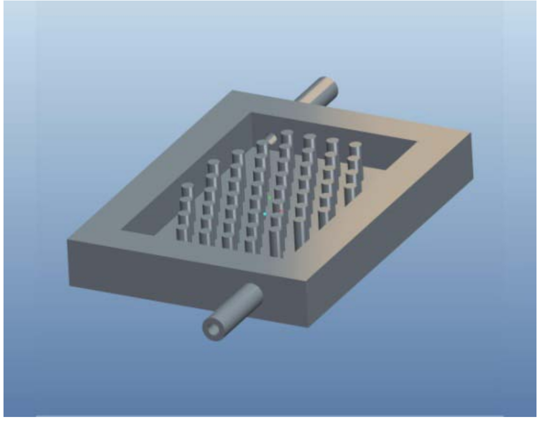

Figure 7 Cylindrical flow path with cylinder diameter 3mm

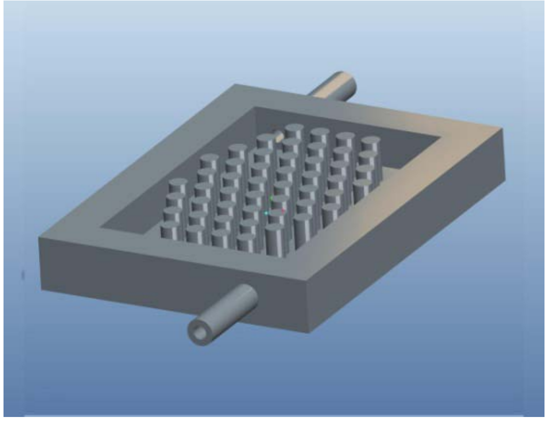

Figure 8 Cylindrical flow path with cylinder diameter 4mm

ANALYSIS ON CYLINDRICAL FLOW PATH FIELD MAPS

The analytical results of field maps show that with the growth of flow volume, Reynolds number increases, and the spiral effect produced around cylinders is more obvious. The position at the top right corner of such thermal module is opposite to the flowing direction of fluid, thus forming a retention zone easily. As found in the distribution diagrams shown in Figure 9 and Figure 10, spiral phenomenon is found in the flow field between cylinders. The purposes of these two thermal modules are to produce spiral strength through separated flow phenomenon on cylinders, and enhance heat transfer effect. In the thermal module with cylinder diameter 4mm, the distanced gap between cylinders is smaller than that of cylinders with diameter 3mm; the spiral strength produced is greater than that of cylinders with diameter 3mm; and heat transfer effect is also relatively better.

Figure 11 and 12 show the distribution diagrams of pressure fields. From these figures, it is observed that when fluid enters the inlet and impacts the cylinders, the pressure becomes great, and the pressure distribution inside flow path is rather even. After fluid come out of the outlet, there is obvious differential pressure. The inlet-outlet differential pressure of cylinders with diameter 3mm is 858.7 N/m^2, whereas the inlet-outlet differential pressure of cylinders with diameter 4mm is 839.1 N/m^2. The pressure loss caused by cylinder diameter 3mm is greater than that caused by cylinder diameter 4mm.

Figures 13 and 14 show the temperature distribution diagrams of cylindrical flow path. As seen from these figures, the temperature distribution of the top right corner and lower left corner is higher than other zones because the flow direction of fluid does not pass through this zone directly. The spiral effect of cylinder brings heat away from this zone. Its flow velocity is not as high as the one directly passing through the central area. Therefore, temperature tends to be rather high. As found in the figures, the temperature distribution at the central areas of these two thermal modules is obviously different. Since the volume of cylinder with diameter 4mm is greater than the volume of cylinder with diameter 3mm, the retention time of fluid in flow path is longer, and heat exchange effect is better than that of cylinder with diameter 3mm. Therefore, the temperature distribution of its central area is higher than that of cylinder with diameter 3mm.

