Numerical simulation of conjugate heat transfer in electronic cooling and analysis based on field synergy principle

Y.P. Cheng*, T.S. Lee, H.T. Low

Laboratory of Fluid Mechanics, Department of Mechanical Engineering, National University of Singapore, Singapore 119260, Singapore

Received 14 December 2006; accepted 13 November 2007
Available online 19 November 2007

Abstract

In this paper, the conjugate heat transfer in electronic cooling is numerically simulated with the newly proposed algorithm CLEARER on collocated grid. Because the solid heat source and substrate are isolated from the boundary, special attention is given to deal with the velocity and temperature in the solid region in the full field computation. The influence of openings on the substrate, heat source height and their distribution along the substrate on the maximum temperature and overall Nusselt number is investigated. The numerical results show that the openings on the substrate can enhance the heat transfer as well as increasing the heat source height, meanwhile, by arranging the heat sources coarsely in the front part and densely in the rear part of the substrate, the thermal performance can also be increased. Then the results are analyzed from the viewpoint of field synergy principle, and it is shown that the heat transfer improvement can all be attributed to the better synergy between the velocity field and temperature field, which may offer some guidance in the design of electronic devices.

© 2007 Elsevier Ltd. All rights reserved.

Keywords: Isolated island; Conjugate heat transfer; Electronic cooling; Field synergy principle

1. Introduction

With the miniaturization of electronic devices and increasing process speed, the high heat flux is often encountered in the electronic industry, which leads to the large temperature rise. According to the survey by the US Air Force [1], more than fifty percents of all electronics failures are related to the temperature. Furthermore, for every 2 °C temperature rise, the reliability of a silicon chip will be decreased by about 10% [2]. Hence, it is crucial to find efficient solution of heat removal from the high flux devices.

Due to the low efficiency of natural convection, forced convection is widely adopted in the electronic cooling. Incopera [3] early provided a comprehensive review on electronic equipment cooling with forced convection. Later Yeh [4] and Sathe and Sammakia [5] reviewed the recent technologies in thermal control and management of electronic equipments. Recently Leung et al. [6] studied the influence of heat source size and their separation between two parallel plates on the local Nusselt number distribution, mean Nusselt number of each block and overall Nusselt number. Furthermore, Young and Vafai [7] investigated the influence of more parameters on the Nusselt numbers, such as heat source height, width, spacing and number, along with the block thermal conductivity, fluid flow rate and heating method.

Besides the influence of conventional geometric parameters, the influence of heat source distribution was also addressed by several researchers. To achieve better thermal performance, by virtue of constructal theory da Silva [8] suggested that the heat sources should be distributed non-uniformly with the smallest distance near the inlet boundary. Liu and Thien [9] also found similar phenomenon and pointed out that the center-to-center distances between two adjacent chips should follow the golden mean (1.618). Chen et al. [10] proved this result by experiment.
and stated that if the geometric ratio is increased beyond 2.0, the thermal performance begins to decrease.

In the electronic devices the solid region usually occupies a large portion of the whole volume, and the conductive heat transfer in the solid and convection heat transfer in the fluid coexist, therefore neither the temperature nor the heat flux at the solid–fluid interfaces can be prescribed as a priori, and available empirical correlations can not be used directly for thermal design and analysis. Amon [11] indicated that such conjugate heat transfer can significantly affect the temperature distribution. Ramadhyani et al. [12] and Sugavanam et al. [13] also reported that through the heat conduction in the substrate, the heat transfer can be enhanced due to the extended surface effect. Hung and Fu [14] and Hung [15] proposed a conceptual design by modifying the attached substrate with openings, and they claimed that by allowing the fluid flow between the upper and lower channels, the flow stagnation between two adjacent obstacles is diminished, hence the cooling capacity of the array of blocks can be enhanced.

Once substrate is arranged with the openings, it will become the ‘isolated island’ problem, which means that solid region is located in the interior and not attached to any wall of the investigated domain, which will increase the difficulty in numerical simulation. Yang and Tao [16,17] and Wang et al. [18] studied this kind of problem both through numerical simulation and flow visualization, and their numerical results agree quite well with the experimental data. However, most of investigation is focused on the problems with simple geometries on staggered grid. In this paper, the fluid flow and heat transfer with complex isolated solid region is tried to be solved on the collocated grid with the newly proposed CLEARER algorithm [19], and special treatment is given on the solid region in the fluid.

From the literature review above, it can be seen that most of previous investigation is focused on the parametric study. By varying one of the influencing parameters while keeping the others unchanged, the resultant friction and thermal performance are obtained. While in this paper, these problems are revisited from a novel viewpoint of the field synergy principle, which can explain the above problem from the essence.

