THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E. 47 St., New York, N.Y. 10017

91-WA-EEP-36

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. Discussion is printed only if the paper is published in an ASME Journal. Papers are available from ASME for fifteen months after the meeting.
Printed in USA.

Depth-Averaged Modeling of Convective Heat Transfer from Printed Circuit Boards

GERALD W. RECKTENWALD
Mechanical Engineering Department
Portland State University
Portland, OR 97207

PETER BUTLER
Intel Corporation
Hillsboro, OR 97214

Abstract

We describe a new approach to using computational fluid dynamics (CFD) in the modeling of convectively cooled printed circuit boards (PCB's). The model is based on a two-dimensional, depth-averaged approximation to the flow field above the devices on the PCB. The depth-averaged flow equations are obtained by integrating the three-dimensional continuity, momentum and energy equations over the direction normal to the plane of the PCB. This approach, which renders the entire PCB in a plan view, is in contrast to other two-dimensional models that analyze the flow in an elevation view. The depth-averaged model provides a more detailed, and a less problem dependent, analysis than lumped parameter models because it computes the flow field and the temperature field of the coolant over the devices. A depth-averaged model also has significantly smaller computing requirements (both run-time and memory) than a fully three-dimensional CFD model of the flow over the PCB. The goal of this research is to develop a tool that will allow packaging engineers to interactively analyze the thermal characteristics of different board layouts at the same time that the electrical and electronics engineers are selecting the devices to be put on the board. Eventually, such an approach could be incorporated into CAD packages.

In this paper we describe the theory of the depth-averaged model and we identify the strengths and weakness of this approach. Several theoretical and practical issues need to be resolved before this strategy can be applied with confidence to the analysis of electronic cooling problems. As a progress report on this research we present preliminary results obtained by simulating related experimental studies in the laminar flow regime.

Introduction

Motivation: an improved design tool

It is well known that thermal analysis and design of enclosures for electronic equipment is becoming more critical as the power dissipation of electronic components and the density of components on boards continues to increase. The importance of good design is also made more critical by ever-shortening product cycles. The goal of the current research is to provide a computational tool for thermal analysis and design of printed circuit boards (PCB'S) with discrete, heat dissipating components.

The need for continued experimental research is obvious, but so is the difficulty of providing specific experimental data for the wide variety of unique component layouts, board orientations, and coolant flow paths found in enclosures. Modeling methodologies are needed to allow designers and packaging engineers to assess the impact of the placement of individual devices with critical cooling needs. An ideal design tool would integrate with existing computer-aided design (CAD) tools, it would allow interactive analysis on an engineering workstation, and, of course, it would be sufficiently accurate to enable the designer to correctly choose the better alternative of two competing design solutions. The integration with existing CAD tools is a job best left to the software engineers. The current research is aimed at developing the underlying methodology. Our ultimate goal is to develop a depth-averaged model that will perform a complete thermal analysis on a high-performance workstation in half an hour. The accuracy of the methodology will only be known after extensive testing and model improvements.

One strategy for thermal analysis of convectively cooled PCB's is to use a CFD code to solve the entire three-dimensional velocity, pressure and temperature fields surrounding the devices on the board. Although this is technically feasible it is not currently viable as a routine design methodology, at least not as an *interactive* design tool. By depth-averaging the flow equations we reduce the fluid dynamic problem from three to two dimensions. The two dimensions in question are in the plane of the PCB, and therefore parallel to the main flow directions of the coolant. This reduces the computational effort at the expense of the accuracy and detail in the results. An essential goal of the current research is to determine whether depth-averaging is sufficiently accurate that it can be used in routine analysis of PCB's.

In this paper we present an overview of the theory behind the depth-averaged model, we discuss the strengths and weaknesses of this approach, and we show preliminary results obtained with the model in its current state of development. An important limitation of the current code is that prediction of component surface temperatures still requires knowledge of local heat transfer coefficients. To eliminate this constraint we are working to couple the depth-averaged

Presented at the ASME Winter Annual Meeting December 1-6, 1991 Atlanta, GA

flow equations with solution to the three-dimensional energy equation. This future development is discussed briefly in a subsequent section of this paper. The current version of our depth-averaged model is also limited to laminar flow problems. In the future we will extend the model so that it can simulate turbulent flows.

Flow over PCB's cooled by forced convection

Much research has been conducted on heat transfer from PCB's and simplified models of PCB's. Our goal with the following rather cursory review is to identify the features of the flow and heat transfer that should be addressed by any reasonable design tool.

Several experimental studies have been conducted on simplified models of PCB's. Typically an array of block or rib elements is placed on one wall of a duct that is much longer and wider than it is tall. One or more of these elements are heated and the effect of varying the geometry and flow rate is reported. When, instead of heating the blocks, the naphthalene sublimation technique is used, only one of the blocks participates in the mass transfer.

Experimental studies of flow past uniform arrays of blocks or rows of ribs aligned across the direction of flow have shown that the flow in these geometries becomes rather quickly fully developed [10, 15, 21, 29]. Thus, for a densely populated circuit board, one would expect that the velocity profile at the inlet would not strongly affect the heat transfer across a large fraction of the board's surface area.