After synthesizing these three kinds of field maps for making comparison, it is found that the cylinder with diameter 4mm has smaller pressure loss. As to the aspect of heat exchange, the cylinder with diameter 3mm has longer result; but for overall heat removal effectiveness, the cylinder with diameter 4mm is better than the cylinder with diameter 3mm. Therefore, the cylinder with diameter 4mm is selected to carry out simulation in experiments for making comparison.

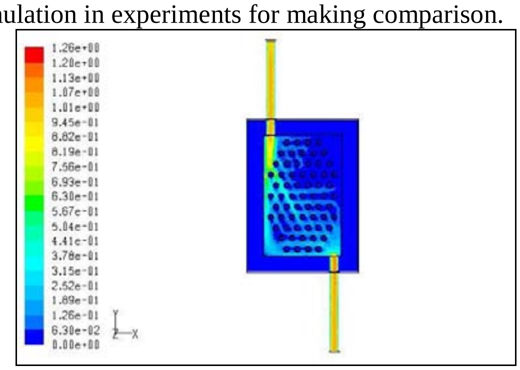

Figure 9 Velocity field distribution diagram of cylindrical flow path with cylinders at diameter 3mm

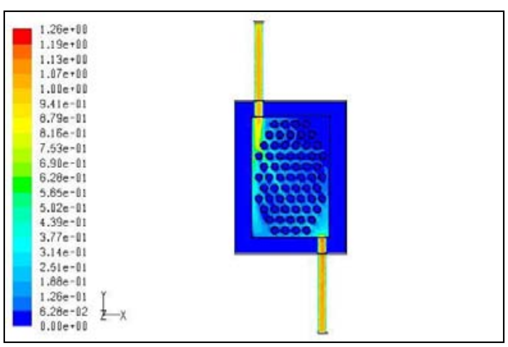

Figure 10 Velocity field distribution diagram of cylindrical flow path with cylinders at diameter 4mm

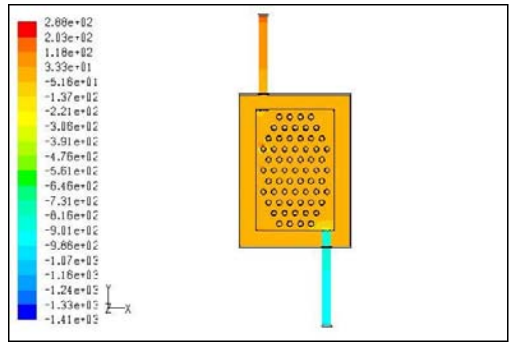

Figure 11 Pressure field distribution diagram of cylindrical flow path with cylinders at diameter 3mm

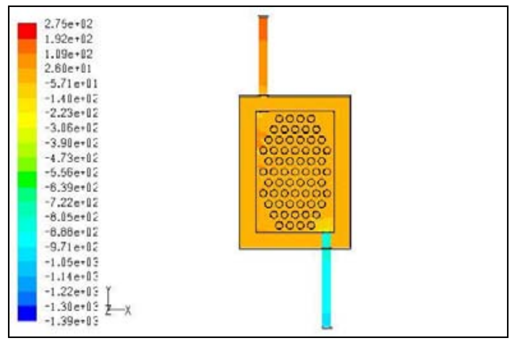

Figure 12 Pressure field distribution diagram of cylindrical flow path with cylinders at diameter 4mm

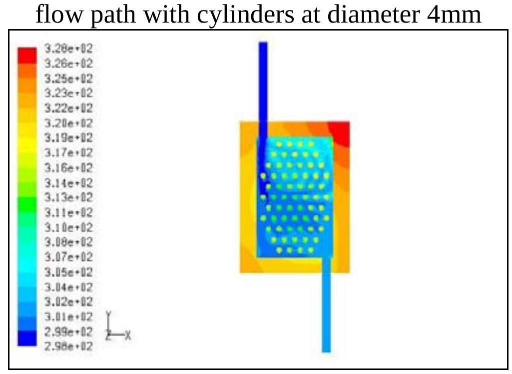

Figure 13 Temperature distribution diagram of cylindrical flow path with cylinders at diameter 3mm