This principle was firstly proposed by Guo and his co-workers [20,21] in 1998 in parabolic convective flow. Later Tao et al. [22] extended it to the elliptic flow when the fluid Peclet number is not too low. The basic idea of the principle is that the heat transfer enhancement is related not only to the velocity field and temperature field, but also to the synergy between them. This paper examines the effects of openings on the substrate, the heat source height and the heat source distribution on the overall thermal performance and the result is explained from the viewpoint of field synergy principle.

2. Physical model and mathematical formulation

The model is simplified from the practical electronic cooling problem, as shown in Fig. 1. Four heat sources made of copper with thermal conductivity 398 W/(m K) are of constant heat flux, and are attached to the substrate, which is made of Teflon with thermal conductivity 0.35 W/(m K). In order to improve the thermal performance, according to Hung’s idea [14,15] the substrate between the adjacent heat sources is slotted. To lower the temperature of the heat sources, the cooling air is forced to flow through the channel from left to the right. The dimensions of some parameters are shown in Table 1. The average
Heat flux density \( q \) \( \text{W/m}^2 \) 1.0
T \( \text{S} \)
Distance between heat sources \( a \) mm 10
Height of channel \( H \) mm 100
Height of heat sources \( h \) mm 1-4
Thickness of substrate \( \delta \) mm 1
Distance between the leading edge \( y \) mm 5
of first heat source and inlet of channel \( S_0 \) mm
Distance between heat sources \( S_1, S_2, S_3 \) mm 7-13

Table 1
Simulation conditions
| Inlet velocity \( U_0 \), m/s | 0.179–1.79 |
| Inlet temperature \( T_0 \), \( ^\circ \text{C} \) | 20 |
| Heat flux density \( q \), W/m\(^2\) | \( 1.0 \times 10^6 \) |
| Length of channel \( L \), mm | 100 |
| Height of channel \( H \), mm | 10 |
| Length of heat sources \( a \), mm | 10 |
| Height of heat sources \( h \), mm | 1-4 |
| Thickness of substrate \( \delta \), mm | 1 |
| Distance between the leading edge \( y \), mm | 5 |

The temperature between the inlet and outlet is taken as characteristic temperature.

The flow is assumed to be two-dimensional steady incompressible laminar flow. Here \( x \) stands for the flow direction and \( y \) stands for the channel height direction. Preliminary study has been conducted to investigate the effect of extension length on the overall Nusselt number. When the extension is increased to two times of the channel length, the variation of Nusselt number is less than 1%, so in order to save the computer resource, the extension length on both sides was taken as half of the channel length, then the uniform velocity distribution can be assigned at the inlet, and the outflow condition is assigned at the outlet. In order to simulate the fluid flow and heat transfer accurately, a non-uniform grid is used in \( x \) direction, with a fine grid in the channel and a coarse grid in the extension region.

The governing equations for continuity, momentum and energy in the fluid region can be expressed as follows:

Continuity equation:
\[
\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0
\]  

Momentum equations:
\[
\frac{\partial (\rho uu)}{\partial x} + \frac{\partial (\rho vv)}{\partial y} = -\frac{\partial P}{\partial x} + \eta \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)  
\]
\[
\frac{\partial (\rho vv)}{\partial x} + \frac{\partial (\rho vv)}{\partial y} = -\frac{\partial P}{\partial y} + \eta \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]

Energy equation:
\[
\frac{\partial (\rho uT)}{\partial x} + \frac{\partial (\rho vT)}{\partial y} = \frac{\lambda_L}{C_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)
\]

In the heat sources:
\[
\lambda_H \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) + q = 0
\]  

In the substrate:
\[
\lambda_S \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) = 0
\]

Because the governing equations are elliptic, boundary condition are required for all boundaries of the computational domain. The required conditions are described as below

(a) In the extension region:
At the inlet \( U_0 = \text{const}, \ v = 0, \ T_0 = \text{const} \)  
At the upper and lower boundaries \( \frac{\partial u}{\partial y} = v = 0, \ \frac{\partial T}{\partial y} = 0 \)  
At the outlet \( \frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial T}{\partial x} = 0 \)

(b) In the channel region:
At the upper and lower boundaries \( u = v = 0, \ \frac{\partial T}{\partial y} = 0 \)  
Velocity in the solid blocks and substrate \( u = v = 0 \)