Practical circuit boards are populated with components of different lateral dimensions, heights and shapes. Each disruption to an otherwise uniform array of components creates wake effects on the downstream components. Sparrow and his co-workers [29-31] used the heat/mass transfer analogy to investigate the effect of irregularities such as missing blocks and barriers protruding above the height of an array of blocks. They found that the heat/mass transfer coefficient of an individual block in the vicinity of an irregularity could be as much as a factor of two above the corresponding heat/mass transfer coefficient for a uniform array of blocks. Flow separation and reattachment are the mechanism responsible for the enhanced heat/mass transfer coefficients.

Details of the local variations in flow and heat transfer over individual heated blocks have been investigated numerically and experimentally. In their computational study of periodically fully-developed, laminar flow past cubes, Asako and Faghri [3] report weak variation in dimensionless temperatures and Nusselt numbers except near corners. Olsen et al. [22] studied heat transfer from an isolated cube in a wind tunnel. They document significant three-dimensional effects in their quantitative measurements and with flow visualization. At the highest Reynolds numbers (22200, based on the length of the cube side) the local Nusselt number varied by a factor of two over the face of the cube normal to the oncoming stream. The importance of these three-dimensional effects increased with Reynolds number

The flow above and between the components on an air-cooled PCB is typically unsteady and/or turbulent. Using flow visualization Lehmann and Wirtz [16] found that the unsteadiness of the flow increased as the spacing between rectangular ribs increased. McEntire and Webb [19] determined that at a fixed streamwise spacing between ribs and streamwise length of rib, the flow becomes turbulent at a Reynolds number of 1500. This result was apparently independent of the rib height. The Reynolds number was based on the average velocity above the ribs and on the spacing between the top of the rib and the opposing wall of the duct. A transition to turbulence is evident in the average Nusselt number correlations of Lehmann and Pembroke [15]. Sparrow et al. [29] conclude that the flow is turbulent in all of their experiments for Reynolds numbers between 2000 and 7000. From this we conclude that any computational model or design tool must account for the transitional and turbulent nature of the flow.

At low velocities buoyancy effects become important. Unless the predominate direction of the forced convection is vertical the buoyancy drives important secondary flows [4, 18, 28, 34]. As it is currently conceived, our depth-averaged model should not be applied to flows with significant mixed convection effects. It is possible to apply depth-averaging to free convection past vertically oriented PCB's.

If the depth-averaged strategy is to succeed as a design analysis tool all of the above physical features of the flow will have to be represented. Our current work is very much in the exploratory states. Rather than trying to model all these flow complications we are now testing the code on laminar flow past regular arrays.

Modeling Strategies

One approach to the analysis of PCB's is to create a thermal network, finite element, or finite difference model of the heat conduction within the board and the components mounted on it [5, 9, 12]. This technique requires knowledge of the heat transfer coefficient between the component and the coolant. The shortage of data on local heat transfer coefficients for the complex geometries found in electronic cooling applications is the primary weakness of this approach.

Another strategy in the analysis of PCB's is to simulate the flow of coolant around the components with a CFD model. Ducts with flush mounted heaters have been simulated [13] as have ducts with two-dimensional ribs transverse to the main flow [2, 8, 34]. In these studies the flow equations are solved in a two-dimensional plane in an elevation view of the components . This approach is severely limited as a design tool because variations across the PCB (normal to the plane of the elevation view) are not allowed, and to date these models include only a small number of components.

Infinitely long arrays of heated modules can be simulated by assuming that the flow is periodically fully-developed. Examples of these calculations are given by Patankar et al. [24], Schmidt and Patankar [27] and Asako and Faghri [3]. The work of Asako and Faghri is especially relevant because they simulated a three-dimensional array of blocks representing components on a PCB.

Because of the computational effort required few fully three-dimensional CFD simulations of PCB's have been attempted. The aforementioned work by Asako and Faghri invoked geometrically simplifying assumptions to limit the problem size. Afrid and Zebib [1] simulated three-dimensional laminar and turbulent natural convection cooling of 10 heated cubes. Linton and Agonafer [17] simulated the flow inside an entire personal computer cabinet with a commercial CFD code.

To our knowledge, depth-averaging has not been applied to electronic cooling problems. Depth-averaged modeling has been used in the environmental engineering and oceanography fields for many years. Important early work was performed by Rodi and his co-workers [20, 25, 26]. Choudhury [6] developed a depth-averaged model of flow and heat transfer in compact heat exchangers. His work provided the inspiration for the current research. A method of thermal-fluid analysis that is superficially similar to the depth-averaged strategy has been proposed by Weiss et al.[32]. They predict the flow field above the PCB by solving the potential flow equations, which strictly apply to inviscid and irrotational flow. Once the flow field is obtained the local coolant temperature is obtained by a superposition of energy additions along the flow path. Prediction of the case temperature from the known heat flux and the local coolant temperature requires knowledge of the local heat transfer coefficient.