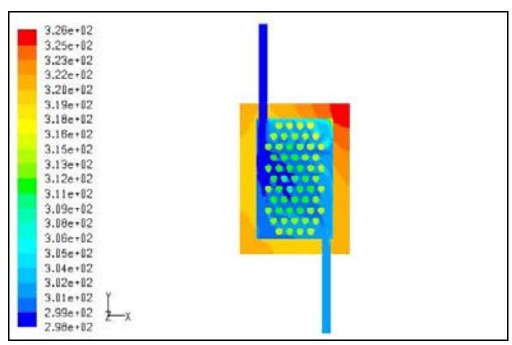

Figure 13 Temperature distribution diagram of cylindrical flow path with cylinders at diameter 4mm

SNAKE-SHAPED FLOW PATH

Put a baffle in the flow path of thermal module, making fluid flow in the established baffle. The fluid is forced to flow in flow path at a velocity being not so fast, increasing the time for fluid to stay in thermal module and enhancing heat exchange effect. As a result, better heat transfer effect is acquired. The snake-shaped flow paths explored and simulated by the paper are in two dimensions, and are placed with 6 baffles and 10 baffles respectively, as shown in Figures 15 and 16. Through the difference in the number of baffles, the paper explores the distribution of flow fields and its heat removal effectiveness.

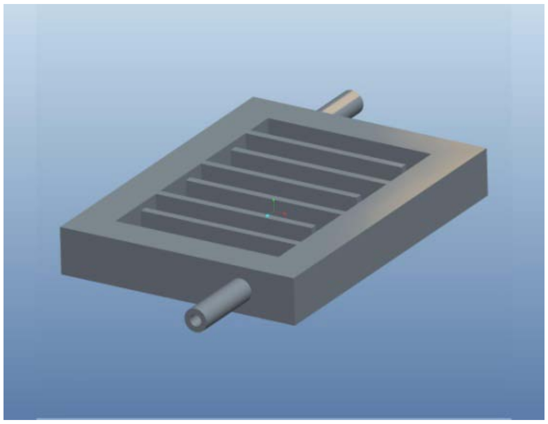

Figure 15 Snake-shaped flow path (6 baffles)

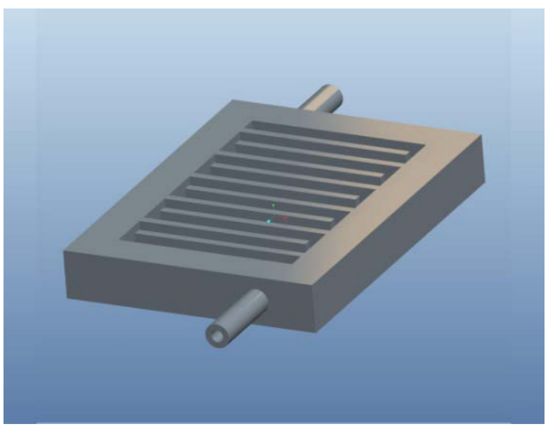

Figure 16 Snake-shaped flow path (10 baffles)

ANALYSIS OF SNAKE-SHAPED FLOW PATH FIELD MAPS

As shown from the velocity field distribution diagram of snake-shaped flow path (6 baffles) in Figure 17, through establishment of baffle, the flow direction of working fluid can also increase the flow velocity of working fluid in thermal module. Especially at the turning corner between two baffles, the flow velocity is more obvious. When the velocity field distribution diagram is compared with that of cylindrical flow path in Figure 9, the uneven flow velocity of flow path in thermal module of Figure 17 can be effectively improved. Figure 18 shows the velocity field distribution diagram of snake-shaped flow path (10 baffles). After it is compared with Figure 17, it is found that the increase in the number of baffles in Figure 18 makes the width of flow path narrowed, and thus increases the flow velocity.