The elliptic equations are solved by full field computational method. Because the heat conduction in the heat sources and substrate, and convective heat transfer are conjugated with each other, special attention should be paid to the treatment of the solid region. To ensure the zero velocity in the solid region, the following methods can be adopted on the staggered grid [23]. (1) The velocity in the solid region is set zero before every iterative level to remove the influence of velocity in the solid region on that in the fluid; (2) the coefficients of the solved velocity in the solid region are set to be a large value before the discretized momentum equations are solved and (3) the coefficients of the velocity correction equations in the solid region are set to be a very small value to ensure the velocity correction there is also zero. With the above methods, the isolated solid island problem on the staggered grid can be easily solved.
However, on the collocated grid those methods are not applicable due to different variable arrangements. In this paper, the method proposed by Qu et al. [24] are adopted, which are (1) the velocities on the nodes are assigned to zero during the whole iteration process; (2) the velocity on the solid–fluid interface should also be kept zero during the whole iteration process; (3) because the pressure in the solid region is meaningless, they should not affect the pressure field in the fluid. When the pressure term is calculated in the momentum equation of the fluid adjacent to the solid, the interface pressure is obtained by interpolation from the fluid side. It is notable that the coefficients of the velocity correction equations of the grid point in the solid region should be set to zero after the pressure correction is solved instead of before it, otherwise the iterative process will diverge.

To guarantee the continuity of the heat flux rate at the solid–fluid interface, the thermal conductivity of the fluid, heat sources and substrate take the values according to their different physical property, i.e. the thermal conductivity of the fluid, heat sources and substrate are taken from the values of air, copper and Teflon, respectively, while the heat capacity of the solid region takes the value of the fluid [25,26]. Meanwhile, in order to speed up the solution procedure, the newly proposed CLEARER [19] is adopted to deal with the coupling between the pressure and velocity, whose reliability and efficiency have been proved with several benchmark solutions in Ref. [19]. The second-order scheme SGSD [27] is adopted to discretize the convection term to enhance the accuracy. The convergence criterion is that the maximum mass unbalance of the cells divided by the inlet mass flow is less than 1.0 \times 10^{-5}, and criterion for temperature is that the difference of the overall Nusselt number between two successive iterative levels is less than 1.0 \times 10^{-6}.

3. Results and discussion

In the electronic cooling, the maximum temperature has great significance to electronic components; hence in the definition of heat transfer coefficient, the maximum temperature is adopted as follows:

\[ Re = \frac{\rho U_0 H}{\eta} \]

\[ Q = \rho U_0 H C_p (T_{out} - T_{in}) \]

\[ h = \frac{Q}{A(T_{max} - T_{in})} \]

\[ Nu = \frac{h H}{\lambda_k} \]

\[ \theta = \arccos \left( \frac{\vec{U} \cdot \nabla T}{|\vec{U}| |\nabla T|} \right) \]

\[ \bar{\theta} = \frac{\int \int \int \theta \, dv}{\int \int \int \, dv} \]

To evaluate the synergy between the velocity field and temperature field based on the field synergy principle, the local synergy angle is defined in Eqs. (12) and (13) is the average synergy angle in the full field.

The preliminary numerical result shows that the Nusselt number is only related to the geometric configuration, and is independent of the heat flux. Computations are also conducted under three sets of grid systems, 82 \times 52, 142 \times 52 and 142 \times 82. The numerical results show that overall Nusselt number on grid 142 \times 52 is only 0.6% higher than that on grid 142 \times 82, therefore, in the following computation, the grid system 142 \times 52 is adopted.

3.1. Influence of openings on the substrate

It can be seen from Fig. 2 that the openings on the substrate have remarkable influence on the maximum temperature of the heat sources. As we know, with the increase of inlet velocity, the maximum temperature will decrease, but the decrease rate will become mild. However, during the variation range of inlet velocity, the maximum temperature with openings is always lower than that without openings, especially under low inlet velocity. For example, when the inlet velocity \( U_0 = 0.179 \) m/s, the openings can lead to 7 °C temperature decrease compared with that without openings, but at high inlet velocity, the openings can only lead to about 3–4 °C temperature decrease. Anyway, the reliability of electronic devices can be greatly enhanced.

The better thermal performance with openings can also be reflected from Fig. 3. It is shown that the Nusselt number with openings on the substrate is always higher than that without openings, and the improvement is more significant at high Reynolds number. Due to the existence of openings, the Nusselt number can be increased by 10% averagely.