Figure 1. Thin layer of fluid suitable for depth-averaging

Model formulation

Governing equations

The depth-averaged equations are derived from the three-dimensional Navier-Stokes and Energy equations [7] for the flow of a Newtonian fluid with negligible body forces,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j} \tag{2}$$

$$\frac{\partial}{\partial t}(\rho c_p T) + \frac{\partial}{\partial x_j}(\rho u_j c_p T) = \frac{\partial}{\partial x_j}\left[k\frac{\partial T}{\partial x_j}\right] \tag{3}$$

To reduce the three-dimensional flow equations to two-dimensional equations assume that the domain of interest is a relatively thin layer of fluid. An arbitrary representation of such a fluid layer is represented by the sketch in Figure 1. At any position in the x-y plane let $B(x,y)$ be the z location of the bottom of the gap and $L(x,y)$ be the z location of the lid of the gap. The gap thickness, h, is

$$h(x,y) \equiv L(x,y) - B(x,y) \tag{4}$$

The depth-averaged equations are obtained by integrating over the presumably narrow gap in the z direction. In the process one applies the Liebnitz rule for differentiation of an integral [11] which may be rearranged as

$$\int_B^L \frac{\partial f}{\partial x} dz = \frac{\partial}{\partial x}\int_{B(x,y)}^{L(x,y)} f(x,y)\, dz - f(L,y)\frac{\partial L}{\partial x} + f(B,y)\frac{\partial B}{\partial x} \tag{5}$$

The resulting depth-averaged equations are (with $i = 1,2$ only)

$$\frac{\partial}{\partial t}(\rho h) + \frac{\partial}{\partial x_i}(\rho \overline{u}_i h) = 0 \tag{6}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \overline{u}_i h) + \frac{\partial}{\partial x_j}(\rho \overline{u}_j \overline{u}_i h) = & -h\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}(h\overline{\tau}_{ji}) \\ & + (\tau_{3i})_L - (\tau_{3i})_B + h\overline{S}_{u_i} \\ & + \textit{Slope-related flux terms} \\ & + \textit{Dispersion terms} \end{aligned} \tag{7}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \overline{\varphi} h) + \frac{\partial}{\partial x_j}(\rho \overline{u}_j c_p \overline{T} h) = & \frac{\partial}{\partial x_j}\left(h\tilde{k}\frac{\partial \overline{T}}{\partial x_j}\right) \\ & + q_{z,L} - q_{z,B} + h\overline{S}_T \\ & + \textit{Slope-related flux terms} \\ & + \textit{Dispersion terms} \end{aligned} \tag{8}$$

Figure 2 Control volume used to in the depth-averaged model.

where the depth-averaged velocities, shear stresses, source terms and temperature are

$$\overline{u}_i \equiv \frac{1}{h}\int_B^L u_i\, dz \qquad \overline{\tau}_{ji} \equiv \frac{1}{h}\int_B^L \tau_{ji}\, dz$$
$$\overline{S}_\varphi \equiv \frac{1}{h}\int_B^L S_\varphi\, dz \qquad \overline{T} \equiv \frac{1}{h}\int_B^L T\, dz \tag{9}$$

To obtain the depth-averaged diffusion terms on the right hand side of equations (7) and (8) one has to define the depth-averaged thermal conductivity $\tilde{k}$, such that [25]

$$\int_B^L k\frac{\partial T}{\partial x_i}dz = h\tilde{k}\frac{\partial \overline{T}}{\partial x_i} \tag{10}$$

The slope related flux terms and dispersion terms in equations (7) and (8) are neglected in the current version of our code. Neglect of the slope-related flux terms in not entirely justifiable and this simplification will be corrected in the future. When there are discontinuous elevation changes in the lid and bottom profiles it is possible to relate these slope terms to the heat flux and shear stress on the sides of the vertical surfaces (cf. sides of the blocks in Figure 8). By neglecting these terms we are effectively ignoring the shear and heat flux on the sides of these surfaces.

The dispersion terms in equations (7) and (8) arise from moments of the vertical profiles of the dependent variables. The dispersion terms for the x-direction momentum equation are

$$\frac{\partial}{\partial x}\int_B^L \rho(\overline{u}-u)^2 dz - \frac{\partial}{\partial y}\int_B^L \rho(\overline{v}-v)(\overline{u}-u)dz$$

For nearly uniform vertical velocity profiles these terms are small, and it is customary in depth-average models of turbulent flow to neglect the dispersion terms altogether [26].

The depth-averaged model has been implemented in a computer code based on the control-volume finite-difference formulation. Coupling between the momentum and continuity equations is achieved by either the SIMPLE or SIMPLER algorithms [23]. The choice of algorithm is specified by the user at run-time. The discrete algebraic equations used in the computer code are obtained by integrating over the control volume sketched in Figure 2. Due to space limitations the discrete equations incorporated in our code will not be presented here. This documentation will be left to a future report, although those familiar with the control-volume finite-difference method should be able to reproduce these equations following standard practices [23]. Much of the relevant formulation is presented by Rodi et al.[26] and Choudhury [6]

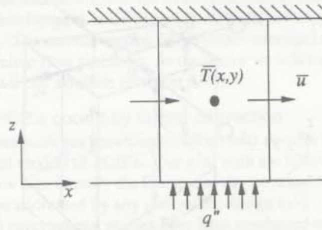

Figure 3 Elevation view of a typical depth-averaged control volume with a constant heat flux boundary condition on the bottom and an adiabatic surface on the lid.

