A Computational Study of a PLCC Package In Mixed Convection

G. A. Quadir, Ko Yun Hung, and K. N. Seetharamu  
School of Mechanical Engineering,  
USM(KCP), 31750 Tronoh,  
Perak, Malaysia.  
Phone: 605-367-6901 Ext. 5322  
Fax: 605-367-7442  
e-mail: mequadir@kcp.usm.my

Abstract

Miniaturization has led to higher dissipation rates with more stringent restraints on thermal management. Conduction analysis alone (with heat transfer coefficient based on correlations, which are not directly applicable to actual cases) is not adequate to determine the junction temperature and the thermal resistance. A conjugate analysis with the board removes the uncertainty in the specification of the convective heat transfer coefficient. Therefore, a three dimensional analysis of the heat and fluid flow over a single 84 pins PLCC package mounted on a Printed Circuit Board (PCB) along the direction of flow is carried out as a conjugate heat transfer problem using the CFD code FLUENT™ under mixed convection regime. In this simulation, the effects of air flow, leaded versus leadless PLCC, presence and absence of the mother board, and its thermal conductivity on junction temperature and thermal resistance have been investigated. It has been observed from the analysis that the increase in air flow velocity reduces the junction temperature and the thermal resistance. The presence of leads acts as mini heat sinks thus reducing the junction temperature. Mother board acts as a conducting wall and thereby the maximum temperature of the junction is reduced considerably. Increase in the thermal conductivity of the board further reduces the junction temperature. Subsequently, the results obtained are used to understand the package’s thermal performance.

Key words:

PLCC Package, 3D Conjugate Analysis, Mixed Convection, Junction Temperature, Thermal Resistance, and CFD.

1. Introduction and Background

Circuit miniaturization has advanced to such an extent that the heat dissipation capability of a system has become one of the major constraints on device and system design. Chip level heat fluxes of 50-100 W/cm² are projected for the year 2000. Hence, thermal management is perhaps the most important of the environmental consideration since the operating temperature of the chip markedly affects the reliability and availability of a system. The failure rate is cut into half for every decrease of 20 °C. Much of the current research focuses on heat transport by means of convective fluid flow. Forced convection is commonly employed in the cooling of electronic equipment operating at high rates of dissipation. Thermal control typically is maintained by direct air cooling or by indirect liquid or air cooling.

Experimental fluid dynamics have played an important role in validating and delineating the limits of the various approximations to the governing equations. The wind tunnel, for example, as a piece of experimental equipment, provides an effective means of simulating real flows. Traditionally, this has provided a cost-effective alternative to full scale measurement. However, in the design of equipment that depends critically on the flow behavior, full-scale measurement as part of the design process is not economical. Furthermore, a number of trials had to be made before obtaining an acceptable result in experiment. The cost of performing an experiment has increased with time. Mean-
while, the cost of computation is decreasing. This situation has led to an enhanced interest in the development of a numerical wind tunnel. In the analysis of heat transfer, advanced computational tools are essential for rapid virtual prototyping of electronic cooling at scale levels ranging from micro scale to system level.

Tucker and Paul have conducted a review on the application of commercial and noncommercial computational fluid dynamics (CFD) programs to systems relevant to electronics. The following commercial programs are discussed in their paper: THEBES, FLOHERM, FLUENT, FLOTRAN, FIDAP, ICEPAK, CFX 4 (formerly CFDS-FLOW3D), PHOENICS/HOTBOX, and STAR-CD. Notably, of the noncommercial programs, work utilizing a spectral element program originating from the Massachusetts Institute of Technology (MIT) is described. General thermo-fluid capabilities, user friendliness, and other peripheral aspects, such as the modeling of thermal stress/strain and dust transport, are assessed.

Plotnik and Anderson utilized the CFD code FLOHERM to design heat transfer enhancement methods for cooling channels. Their results showed an increase in channel Nusselt number of 10-160% while the friction factor increased by 10-5200%. Jayakanthan et al. have carried out a CFD simulation using commercial CFD code FLUENT on single and two packages along with a Printed Circuit Board in wind tunnel for various flow conditions. The packages are subject to conjugate heat transfer. Conjugate heat transfer involves the coupling of conduction in a solid and convection in a fluid. Where as, Jeffery Low continued Jayakanthan et al. work and have carried out his simulation on multiple chips using a 2D model. Hong and Yuan proved that a constant and uniform heat transfer coefficient across the whole package is inadequate in the accurate prediction of thermal stresses. This is due to the significant effect of local temperature distribution resulted from the variation of local heat transfer coefficient. Thus, Hong and Yuan have demonstrated the importance of considering the conjugate problem for electronic packages. Burgos et al. have performed a parametric study by using the 84-PLCC component with an approach velocity of 0.76 m/s to 3.05 m/s (forced convection). Papanicolaou and Jaluria developed a numerical procedure to simulate the laminar mixed convection mode has been considered for a three dimensional analysis of a PLCC package, where the approach velocity varies from 0.01 m/s to 0.25 m/s.

