Simplified Simulation of Convective Heat Transfer from an Array of Heated Blocks in a Rectangular Channel

Yidong Ma
Gerald W. Recktenwald
Mechanical Engineering Department
Portland State University
Portland, Oregon

ABSTRACT

The convective heat transfer from an array of heated blocks that simulate electronic devices is predicted with a simplified numerical model. A control-volume finite difference technique is used to solve the three-dimensional energy equation in the domain occupied by the heated blocks and the fluid flowing around the blocks. Rather than also solve the three-dimensional momentum and continuity equations for the fluid, simple assumptions for the fluid velocity profile are used to construct the convection-diffusion coefficients that appear in the discrete form of the energy equation. These assumptions are made to speed the numerical solution so that the model will be feasible for routine design use. The accuracy of the procedure is tested by comparison to experimental results obtained for arrays of flush and protruding heaters for both laminar and turbulent flow. The results show that these simplifications yield reasonable predictions that do not rely on specification of a heat transfer coefficient. Unfortunately, since the quantitative accuracy of the model depends on the assumed shape of the velocity profile, the model is not yet capable of being used as a universal design tool.

NOMENCLATURE

A	surface area of the heated blocks
B	height of the protruding, heated blocks
D	pipe diameter in experiments on fully-developed flow.
H	spacing between the plates
L_x, L_y	length of the heated blocks or heated patches in the x and y directions, cf. Figure 1.
Nu	Nusselt number, defined separately for each experimental data set
Pr	Prandtl number
Q	rate of heat generation (W) in a heated block
Re	Reynolds number, defined separately for each experimental data set
S_x, S_y	spacing between the heated blocks or heated patches in the x and y directions cf. Figure 1.
T	temperature
U	average velocity in the duct
W	length of the plates in the spanwise (y) direction
c_p	specific heat of the material
f	Darcy friction factor
h	heat transfer coefficient
k	thermal conductivity
s	$\min[(z-B), (H-z)]$, the shortest vertical distance to the edge of the gap shown in Figure 2.
u_i	velocity component in direction x_i
x_i	Cartesian coordinate direction
ε_H	effective diffusivity for thermal energy transport
κ	von Kármán's constant
ν	kinematic viscosity of the fluid
ν_t	eddy viscosity
ρ	density of the material
σ_H	turbulent Prandtl number

INTRODUCTION

The goal of this research is to devise a simplified tool for thermal analysis of electronic components mounted on printed circuit boards. The criterion of simplicity requires that a user can quickly define the model and obtain the results. In addition the

model should be relatively free of problem-dependent parameters. Considerable progress has been made in developing heat transfer correlations for regular arrays of heated blocks(Anderson and Moffat, 1990; Anderson and Moffat, 1992b; Wirtz and Chen, 1992). It seems unlikely, however, that a general heat transfer correlation exists for the irregular geometries found in many printed circuit boards. Therefore it is desirable to create a model that does not require specification of a heat transfer coefficient. Our research is aimed at determining whether such a model can be developed.

It is possible to analyze electronic cooling problems with a three-dimensional, computational fluid dynamics (CFD) model of the flow and heat transfer. It is not clear that such a model is likely to be quick enough, even with the increasing computational power modern computers, for routine design use. On the other end of the spectrum, resistive network and finite element models of heat conduction in the solid components on printed circuit boards are computationally efficient, but these techniques require specification of heat transfer coefficients between the component surfaces and the coolant. We are attempting to take the middle ground between these strategies.

An earlier effort involved development of a depth-averaged model of the flow field in the gap between two printed circuit boards (Recktenwald and Butler, 1991). In this paper we report on an extension of that model to thermal analysis without using heat transfer coefficient data. The thermal energy equation is solved in its three-dimensional form (no depth-averaging) by assuming that the flow field is known. The results of these computations are compared to results of several experimental studies for idealized physical models of electronic cooling situations.

The results reported here do not involve direct coupling of the depth-averaged model with the three-dimensional energy equation. This paper documents the capabilities and limitations of a subset of our overall approach, viz., given a depth-averaged velocity distribution, expand the depth-averaged velocity across the narrow gap so that a solution to the three-dimensional energy equation can be obtained. The three-dimensional energy equation must be solved in order to avoid specification of a heat transfer coefficient. Since the available experimental data involve predominantly uni-directional flow, solution of the depth-averaged equations does not add information to the analysis. The approach used here is equivalent to assuming that the depth-averaged velocity is uniform over the heater array.

The procedure for obtaining the results presented in this paper is as follows. We choose an experimental data set involving flow past heater arrays in ducts. We build a computational model with the same geometry and thermal boundary conditions as the experiment under consideration. For different assumptions about the velocity profile, the model is used to solve the energy equation in the fluid and in the solid heaters. From the temperature field the heat transfer coefficients and Nusselt numbers are computed and these results are compared with the experimentally determined quantities.