The enhancement of thermal performance can be explained from the traditional viewpoints. In Fig. 4, the flow field and temperature field are plotted at \( U_0 = 1 \) m/s.
i.e. \( Re = 557 \), from which it can be seen that for the flow without openings the recirculation is formed behind each heat source. Due to the stagnation there, the heat generated by the heat sources can only be transported to the cooling fluid by heat conduction instead of convection, hence the heat transfer efficiency is quite low, which can be reflected from the coarse isotherms in Fig. 4. However, when the substrate between the heat sources are slotted, the fluid beyond and below the substrate can flow freely, as seen in Fig. 5 two vortices are formed between the heat blocks, hence the stagnation will be greatly diminished there, the heat can be transported to the cooling fluid by forced convection efficiently. Hence the isotherms in Fig. 5 become denser than that without openings, which will improve the heat transfer from the heat sources to the cooling fluid.

It can also be revisited from a novel viewpoint of field synergy principle. According to this principle the enhancing heat transfer can be attributed to the increased synergy between the velocity field and temperature field, which is indicated with the average synergy angle, as defined in Eq. (13). From Fig. 6 it is shown that with the increasing Reynolds number, the synergy angle also increases, which means the worse synergy between the velocity and temperature field, therefore, the increase rate of Nusselt number with Reynolds number becomes mild, seen in Fig. 3. Meanwhile, it is notable that average synergy angle with openings is always lower than that without openings during the variation range of Reynolds number. i.e. the openings on the substrate can improve the synergy between the velocity and temperature field, which is probably the essence of better thermal performance with openings.

### 3.2. Influence of heat source height

The influence of the heat source height on the thermal performance is investigated in Figs. 7 and 8. Due to the constant heat flux density, the higher heat source means more heat flux generated by the heat sources, hence the maximum temperature at \( h = 4 \text{ mm} \) is much higher than that at \( h = 1 \text{ mm} \), which also indicates that more efficient electronic cooling technology is required for high power electronic devices. Meanwhile, the overall Nusselt number at

---

Fig. 3. Comparison of Nusselt number vs. Reynolds number with and without openings.

Fig. 4. Fluid flow and heat transfer in the channel without openings at \( Re = 557 \).

Fig. 5. Fluid flow and heat transfer in the channel with openings at \( Re = 557 \).
**3.2. Influence of height of heat source (h = 1, 3, 4 mm)**

$h = 4$ mm is always the highest during the variation of Reynolds number, and it is averagely 28% higher than that at $h = 1$ mm, while Nusselt number at $h = 3$ mm is about 18.5% higher. The much difference among the Nusselt numbers at three heat source heights can be easily explained from the field synergy principle, as shown in Fig. 9. The case at $h = 4$ mm owns the lowest average synergy angle, hence the best synergy between the velocity field and temperature field leads to its best thermal performance. Meanwhile the case at $h = 1$ mm with the highest average synergy angle owns the poorest thermal performance, while the case at $h = 3$ mm is always somewhere in between.

**3.3. Influence of heat source distribution ($S_2/S_1 = S_3/S_2 = 0.7, 1.0, 1.3$)**

The heat source distribution can also exert great influence on the thermal performance. Here the distance between the first two heat sources is kept constant, then the heat source...
distance in the rear are adjusted. When the ratio is 0.7, the heat sources are arranged densely in the front part and coarsely in the rear part of the substrate, while when the ratio is 1.3, more heat sources are arranged in the rear part. From Figs. 10 and 11, it can be seen that better cooling effect can be obtained when the ratio is larger, for example, the overall Nusselt number when the ratio = 1.3 is averagely 18% higher than that when the heat sources are arranged uniformly, which is still 30% higher than that when the ratio is 0.7. Therefore, the traditional uniform distribution is not the optimum choice, meanwhile, it should be avoided that more heat sources are be arranged in the front part, which is in consistent with the current open literature. However, it is difficult to explain this phenomenon from the traditional viewpoint, while the field synergy principle can be adopted to explain it. From Fig. 12 it is noted that under the variation range of Reynolds number, the average synergy angle with large heat source distribution ratio is always the lowest, while that with the small heat source distribution ratio is the highest, which means that the synergy between velocity and temperature field with large ratio is better than that with the small ratio, thus the better thermal performance is obtained.

4. Conclusion

In this paper, the conjugate heat transfer in electronic cooling is investigated on collocated grid with CLEARER algorithm, and special attention is given to the treatment of the isolated solid region for both velocity and temperature. The influence of three geometric parameters on the thermal performance are numerically investigated and analyzed with the field synergy principle, the major conclusion are summarized as follows:

1. The openings on the substrate in the channel can lower the maximum temperature and increase the overall Nusselt number.
2. The increase of heat source height can increase both the maximum temperature and overall Nusselt number.
3. The thermal performance can be greatly improved by arranging the heat sources coarsely in the front and densely in the rear part of the substrate.
4. The heat transfer enhancement can all be attributed to the better synergy between the velocity and temperature field according to the field synergy principle.

References