The gap profile $h(x,y)$ is specified at each computational node and is assumed to be uniform over each control volume. Thus, whenever there is a change in gap height, there is a discontinuous change in h at the boundary between adjacent control volumes. We chose discontinuous gap height variation instead of the continuous gap variation as suggested by Figure 2 because it leads to simpler data structures and because the geometry of PCB's involves step changes in component height. Step changes in gap height means that the area of adjacent control volume faces is not equal. In computing the flow of conserved variables from one control volume to the next the flux is continuous and the flow area is the minimum of the flow area of the two adjacent control volume faces. In this way we maintain the conservative properties of the control-volume finite-difference formulation.

Normal flux on bottom and lid surfaces

Flows of the conserved variables through the bottom and lid surfaces appear as the normal flux terms in equations (7) and (8). These flux terms must be expressed as functions of the depth-averaged variables in the x-y plane.

First consider the x-direction momentum equation. Suppose that the depth-averaged model is used to compute the flow between infinite parallel plates. The shear stress at the bottom and lid plates is obtained by multiplying the velocity gradient at the plate by the fluid viscosity. Evaluating this stress from the fully-developed velocity profile yields [33]

$$\tau_{zx} = \frac{6\mu\overline{u}}{h} \tag{11}$$

Since the shear stress is determined by the velocity gradient at the wall, evaluation of the unknown shear stresses in equation (7) requires an assumption for the shape of the velocity profiles. Ultimately the shear stresses must be expressible as functions of the depth-averaged velocities. For the results presented below the fully-developed velocity profile for laminar flow between infinite parallel plates is used. Models for turbulent depth-averaged flow are available [26].

Evaluation of the bottom and lid heat flux depends on the type of boundary condition imposed on the bottom and lid surface. Consider the elevation view of the depth-averaged control volume represented by Figure 3. If the heat flux on the bottom or lid is prescribed, this known value is substituted directly into the discrete form of equation (8). If the surface temperature is prescribed one needs to specify a heat transfer coefficient to relate the heat flux to the difference between the surface temperature and the depth-averaged fluid temperature. If the flow can be considered to be thermally fully-developed, this heat transfer coefficient is obtainable from the exact temperature profile [6]. Since it is unlikely that flow over PCB's is ever thermally fully-developed, practical application of the depth-averaged energy equation will require an empirical heat transfer coefficient.

At this point the reader may well wonder why go to all this trouble of solving the depth-averaged flow equations if the thermal analysis still relies on an empirical heat transfer correlation. Indeed the added computational work may seem like an inefficient way to achieve the same results obtainable by a heat conduction analysis of the PCB and its attached electronic components. There are, however, at least two reasons for pursuing the depth-averaged analysis. First, the depth-averaged flow equations do provide a means of predicting the local fluid velocities above the electronic components. This will likely yield a more reasonable basis for computing the heat transfer coefficient. It will also allow proper accounting for the thermal wakes of upstream components.

The second reason for pursuing the depth-averaged flow calculations is that the energy equation need not be solved in its depth-averaged form. As long as bulk vertical mixing of the fluid does not dominate the flow, the depth-averaged momentum and continuity equations will do a reasonable job of representing the flow over the PCB. If bulk vertical mixing is relatively weak a conduction analysis in the z-direction will eliminate the need to specify a heat transfer coefficient. In a turbulent flow the enhanced vertical transport due to turbulent eddies can be simulated with a turbulence model.

Conduction in the vertical direction and convection in the x-y plane can be conveniently combined into solution of the three-dimensional energy equation. Vertical profiles of the x and y velocity components would be specified in terms of the depth-averaged velocities and the profiles used to compute the lid and bottom shear stress (cf. equation 11). Though this has yet to be implemented in the code, our plan is to do so in the near future. An added benefit of this strategy is that the conduction analysis can be extended into the PCB and power dissipating components. Thus, conjugate heat transfer analysis can be seamlessly integrated into the model.

In forced convection of a fluid with uniform properties the solution of the flow equations (1) and (2) or (6) and (7) account for nearly all the computational work of the control-volume finite difference strategy. The energy equation in this type of flow is linear and it is solved after the flow equations have converged. Solving the three-dimensional energy equation will certainly increase the execution time of the model, but efficient elliptic solvers such as multigrid can be used to keep the added work to a minimum.

Summary of Limitations

The depth-averaged approach is less rigorous than more conventional CFD strategies. The aim is not to replace detailed flow simulation using CFD or comprehensive experimental investigations. Rather we are proposing a methodology that may some day be used as a design tool. In an effort to promote discussion of our strategy we briefly summarize the fundamental limitations of the model. We have taken the tack of empirically studying the results of the model over the more conservative approach of rigorously proving the theory before writing any code.

In the current implementation of the depth-averaged model we have simply ignored the effect of discontinuities on the top and bottom surfaces. This is risky on at least two counts, one mathematical and one physical. The mathematical concern is that we have ignored any complications due to these discontinuities in the process of transforming the governing equations. In the derivation we have presumed that the bottom and top surface elevations vary continuously with x and y. The discrete equations used in the code *do* account for discontinuous changes in flow area of the control volume faces so that mass and energy are conserved exactly.