Several researchers have studied heat transfer from arrays of heaters that simulate electronic devices on printed circuit boards. Rather than provide a comprehensive review of the literature on these heat transfer experiments we briefly summarize the experiments used to test our computational model.

Incropera et al. (Incropera et al., 1986) performed experiments and computations for turbulent flow past a 4×3 array of heaters mounted flush on the bottom of a duct. They also developed a two-dimensional numerical model of their experiment that is very similar to ours in that it uses an assumed velocity profile and an eddy-viscosity turbulence model. By neglecting variations in the spanwise direction they solved the two-dimensional form of the energy equation.

Mahaney et al. (Mahaney et al., 1990) extended the work of Incropera et al. to the low Reynolds number regime where mixed convection becomes important. Their numerical model used a partially parabolic formulation of the Navier-Stokes and energy equations. By marching in the stream-wise direction solutions were obtained to the two-dimensional elliptic equations in the duct cross-section. After finding an appropriate scaling for the effect of variable properties, the computational and experimental results were in excellent agreement.

Garimella and Eibeck (Garimella and Eibeck, 1990) studied forced convection heat transfer from 6×5 arrays of block heaters in laminar and turbulent flow. The effect of channel height, heater spacing and heater arrangement (in line versus staggered) were studied. In a companion paper (Garimella and Eibeck, 1991) they reported extensive fluid dynamic measurements for their heater array.

Heindel et al. (Heindel et al., 1992) measured the heat transfer coefficients for a single column of ten heated blocks in a duct. The blocks were made of Lexan and in the top of the block a square heater with dimensions smaller than the Lexan block was exposed to the flow. Measurements were conducted for different channel heights and a wide range of Reynolds numbers. Clear demarcation between natural convection, mixed convection, forced laminar convection and forced turbulent convection was observed.

Anderson and Moffat (Anderson and Moffat, 1992a) investigated the forced convection of air past an 8×6 array of rectangular heater blocks. They reported the adiabatic heat transfer coefficient and the superposition kernel functions.

Physical Model of the Flow

As depicted in Figure 1 the physical situation involves flow in a duct formed by parallel plates with sealed edges. The z-direction distance between the plates, H, is small compared to the length of the plates in the x and y directions. Attached to the bottom plate is an array of heating elements, which are either flush with the plate or are blocks protruding into the duct. The hydrodynamic effect of the duct edges at $y = 0$ and $y = W$ are relatively unimportant compared to the effects of the protruding blocks. Table 1 summarizes the geometrical parameters for the experimental data sets we used to test our model.

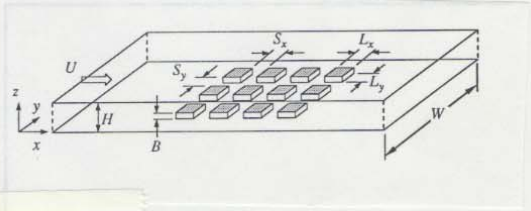

Figure 1 Geometry and nomenclature for an array of heated blocks

Data Set	S_x/L_x	S_y/L_x	L_y/L_x	L_h/L_x	H/B
Mahaney et al.	0.25	0.25	1.0	1.0	∞
Incropera et al.	0.25	0.25	1.0	1.0	∞
Garimella & Eibeck	0.87	0.87	1.0	1.0	1.2, 1.9, 2.7, 3.6
Heindel et al.	0.25	∞	1.0	0.25	3.48
Anderson & Moffat	0.34	0.34	0.81	1.0	1.5, 2.25, 3.0, 4.6

Table 1 Geometrical parameters for data sets used to test the computaional model.

We use a very simple model of the flow field. We assume that the flow is confined to the gap between the top surface of the heaters and the upper duct wall. This situation is depicted schematically in Figure 2, which is an elevation view of the flow past two block heaters in an array. We have performed computations with three velocity profile assumptions: uniform flow, fully-developed laminar flow, and fully-developed turbulent flow. Since edge effects are assumed to be negligible we use the fully-developed velocity profiles for flow between parallel plates. In this case the velocity field is a function of z alone. Though this model of the flow field is rather simplistic, our results show that in most cases the heat transfer predictions are in reasonable agreement with the experiments.

THEORY

The computational model is based on solution of the three-dimensional energy equation with a prescribed velocity field. The flow is steady, the fluid properties are uniform, and a turbulence model is used. The computer code solves

$$\frac{\partial}{\partial x_j}(\rho u_j T) = \frac{\partial}{\partial x_j}\left[\left(\frac{k}{c_p} + \rho\varepsilon_H\right)\frac{\partial T}{\partial x_j}\right] \tag{1}$$

in the three-dimensional space occupied by the fluid and the solid blocks. Our prescription for the velocity field and eddy viscosity distribution are described in the next sections. The boundary conditions for equation (1) are as follows. The temperature at the inlet is given. The upper surface of the channel is adiabatic. At the outflow boundary the conventional specification of zero heat transport by conduction (Neumann boundary condition) is used.