Figures 19 and 20 show the pressure field distribution diagram of snake-shaped flow paths, with the flow paths placed with 6 and 10 baffles respectively. As found in these figures, when it is closer to the outlet, the pressure is greater. Besides, the number of baffles in Figure 20 is greater than that in Figure 19, making the width of flow paths in thermal module narrowed a lot, and relatively creating greater pressure. The inlet-outlet differential pressure in Figure 19 is 6283.3 N/m^2, whereas the inlet-outlet differential pressure in Figure 20 is 15523.4 N/m^2.

Figure 21 and Figure 22 are temperature field distribution diagrams of snake-shaped flow paths, which are placed with 6 baffles and 10 baffles respectively. From the comparison between Figure 21 and 22, it is found that the central area's temperature in Figure 21 is higher than that in Figure 22. The CPU base plate temperature of Figure 21 is 48°C, whereas the CPU base plate temperature of Figure 22 is 43°C, with a difference of 5°C in between. It also proves that increase in the number of baffles would make fluid stay in thermal module for some time and increase heat exchange effect, thus enhancing heat removal effectiveness.

Synthesizing the field maps of the above three kinds, it is found that both the flow velocity and pressure of snake-shaped flow path placed with 10 baffles are greater than those of snake-shaped flow path placed with 6 baffles. Besides, as observed from the CPU base plate temperature, the flow path placed with 10 baffles is 5°C lower than that with 6 baffles. Therefore, the snake-shaped flow path with 10 baffles is selected to carry out experiments and compare with the simulation.

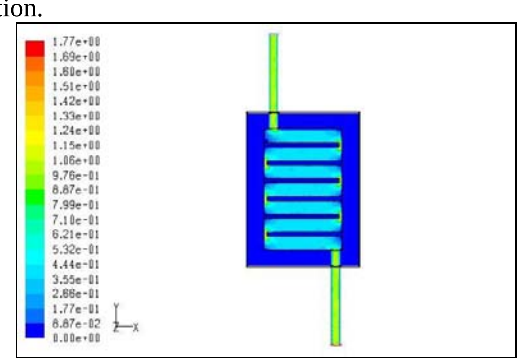

Figure 17 Velocity field distribution diagram of snake-shaped flow path (6 baffles)

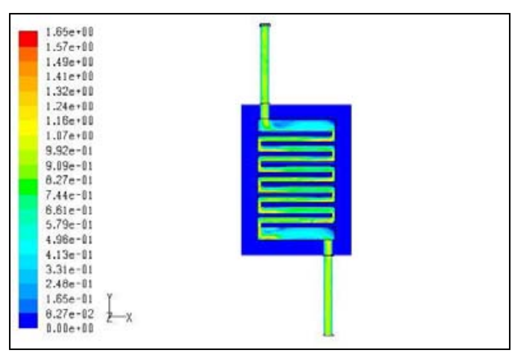

Figure 18 Velocity field distribution diagram of snake-shaped flow path (10 baffles)

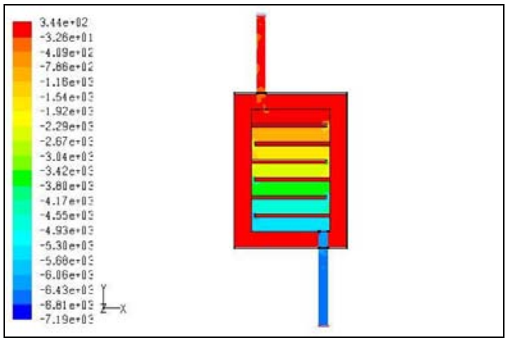

Figure 19 Pressure field distribution diagram of snake-shaped flow path (6 baffles)