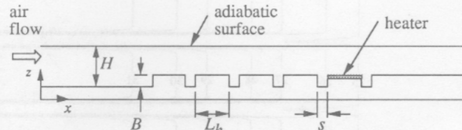

Figure 4 Numerical model of Lehmann and Wirtz's [16] experiment.

The obvious physical flaw in neglecting discontinuities is that step changes are bound to lead to recirculation flow patterns with vertical velocity components. Furthermore such recirculation zones invalidate the profile assumption used to compute the shear stress as a function of depth-averaged velocity. Whenever such recirculation patterns occupy major parts of the flow field the model is expected to fare poorly. We expect, however, that thermal analysis of high density circuit boards with closely spaced components of approximately equal height will be relatively insensitive to these recirculation zones.

In addition to the interaction of flow above the devices and flow between and in the plane of the devices, there is a vertical variation of the heat transfer rate over the sides of the components. We suspect that neglecting these variations will not produce large errors because the non-uniformities are confined to relatively small surface areas. Of course, non-uniformities are problem-dependent.

Experimental observations demonstrate that the flow above the components is unsteady at some flow rates, e.g. [16, 19]. Currently we are restricting our analysis to steady flow. Although it is possible to perform unsteady depth-averaged simulations, the depth-averaged equations may not admit the proper unsteady solutions regardless of any numerical accuracy limitations.

Preliminary Results

A series of test problems has been chosen for code validation. Here we report preliminary results of simulating three experiments on forced convection electronic cooling [10, 13, 16].

Simulation of the first experiment is not a test of the depth-average code, but rather of the depth-average concept. We used a two-dimensional SIMPLER-based code (not our depth-averaged code) to simulate the experimental research of Lehmann and Wirtz [16]. In our simulation we simplified the geometry by neglecting the effects of the side walls and shortening the duct so that it included only five ribs instead of twelve. The idealized geometry of the computational model is represented by the sketch in Figure 4. A recent computational study by Agonafer and Moffatt [2] employed a similar idealization. Shortening the duct was justified because the experimental results showed that the flow was fully developed by the third rib.

We simulated the experiments of Lehmann and Wirtz using two different flow fields. In the first set of runs we computed the flow field in the x-z plane. In the second set of runs we prescribed the x-direction velocities as the parabolic profile of a fully-developed flow between parallel plates.

$$u(x,z) = 6\overline{u}(x)\left[\left(\frac{z-z_B}{z_L-z_B}\right)-\left(\frac{z-z_B}{z_L-z_B}\right)^2\right] \quad (12)$$

In both runs the energy equation was computed throughout the x-z plane. The second set of runs *simulates* the combination of a depth-averaged velocity field with a three-dimensional energy equation.

Figure 5 is a comparison of the simulation with the experimental results. Numerical values of experimental data were taken from Lehmann's Ph.D. Dissertation [14]. The data presented is the average Nusselt number across the surface of the heated rib versus the duct Reynolds number based on the gap between the top of the rib and the upper wall of the duct. The data labelled Nu_{num} are from the first set of numerical simulations for which the velocity field was calculated. Data from the simulations using a equation (12) to prescribe the velocity field are labelled Nu_{da}.

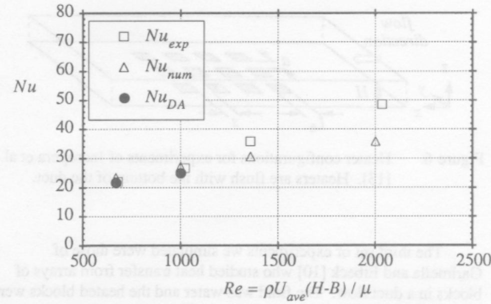

Figure 5. Comparison of simulation and experimental data for the experiments of Lehmann and Wirtz [16].

The data in Figure 5 show that both numerical simulations underpredict the Nusselt number for Reynolds numbers above 1000. This is expected because the simulations enforce steady laminar flow at all Reynolds numbers whereas the flow in the experiments undergoes transition at Reynolds numbers between 1000 and 2000. At lower Re, where the flow is indeed laminar, the depth-averaged velocity field yields good heat transfer results. For the two data points in Figure 5 the maximum discrepancies occur at $Re = 1000$ where the value of Nu_{num} is low by 11 percent compared to Nu_{exp}, and the value of Nu_{da} is low by 7 percent. These limited results do not guarantee that depth-averaging will work in all simulations of electronic cooling. Nevertheless, the data in Figure 5 are encouraging.

The second experiment we simulated involves the flow of either water or FC-77 in a large aspect ratio duct [13]. An array of twelve heaters, four rows of three heaters each, are flush with the bottom of the duct as depicted in Figure 6. Although the flow is symmetrical about the centerline we did not take advantage of this simplification in our computations. There is no need to compute the flow since it is fully-developed and the heaters are flush with the bottom of the duct. The velocity field was simply prescribed from the fully-developed profile of equation (12). The effect of the end walls on the depth-averaged velocity profile was neglected. Figure 7 is a contour plot of the depth-averaged water temperature when all twelve heaters are turned on. The inlet temperature is 25 °C, and the Reynolds number based on the hydraulic diameter is 1000. The heater input power was arbitrarily adjusted so that the temperature rise of the water is approximately 5 °C. There is very little lateral spreading of the temperature contours because the rate of conduction in the cross-stream direction is much slower than the rate of downstream convection. The data in Figure 7 cannot be quantitatively compared to the results of the experiments because Incropera et al. did not report the depth-averaged fluid temperature. Furthermore, computation of a local or average Nusselt number is meaningless because we would need to use a heat transfer correlation to compute the local wall temperature, from which we would then re-compute the heat transfer coefficient.