2. Mathematical Model

Due to the low velocity of air, the flow in the channel is certainly laminar. The three dimensional incompressible steady state laminar flow in a rectangular duct may be represented by the following continuity and momentum equations, equations (1) and (2), respectively.

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]  
\[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p}{\partial x} \]  

Assuming the body is subjected to gravity only,

\[ \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial v}{\partial z} = \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p}{\partial y} + G \]

The energy equation is of the form,

\[ u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{k}{\rho C} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \]

No slip boundary conditions are applied at the stationary solid surfaces. The upper and lower surfaces are insulated.

\[ 0 < x < L, \ y = H, \ u = v = w = 0; \ \frac{\partial T}{\partial y} = 0 \]
\[ 0 < x < L, \ y = 0, \ u = v = w = 0; \ \frac{\partial T}{\partial y} = 0 \]
\[ 0 < x < L, \ z = 0, \ u = v = w = 0; \ \frac{\partial T}{\partial z} = 0 \]
Similarly, no slip boundary conditions are applied on the surface of the Printed Circuit Board and the electronic package,

\[ x = 0, \ 0 < y < H, \ u = U_a; \ T = T_a \]  \hspace{1cm} (10)

\[ x = L, \ 0 < y < H, \int_0^H ud\mathcal{y} = U_aH; \partial T/\partial x = 0 \]  \hspace{1cm} (11)

\[ x = 0, \ z = B; \ T = B \]  \hspace{1cm} (12)

\[ x = L, \ 0 < z < B, \int_0^B udz = U_aB; \partial T/\partial x = 0 \]  \hspace{1cm} (13)

The important variables that will be obtained through this simulation are the maximum temperature in the package, \( T_j \), and the thermal resistance, \( \Theta_{ja} \), given by the following,

\[ \Theta_{ja} = \frac{T_j - T_a}{S} \]  \hspace{1cm} (14)

\( \Theta_{ja} \) includes not only the package internal thermal resistance but also the convective thermal resistance from the package exterior to the ambient. \( \Theta_{ja} \) values depend on material thermal conductivity and package geometry as well as ambient conditions such as flow rates and coolant physical properties.

### 3. Modeling Approach

This problem consists of flow over single electronic package mounted onto a Printed Circuit Board in the flow. Air enters the inlet with a uniform velocity \( U \) and leaves the outlet after picking up heat dissipated by the package. The power generated in the electronic package is specified as volumetric heat source. Thus, 1 W of heat generation is equivalent to 355695.7 W/m³ of volumetric heat since the volume of an 84 pins PLCC package is 2.72 × 2.72 × 0.38 cm³. The surface of the electronic package and the Printed Circuit Board exchange heat with the air by convection. From Jayakanthan et al.⁶, the size of wind tunnel is taken as 10 cm × 52.528 cm for their 2D simulation. In this 3D simulation, the third dimension is chosen as 10 cm. Figure 1 illustrates a 3D drawing (reference coordinates) of the wind tunnel and the mother board with a lumped model of PLCC package attached to it. The distance between the inlet and the package was approximately six times package lengths upstream. Higher density of grid is applied to region near package in order to capture more subtle details. In this simulation, greater attention (higher grid density) has been imposed to region;

- Close to the wall of wind tunnel
- Package
- Mother board.

Total number of node points used is 43,860. The cells are assigned with different boundary conditions as inlet, outlet, wall 1, wall 2, wall 3 and wall 4 as illustrated in Figure 2, whereas the boundary conditions are set as shown in Table 1.