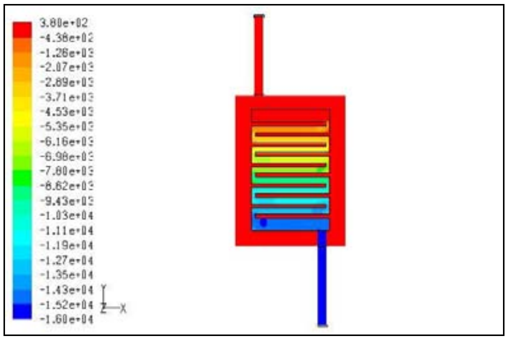

Figure 20 Pressure field distribution diagram of snake-shaped flow path (10 baffles)

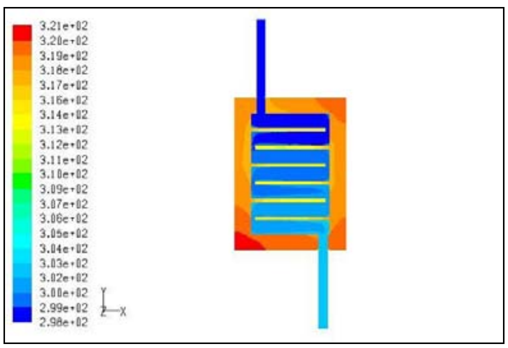

Figure 21 Temperature field distribution diagram of snake-shaped flow path (6 baffles)

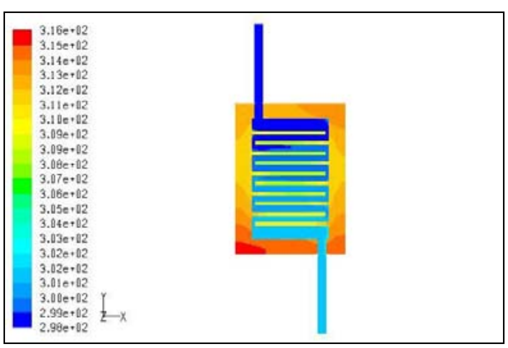

Figure 22 Temperature field distribution diagram of snake-shaped flow path (10 baffles)

SNAKE-SHAPED CYLINDRICAL COMPOSITE FLOW PATH

Regarding the influence of cylindrical flow path and snake-shaped flow path on the heat removal effectiveness of thermal module, placing cylinders in thermal module can increase spiral phenomenon in flow path and increase its heat removal effectiveness, whereas placing baffles in thermal module can stabilize the flow velocity of fluid, increase the time for fluid to stay in flow path, and enhance heat exchange effect. Therefore, the researchers combine these two geometrical flow paths to be a new kind of flow path, which we call snake-shaped cylindrical composite flow path, as shown in Figure 23.

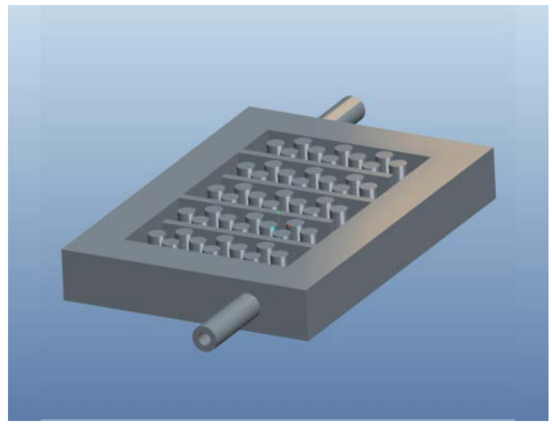
Figure 23 Snake-shaped cylindrical composite flow path

ANALYSIS ON SNAKE-SHAPED CYLINDRICAL COMPOSITE FLOW PATH FIELD MAPS

Figure 24 is the velocity field distribution diagram of snake-shaped cylindrical composite flow path. It is found in the figure that since cylinder is placed at the inlet to narrow the flow path, flow velocity is partially increased. And since baffle is placed at the corner of flow path, the width of flow path is narrowed, producing greater flow velocity, raising the circulation velocity of working fluid, and enhancing the heat removal effectiveness of thermal module.