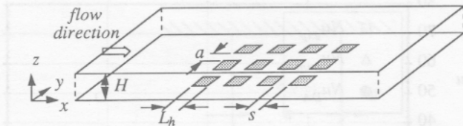

Figure 6. Heater configurations for experiments of Incropera et al. [13]. Heaters are flush with the bottom of the duct.

The third set of experiments we simulated were those of Garimella and Eibeck [10] who studied heat transfer from arrays of blocks in a duct flow. The fluid was water and the heated blocks were arranged in in-line and staggered arrays. Here we report on simulation of the in-line array depicted in Figure 8. The blocks protrude into the duct, but only the tops of the blocks were heated. This experiment is a more serious test of the depth-averaged model because we computed the flow and temperature fields. The results discussed below were obtained on a 152 by 112 grid at a Reynolds number of 500 based on the space above the blocks. The ratio of the block height to the total height of the duct (b/H) is 0.37. Although the flow at the inlet in the experiments was fully-developed, we prescribed a uniform inlet velocity, $\overline{u}(x,y) = constant$.

Figure 9 shows the depth-averaged x-direction velocity profiles normalized by the inlet velocity. The curve labelled "block" is the profile taken above the second row of heaters and the curve labelled "gap" is taken between the second and third row of heaters. For reference the y locations of the blocks are indicated by the boxes along the vertical axis. Both profiles show that the depth averaged velocity at downstream locations is higher than the depth-averaged inlet velocity. The higher velocities in the gap are due to the growth of the boundary layer on the end walls which can be seen at y/y_{max} values near 0.0 and 1.0. Still higher velocities above the blocks are due to the restriction in flow area caused by the blocks. It is somewhat surprising that the depth-averaged velocity above the blocks is higher than the velocity in the y-direction spaces between the blocks. There is some turning of the flow around the blocks, but it is slight. The data is the depth-averaged velocity, which represents the integral of the x and y-direction momentum across the space in the z-direction. In an x-y plane below the top of the blocks the flow surely turns around the blocks, but most of the x-direction momentum is contained in the fluid that passes over the top of the blocks. Hence the depth-averaged velocity vectors are essentially parallel to the x-axis, and the depth-averaged velocity is larger in the small spaces above the blocks.

Contours of the depth-averaged temperature field are shown in Figure 10. As with the data in Figure 7 there is little mixing of sensible energy in the y-direction. The rate of downstream convection is much higher than lateral conduction.

Figure 11 shows the execution time for one run of the simulation at three grid densities. The results were obtained on a Tektronix XD88/10 workstation and on a Sun SPARCstation 2. The abscissa, N, is the total number of grid points in the domain, including boundary points. The execution times should be considered representative because we have not put any concentrated effort into improving the solver or otherwise optimizing the code for speed. The data in Figure 11 show that the goal of analyzing a problem in half an hour on a workstation puts a limit on the grid density. For example, the largest grid (finest grid spacing) that can be run on the SPARCstation 2 in less than half an hour is approximately 99×99. Keep in mind that the execution time is also somewhat problem-dependent.

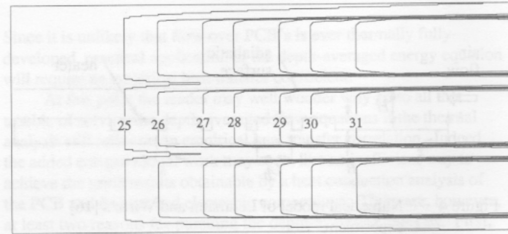

Figure 7. Contour plot of the depth-averaged water temperature for experiments of Incropera et al. [13]. $\mathrm{Re} = \rho U_{ave} D_h / \mu = 1000$.

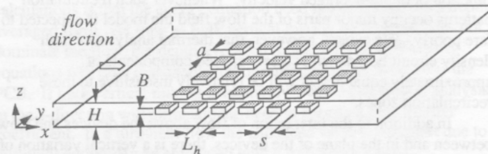

Figure 8 In-line heater layout for experiments of Garimella and Eibeck [10].

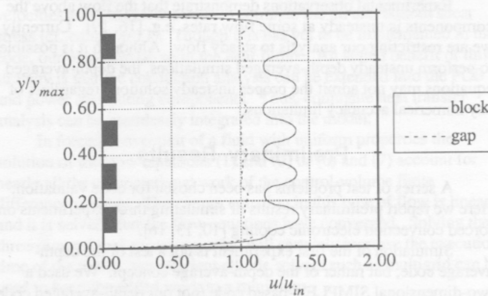

Figure 9. Normalized velocity profiles for experiments of Garimella and Eibeck [10]. Curve labelled "block" is the velocity profile above the second block. Curve labelled "gap" is the velocity profile in the gap midway between the second and third rows of blocks.