### Table 1. Details of boundary conditions applied.

<table>
<thead>
<tr>
<th>Cells</th>
<th>Boundary Conditions</th>
</tr>
</thead>
</table>
| Wall 1         | • Representing the front, rear, top and bottom walls of the wind tunnel  
|                | • Insulated                                               |
| Wall 2         | • Mother board defined as a conducting wall              |
|                | • Thermal conductivity = 0.76 W/m K                      |
|                | • No heat generation                                     |
| Wall 3         | • Chip of the package defined as a conducting wall       |
|                | • Heat generation varying from 1 W to 3 W                |
| Wall 4         | • Lead section of the package defined as a conducting wall|
|                | • Thermal conductivity = 68.9 W/m K                       |

As expected, the increase in velocity reduces the thermal resistance. This effect is due to the higher rate of heat being transferred or carried away by moving air from the chip. In order to justify this effect of velocity, the simulation has been carried out to investigate the heat transfer coefficient under mixed convection where inlet air velocity is ranging from 0.01 m/s to 0.25 m/s. The thermal conductivity value of PLCC package is varied from 0.50 W/m K to 0.67 W/m K. Meanwhile, the effect of lead existence is also investigated to know the role of lead in heat transfer from the packaging. Three cases have been carried out to understand the effect of heat generation in the package where the heat generated is increased from 1 W to 2 W and finally to 3 W. The effect of mother board as a conducting wall (heat sink) is studied by attaching the package with it.
4. Results and Discussion

The results are obtained solving the fluid flow problem with heat transfer, following the mathematical approach and the modeling setup discussed in the previous section. The first step in any CFD is the validation of the methodology used in the prediction. This validation has been achieved by Burgos et al.10 have performed a case study for plastic packages where the approach velocity changes from 0.76 m/s to 3.05 m/s. Their work also showed that the numerical prediction has a good agreement with measurement with an average error of 6.5% (over-prediction).

The temperature variation along a horizontal plane corresponding to the top surface of the package, under a condition of mixed convection with an air flow at 0.01 m/s, is shown in Figure 4. In this Figure, AB shows the air temperature approaching the package; BC shows the variation of the package surface temperature and CD shows the temperature of the air leaving the package. A nonsymmetrical temperature distribution is observed with respect to the central line of the modeled package. Before the air enters the chip portion, it remains constant at inlet air temperature (298 K). Surface temperature increases up to a maximum (436.4 K) as the air moves over the chip region. As the air leaves the chip portion, the temperature decreases slowly until it became almost constant at 313.5 K. Figure 5 shows the temperature distribution along the surface of the package(portion BC) for different air velocities. It can be observed that the package surface is neither isothermal nor symmetrical with reference to the centre line of the package indicating the effect of the mixed convection even for the low air flow velocity of 0.01 m/s. Further, a significant reduction in maximum temperature is observed as the velocity of air is increased. This is due to the greater amount of heat being carried away by the faster moving air. The boundary layer also gets contracted as the velocity of air increases as evidenced from Figures 6 and 7.

Figure 3. The validation based on the theoretical results for the case of laminar flow over heated isothermal plate.6

Figure 4. Surface Temperature (K) of the package versus length (m) in x-direction for air velocity of 0.01 m/s.

Figure 5. A comparison of temperature distribution along the surface of the package at different air velocities under mixed convection.

Figure 6. Temperature profile of single PLCC package at air velocity of 0.01 m/s.
The heat flux variation is presented in Figure 8 for different air flow velocities. It is noticed that the front portion of the chip is cooled faster than other places as an expected characteristic of forced convection cooling at the leading edge. The value of heat flux at the leading edge of the package is 662.3 W/m² compared to its value of 345.0 W/m² at the trailing edge of the package for an air velocity of 0.01 m/s. Hence, the heat flux on the surface of the package (BC) is not uniform. It is also observed from the Figure 8 that heat flux increases as the velocity of air is increased. However, the maximum is attained at the same location of the package at different air flow velocities.

Figure 8. A comparison of heat flux distribution along the surface of the package at different air velocities under mixed convection.

Table 2. Effect of air velocity variation on average heat transfer coefficient and thermal resistance.

<table>
<thead>
<tr>
<th>S. No.</th>
<th>Air Flow (m/s)</th>
<th>Max Temperature (K)</th>
<th>Average Heat Transfer Coefficient (W/m²K)</th>
<th>( \theta_{ja} = \frac{T_j - T_a}{S} ) K/W</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.01</td>
<td>436.4</td>
<td>3.082</td>
<td>138.4</td>
</tr>
<tr>
<td>2</td>
<td>0.10</td>
<td>370.1</td>
<td>9.247</td>
<td>72.1</td>
</tr>
<tr>
<td>3</td>
<td>0.25</td>
<td>351.8</td>
<td>12.667</td>
<td>53.8</td>
</tr>
<tr>
<td>4</td>
<td>0.5</td>
<td>340.2</td>
<td>14.47</td>
<td>42.2</td>
</tr>
<tr>
<td>5</td>
<td>1.52</td>
<td>325.5</td>
<td>15.87</td>
<td>27.5</td>
</tr>
<tr>
<td>6</td>
<td>3.05</td>
<td>321.9</td>
<td>16.30</td>
<td>23.9</td>
</tr>
</tbody>
</table>

The predicted heat transfer coefficients agree reasonably well with those of Hong and Yuan⁹. At air velocity of 0.5 m/s, the average heat transfer coefficient in the present case is 14.47 W/m²K compared to the value given as 14.9 W/m²K. At low velocity (0.01 to 0.10 m/s), the temperature distribution is asymmetrical (skew towards the trailing edge) implying that the maximum temperature is not attained near the middle of the package. This shows that the cooling is mainly controlled by a mixed convection mode at very low velocity of air flow. Burgos et. al.¹⁰ have neglected this gravitational effect since their test were well beyond the mixed convection regime.