Figure 25 is the pressure field distribution diagram of snake-shaped cylindrical composite flow path. It is found in the figure that since the flow velocity at the corner of flow path is faster, its pressure is higher. When it is close to the outlet, the pressure is greater. The pressure at the inlet is 145.4 N/m^2, but the pressure at the outlet is 6922.8 N/m^2, with an inlet-outlet differential pressure 6777.4 N/m^2.

Figure 26 is the temperature field distribution diagram of snake-shaped cylindrical composite flow path. It is found in the figure that the temperature distribution in flow path is even, only that the temperature at the place of corner appears to be rather low because of the faster flow velocity there. As to the cylinders in thermal module, they are arranged in up-and-down symmetrical way, making fluid flow past the surface of different cylinders smoothly. Not only the heat on cylinders can be effectively carried away, spiral phenomenon can be more obviously and more regularly distributed in flow path. The CPU chip temperature is 39°C. After it is compared to the CPU base plate temperature in Figure 14 and Figure 22, it is found that the CPU chip temperature of snake-shaped cylindrical composite flow path is 14°C lower than that of cylindrical flow path, and 4°C lower than that of snake-shaped flow path. It is proved that the design of snake-shaped cylindrical composite flow path is better than both cylindrical flow path and snake-shaped flow path. And the CPU chip temperature of the former is lower than the latter ones, implying that the heat removal effectiveness of such thermal module is the best among these three modules.

As seen from the viewpoints synthesized above, placing snake-shaped cylindrical composite flow paths in thermal module can stabilize flow velocity and increase circulation velocity of working fluid. Its pressure loss is also less than snake-shaped flow path, and the CPU chip temperature is also lower than that of snake-shaped flow path and cylindrical flow path. After comparison is made among these three flow paths, it is found that snake-shaped cylindrical composite flow path possesses the best heat removal effect. It also reveals that the distribution of geometrical structure in thermal module would influence its heat removal effectiveness.

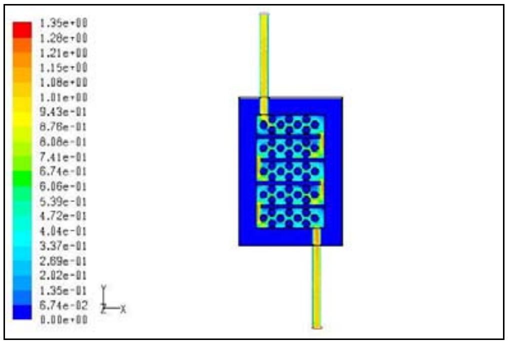

Figure 24 Velocity field distribution diagram of snake-shaped cylindrical composite flow path

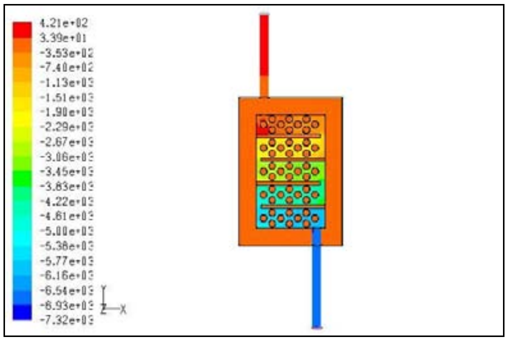

Figure 25 Pressure field distribution diagram of snake-shaped cylindrical composite flow path

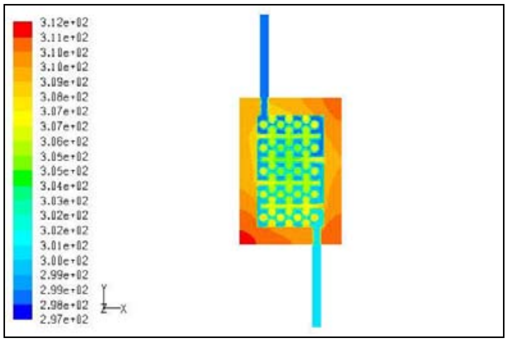

Figure 26 Temperature field distribution diagram of snake-shaped cylindrical composite flow path