Conclusions

We have presented a new strategy for using CFD in the analysis of convectively cooled PCB's. The model is applicable to the gap between circuit boards when the bulk vertical mixing of the fluid is weak relative to the convection in the plan view of the circuit board. Preliminary results indicate that the model gives plausible predictions of the depth-averaged velocity and temperature fields. No quantitative comparisons with experimental data have been made yet.

Work is currently underway to incorporate a three-dimensional energy equation with the depth-averaged momentum and continuity equations. This will allow calculation of the heater surface temperature and heat transfer coefficient. It will also provide a basis

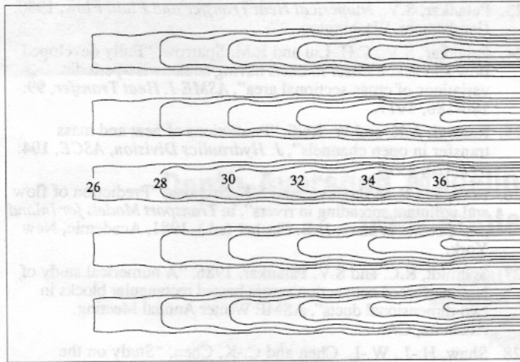

Figure 10. Contours of depth-averaged temperature for the experiments of Garimella and Eibeck [10].

for quantitative comparison with experimental results. Future plans call for inclusion of a turbulence model and improvement of the computational efficiency. Still further in the future we hope to include the conjugate heat transfer problem of conduction in the PCB and convection in the coolant.

Acknowledgement

Grants from Intel Corporation and Tektronix Incorporated in support of this research are gratefully acknowledged.

Nomenclature

overbars	denote depth-averaged quantities
a	spanwise gap between heater elements
$B(x,y)$	elevation of the bottom of the gap
b	height of the heater blocks protruding into the duct
c_p	fluid specific heat at constant pressure
$h(x,y)$	local gap height
H	total height of the duct in the simulated experiments
k	thermal conductivity
$\tilde{k}$	effective depth-averaged thermal conductivity defined in equation (10)
$L(x,y)$	elevation of the lid of the gap
L_h	streamwise length of heater elements
S	source term
s	streamwise gap between heater elements
T	temperature
t	time
u, v, w	velocity components in the x,y and z directions
u_i	velocity components in Cartesian tensor notation
x, y, z	Cartesian coordinate directions
x_i	coordinate directions in Cartesian tensor notation

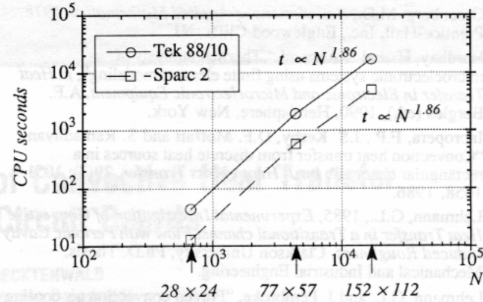

Figure 11 Execution time to simulate the experiments of Garamella and Eibeck with different grid densities

GREEK SYMBOLS

μ	fluid viscosity
ρ	fluid density
φ	generic scalar variable
τ_{ji}	shear stress components in Cartesian tensor notation

References

1. Afrid, M. and A. Zebib, "Three-dimensional laminar and turbulent natural convection cooling of heated blocks", in Heat Transfer in Turbulent Flow, HTD-Vol. 138, R.S. Amano, M.E. Crawford and N.K. Anand (ed.), 1990, ASME, New York.
2. Agonafer, D. and D.F. Moffatt, "Numerical modeling of forced convection heat transfer for modules mounted on circuit boards", *ASME J.Electronic Packaging*, **112**: 333-337, 1990.
3. Asako, Y. and M. Faghri, "Three-dimensional heat transfer analysis of arrays of heated square blocks", *Int.J.Heat Mass Transfer*, **32**: 395-405, 1989.
4. Braaten, M.E. and S.V. Patankar, "Analysis of laminar mixed convection in shrouded arrays of heated rectangular blocks", *Int.J.Heat Mass Transfer*, **28**: 1699-1709, 1985.
5. Chen, K.S. and Y.R. Chang, "Thermal-network simulation of printed circuit boards within a naturally ventilated cabinet", in *Temperature/Fluid Measurements in Electronic Equipment*, HTD-Vol. 89, J. Bartoszek, et al. (ed.), 1987, ASME, New York.
6. Choudhury, D., 1986, *A Quasi-three-dimensional calculation procedure for the prediction of flow and heat transfer in heat exchanger passages*, University of Minnesota, Ph.D. Thesis, Mechanical Engineering Department.
7. Currie, I.G., *Fundamental Mechanics of Fluids*, 1974, McGraw-Hill, New York.
8. Davalath, J. and Y. Bayazitoglu, "Forced convection cooling across rectangular blocks", *ASME J.Heat Transfer*, **109**: 321-328, 1987.
9. De Mey, G. and L. Van Schoor, "Thermal analysis of hybrid circuits with mounted components", *Hybrid Circuits*, **15**: 28-32, 1988.
10. Garimella, S.V. and P.A. Eibeck, "Heat transfer characteristics of an array of protruding elements in single phase forced convection", *Int. J. Heat Mass Transfer*, **33**: 2659-2669, 1990.