Natural convection mode affords a means of thermal control that eliminates the use of fan or pump. It also provides a noise- and vibration-free environment. However, its relatively low convective heat transfer coefficient makes it only suitable for small heat-source component (computer chips) mounted on a substrate (Printed Circuit Board); where the conduction through the substrate is substantially significant.
The thermal resistance drops significantly with the increase in air velocity as shown in Table 2 where additional results are taken from Hung. When the velocity is increased, the hottest region shifts towards the rear portion of chip. This is due to higher rate of the cooling effect at the leading edge. However, this effect reduces substantially where the velocities are greater than 2.29 m/s as evidenced from Figure 8. When velocity is increased 2 times from 0.25 m/s to 0.50 m/s, \( \theta_{ja} \) reduced from 53.8 K/W to 42.2 K/W, a reduction of 21.6%. In the case of higher velocity, the increment from 1.52 m/s to 3.05 m/s, it results in a reduction of 13.1% in \( \theta_{ja} \). This suggests that, given an arrangement of electronic devices on a PCB, there exists a particular air speed whereby any further increase will not provide an economic cooling. Therefore, in designing a cooling system by means of airflow, it is not necessary to have a bigger fan or blower with higher fluid flow capacity, as this will only incur additional cost. Moreover, the bigger fan may contribute higher heat source to the particular electronic system as well as noise to the surroundings. Apart from this, the space used by these gadgets can be utilized for other purpose.

It has been observed during simulation that the junction temperature increases with the ambient temperature to the same extent. However, the thermal resistance remains constant as expected. The average heat transfer coefficient remains sensibly constant at 15.15 W/m²K for the variation of ambient temperature from 288 K to 318 K. In the above cases, \( \theta_{ja} \) remains almost constant around 35.5 K/W. Similar trend is observed with the variation of chip power also.

The important indication from this study is that cooling atmospheric air prior to entering the electronic system will provide better cooling to the devices compared to using the uncooled atmospheric air. Higher ambient temperature will cause a higher failure rate of the package. Hence, it is important in the initial design of electronic devices, the operating temperature range be clearly identified. The precooling of air can always act as a safety factor in electronic package system design if it is possible. Usually, the maximum operating temperature of package is designed to be lower than 100 °C in order to avoid the moisture turning into vapor and lead to popcorn failure especially during reflow where the maximum temperature reached is 260 °C for a short duration of about 6 seconds at most.

For the lower value of thermal conductivity of PCB, the maximum temperature in the package increases. PCBs made of higher thermal conductivity value are capable of dissipating heat more efficiently. With a reduction of 25.4% in thermal conductivity of the board, the maximum temperature has increased from 333.5 K to 334.9 K, whereas, the thermal resistance \( \theta_{ja} \) increases from 35.5 K/W to 36.9 K/W.

The simulation results also indicated that leads play a role in dissipating heat from the package. In this study, the leads have reduced the junction to ambient thermal resistance about 2.2% from 36.3 K/W to 35.5 K/W. The leads decrease the maximum temperature in the package from 334.3 K to 333.5 K. Figure 10 shows the top surface temperature distribution of the package with and without leads. The maximum surface temperature reduces, as evidenced from Figure 11, since the leads provide additional area for conducting heat from the package and subsequent convection of the heat to the environment. The above comparison is made for an air velocity of 0.76 m/s. However, in the mixed convection regime, the effect of lead will be more pronounced.

Through this simulation, one can notice that the board plays an important role in reducing the maximum temperature and the thermal resistance. When the package is attached to the board, it will act like a heat sink and dissipates more heat from the package. Thus, it is appropriate to categorize it as a conducting wall in the FLUENT™ modeling. Heat is dissipated through the multiple layers of mother board; especially in the through hole mounting where the lead pins are soldered through multiple layer that exist in the board. This multiple layer is made of conducting material. From Table 3, the package attached to the mother board shows a reduction of 21.1% of junction to ambient thermal resistance, consequent to the reduction in the peak temperature from 333.5 K to 326.0 K.