CONCLUSION

The study mainly explores the heat removal effectiveness of liquid-cooled thermal module, and analyzes the influence of geometrical design of flow path in thermal module on the heat removal effect of CPU chips. The conclusions drawn are as follows:

1. Simulation results show that snake-shaped cylindrical composite flow path is the best, with heat resistance value being 0.07℃/W; and snake-shaped flow path is the second best, with heat resistance value being 0.14℃/W.
2. Simulation results show that in the aspect of pressure loss, snake-shaped flow path has the greatest pressure loss; snake-shaped cylindrical composite flow path has the second greatest result; and cylindrical flow path has the least result.
3. After making judgment of pressure loss and heat removal effect, snake-shaped cylindrical composite flow path is considered the best design.

REFERENCES

[1]Laszlo B. Kish, End of Moore's law: thermal (noise) death of integration in micro and nano electronics, *Physics Letters A 305* (2002) 144–149。

[2]Qiang Yu, Masaki Shiratori, Fatigue-strength prediction of microelectronics solder joints under thermal cyclic loading, IEEETRANS ON COMPONENTS AND PACKAGINGTECHNOLOGIES, Vol. 20, NO. 3, 1997

[3]Tien-Yu Lee, Design optimization of an integrated liquid-cooledIGBT power module using CFD technique, IEEE TRANS ONCOMPONENTS AND PACKAGING TECHNOLOGIES, Vol. 23, No. 1, 2000.

[4]Bret A. Zahn, Evaluating thermal characterization accuracy using CFD codes-A Package Level Benchmark Study of IcePak and Flotherm, IEEE , 1998.

[5] Kevin A. Moores, Yogendra K. Joshi, Gerhard H. Schiroky, Thermal characterization of a liquid cooled AlSiC base plate with integral pin fins, IEEE TRANS ON COMPONENTS AND PACKAGING TECHNOLOGIES, Vol. 24, No. 2, 2001.

[6] Ryu J.H., Choi D.H., Kim S.J., Numerical Optimization of the Thermal Performance of a Microchannel Heat Sink, International Journal of Heat and Mass Transfer, Vol.45, pp.2823-2827, 2002.

[7]Yingjun Cheng, Gaowei Xu, Dapeng Zhu, Wenjie Zhu, Le Luo, Thermal analysis for indirect liquid cooled multichip module using 113 computational fluid dynamic simulation and response surface methodology, IEEE TRANS ON COMPONENTS AND PACKAGING TECHNOLOGIES, Vol. 29, No. 1, 2006.

[8]H.Y. Zhang, D. Pinjala and Poi-Siong Teo, Thermal management of high power dissipation electronic packages from air cooling to liquid cooling, IEEE 2003.

[9]Lytle D, Webb B. W. Air jet impingement heat transfer at low nozzle-plate spacing, Int. J. Heat Mass Transfer , Vol. 37 , No. 12 , 1994, pp. 1687-1697.

[10]T.E. Salem, D. Porschet, S.B. Bayne, Thermal performance of water-cooled heat sinks a comparison of two different designs, IEEE SEMI-THERM Symposium, 2005.

[11]J.G. Maveety, H.H. Jung, Design of an Optimal Pin-fin Heat Sink with Air Impingement Cooling, International Communications in Heat and Mass Transfer, Vol. 27, pp. 229-240, 2000

[12]Shaukatullah, H., Storr, W.R. , Hansen, B.J. and Gaynes, M.A., Design and Optimization of Pin Fin Heat Sink for Low Velocity Application, Twelth IEEE SEMI-THERM Symposium, 1996, pp.151-163.

[13]Wirtz, R.A., Sohal, R. and Wang, H., Thermal Performance of Pin-Fin Fan-Sink Assemblies, Journal of Electronic Packaging, Vol. 119, 1997, pp.26-31.