11. Greenberg, M.D., *Foundations of Applied Mathematics*, 1978, Prentice-Hall, Inc., Englewood Cliffs, NJ.
12. Hardisty, H. and J. Abboud, "Thermal analysis of microelectronic systems using finite element modeling", in *Heat Transfer in Electronic and Microelectronic Equipment*, A.E. Bergles (ed.), 1990, Hemisphere, New York.
13. Incropera, F.P., J.S. Kerby, D.F. Moffatt and S. Ramadhyani, "Convection heat transfer from discrete heat sources in a rectangular channel", *Int.J. Heat \ Mass Transfer*, **29**(7): 1051-1058, 1986.
14. Lehmann, G.L., 1985, *Experimental Investigation of Convective Heat Transfer in a Transitional channel Flow with Periodic Cavity Induced Roughness*, Clarkson University, Ph.D. Thesis, Mechanical and Industrial Engineering.
15. Lehmann, G.L. and J. Pembroke, "Forced convection air cooling of simulated low profile electronic components: Part 1–Base Case", *ASME J. Electronic Packaging*, **113**: 21-26, 1991.
16. Lehmann, G.L. and R.A. Wirtz, "The effect of variations in stream-wise spacing and length on convection from surface mounted rectangular components", *ASMEJ. Electronic Packaging*, **111**: 26-32, 1989.
17. Linton, R.L. and D. Agonafer, "Thermal model of a PC", in *Numerical Simulation of Convection in Electronic Equipment Cooling*, HTD-Vol. 121, A. Ortega and D. Agonafer (ed.), 1989, ASME, New York.
18. Mahaney, H.V., F.P. Incropera and S. Ramadhyani, "Comparison of predicted and measured mixed convection heat transfer from an array of discrete sorces in a horizontal rectangular channel", *Int. J.Heat Mass Transfer*, **33**: 1233-1245, 1990.
19. McEntire, A.B. and B.W. Webb, "Local forced convective heat transfer from protruding and flush-mounted two-dimensional discrete heat sources", *Int. J.Heat Mass Transfer*, **33**: 1521-1533, 1990.
20. McGuirk, J.J. and W. Rodi, "A depth-averaged mathematical model for the near field of side discharges into open-channel flow", *Journal of Fluid Mechanics*, **86**(4): 761-781, 1978.
21. Moffat, R.J. and A. Ortega, "Direct air-cooling of electronic components", in *Advances in Thermal Modeling of Electronic Components and Systems, Vol. 1*, 1, A. Bar-Cohen and A.D. Kraus (ed.), 1988, Hemisphere, New York.
22. Olsen, J., B.W. Webb and M. Queiroz, 1989, "Local three-dimensional convective heat transfer from a heated cube", ASME Winter Annual Meeting, HTD-Vol-123, San Francisco, CA.
23. Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, 1980, Hemisphere, Washington.
24. Patankar, S.V., C.H. Liu and E.M. Sparrow, "Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area", *ASME J. Heat Transfer*, **99**: 180-186, 1977.
25. Rastogi, A.K. and W. Rodi, "Predictions of heat and mass transfer in open channels", *J. Hydraulics Division, ASCE*, **104**: 397-420, 1978.
26. Rodi, W., R.N. Pavlovic and S.K. Srivatsa, "Prediction of flow and pollutant spreading in rivers", in *Transport Models for Inland and Coastal Waters*, H.B. Fischer (ed.), 1981, Academic, New York.
27. Schmidt, R.C. and S.V. Patankar, 1986, "A numerical study of laminar forced convection across heated rectangular blocks in two-dimensional ducts", ASME Winter Annual Meeting, Anaheim, CA.
28. Shaw, H.-J., W.-L. Chen and C.-K. Chen, "Study on the laminar mixed convective heat transfer in three-dimensional channel with a thermal source", *ASME J. Electronic Packaging*, **113**: 40-49, 1991.
29. Sparrow, E.M., J.E. Niethammer and A. Chaboki, "Heat transfer and pressure drop characteristics of arrays of rectangular modules encountered in electronic equipment", *Int. J. Heat Mass Transfer*, **25**: 961-973, 1982.
30. Sparrow, E.M., S.B. Vermuri and D.S. Kadle, "Enhanced and local heat transfer, pressure drop, and flow visualization for arrays of block-like electronic components", *IInt. J. Heat Mass Transfer*, **26**: 689-699, 1983.
31. Sparrow, E.M., A.A. Yanexmoreno and D.R. Otis Jr., "Convective heat transfer response to height differences in an array of block-like electronic components", *Int. J. Heat Mass Transfer*, **27**: 469-473, 1984.
32. Weiss, J., P. Fortner, B. Pearson, K. Watson and T. Monroe, "Modeling air flow in electronic packages", *Mechanical Engineering*, **111**: 56-58, 1989.
33. White, F.M., *Fluid Mechanics*, 1986, McGraw-Hill, New York.
34. Zebib, A. and Y.K. Wo, "A two-dimensional conjugate heat transfer model for forced air cooling of an electronic device", *ASME J. Electronic Packaging*, **111**: 41-45, 1989.