Table 3. Effect of mother board on thermal response of the
package.

<table>
<thead>
<tr>
<th>S. No</th>
<th>Board Attached</th>
<th>Max Temperature (K)</th>
<th>Average Heat Transfer Coefficient (W/m² K)</th>
<th>$\frac{\theta_{ja}}{S}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Yes</td>
<td>326.0</td>
<td>22.539</td>
<td>28.0</td>
</tr>
<tr>
<td>2</td>
<td>No</td>
<td>3.5</td>
<td>15.157</td>
<td>33.5</td>
</tr>
</tbody>
</table>

### 5. Conclusions

A three dimensional conjugate analysis of heat and fluid flow over a single 84 Pins PLCC package mounted on a Printed Circuit Board is carried out. The conjugate analysis(conduction-convection) removes the uncertainties associated with the specification of surface heat transfer coefficients for a conduction type of analysis. The analysis is carried out in a mixed convection regime associated with low velocity of air flows combined with free convection using a CFD code. The effects of air flow velocities, package leads, presence of board, and its thermal conductivities on junction temperature and thermal resistance have been determined. From the analysis, it is found that the presence of board and increase in its thermal conductivities lead to a significant reduction in junction temperature and thermal resistance. The thermal resistance of the package is observed to be independent of ambient temperature and chip power.

In practice, several packages are mounted on a Printed Circuit Board. The packages may be mounted along the flow or across the flow. Hence, it is worthwhile, in future, to investigate the effect of such distribution of packages mounted on the board on the junction temperature and thermal resistance for various flow and other parameters.

### Nomenclature

- $B$ = Width in z-direction, m
- $C$ = Specific heat capacity, J/kg °C
- $G$ = Body force in y direction, N
- $H$ = Height in y-direction, m
- $k$ = Thermal conductivity, W/m K
- $L$ = Length in x-direction, m
- $p$ = Pressure, N/m²
- $S$ = Device power dissipation, W
- $T_a$ = Ambient temperature, K
- $T_j$ = Junction temperature of the chip, K
- $U_a$ = Air velocity, m/s
- $u$ = Component of velocity along x-direction, m/s
- $v$ = Component of velocity along y-direction, m/s
- $w$ = Component of velocity along z-direction, m/s
- $\rho$ = Density, kg/m³
- $\nu$ = Kinematic viscosity, m²/s
- $\theta_{ja}$ = Junction to ambient thermal resistance, K/W

### References

12. C. P. Tso, G. P. Xu, and K. W. Tou, “An Experimental Study on Forced Convection Heat Transfer from flush-Mounted dis-
A Computational Study of a PLCC Package In Mixed Convection


About the authors

G.A. Quadir obtained his first Degree in Mechanical Engineering in 1966 from Bhagalpur University, India. He joined Aligarh Muslim University (AMU), Aligarh as a lecturer in 1968. He completed his Ph.D. Degree from Indian Institute of Science (IISc), Bangalore, India in the year 1979. He became Reader in AMU, Aligarh in 1979. He went to Algeria on a teaching assignment and served there for six years (1980-86) in the University of Annaba. Later, he served in Libya for three years (1986-89) in the Higher Institute of Mechanical and Electrical Engineering, Hoon. Before joining his present faculty position in the University of Science, Malaysia, he was Reader in the Faculty of Engineering & Technology, Jamia Millia Islamia, New Delhi, India (1990-93). His field of interest is in Fluid Mechanics, Turbomachines, and CFD. His current research involves the analysis of different types of heat exchangers including the ones used in electronic package cooling using FEM.

Ko Yun Hung received the B. Eng. Degree in Mechanical Engineering in 1999 from University of Science Malaysia. His major interests include research in heat transfers analysis, application of computational fluid dynamics, design- ing and modelings. Currently, he is working in an engineering consultation firm, SNS Network.

K. N. Seetharamu, Professor, received his B. E. Degree in 1960; his M. E. Degree in 1962 from IISc., Bangalore, and his Ph. D. Degree in 1973 from IIT, Madras. His is actively engaged in the areas of fluid flow, heat transfer, and FEM simulation. At present, he is engaged in the Thermomechanical Analysis of Electronic Packages. He worked as a Professor of Mechanical Engineering at IIT, Madras, for almost three decades before joining USM, Malaysia. He has published more than 200 papers in International Journals and Conferences. He is one of the authors of the book Finite Element Method in Heat Transfer Analysis published by John-Wiley. He has guided more than 20 Ph. D. students during his Academic career. He is the secretary/treasurer of IMAPS’s Malaysia Chapter.