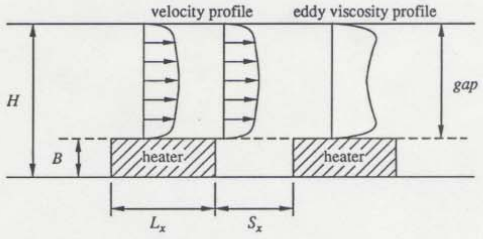

Figure 2 Sketch of the velocity and eddy viscosity profiles used in the model. The dashed line delineates the extent of the gap region where the flow is assumed to occur.

The boundary condition on the bottom surface of the domain depends on the experimental configurations, which are described in the results section below. For the case of flush heaters the bottom surface is adiabatic except for the heated patches. For the case of protruding heated blocks the bottom surface of the domain is adiabatic. The energy input in the latter case occurs as a distributed heat source in the blocks.

Velocity Profiles

The velocity profile is taken to be a known function of z in the gap between $z = B$ and $z = H$. The fluid velocity is assumed to be zero for $z < B$. In other words the normalized velocity profile for protruding block heaters with gap distance $H - B$ is assumed to be identical to the normalized velocity profile for flush heaters in a duct with plate spacing H.

We compare heat transfer coefficient predictions for three velocity profile functions: uniform flow, fully-developed laminar flow, and fully-developed turbulent flow. For fully-developed laminar flow the velocity profile is

$$u = 6U\left[\frac{z-B}{H-B} - \left(\frac{z-B}{H-B}\right)^2\right] \tag{2}$$

where U is the average velocity for the duct cross-section.

The velocity profile for fully-developed turbulent flow is constructed from the law of the wall. If we define the vertical coordinate s as the shortest vertical distance to the edge of the gap shown in Figure 2, i.e.,

$$s \equiv \min[(z-B), (H-z)] \tag{3}$$

then the turbulent velocity profile is

$$u^+ = \begin{cases} s^+ & s^+ \le 5 \\ 5\ln\frac{s^+}{5} + 5 & 5 < s^+ \le 30 \\ 2.5\ln s^+ + 5.5 & s^+ \ge 30 \end{cases} \tag{4}$$

where u^+ and s^+ are the dimensionless velocity and distance in wall coordinates

$$u^+ \equiv \frac{u}{u^*} \qquad s^+ \equiv \frac{u^* s}{V} \tag{5}$$

u^* is the friction velocity

$$u^* \equiv \sqrt{\tau_w/\rho} \tag{6}$$

and τ_w is the turbulent wall shear stress.

To specify the turbulent velocity profile we calculate the turbulent wall shear stress from correlations for fully-developed pipe flow. The wall shear stress is related to the Darcy friction factor

$$\tau_w = f\frac{\rho U^2}{8} \tag{7}$$

The friction factor for fully developed pipe flow is given by the well known Colebrook equation (White, 1986). Rather than use this equation, which is implicit in f, we use the explicit formula suggested by Techo et al. (Techo et al., 1965)

$$f = \left[0.8686\ln\left(\frac{Re_D}{1.964\ln(Re_D)-3.8215}\right)\right]^{-2} \tag{8}$$

where Re_D is the Reynolds number based on pipe diameter. For friction calculations White (White, 1986) recommends using an effective diameter, D_{eff}, instead of either D or the hydraulic diameter. The effective diameter is computed by requiring that the correct value of the laminar friction factor for the duct in question be obtained when the friction factor is calculated from the formula

$$f_{lam} = \frac{64}{Re_{Dh}} \tag{9}$$

The exact solution for fully developed laminar flow between parallel plates yields

$$f_{lam,PP} = \frac{96}{Re_{Dh}} \tag{10}$$

Thus, for flow between parallel plates

$$D_{eff} = \frac{64}{96}D_h = \frac{2}{3}D_h \tag{11}$$

In summary, the procedure for prescribing the turbulent velocity profile is as follows. An experimental condition is specified by the geometry of the duct and the flow rate. The effective diameter of the *gap* is computed from

$$D_{eff} = \frac{2}{3}D_h = \frac{4}{3}(H-B) \tag{12}$$

This effective diameter is used in equation (8) to compute the friction factor. The velocity profile is then computed using equations (3) through (7).

Turbulence Model

For the turbulent flow regime we use an algebraic eddy viscosity model to compute the effective diffusivity of the moving fluid in the gap. Like the fluid velocity the eddy viscosity is assumed to vary only in the z direction.

In the case of protruding heater blocks the fluid below the top surface of the block ($z < B$) is assumed to be stationary and have no enhanced diffusivity due to the turbulence. To be precise the eddy diffusivity is

$$\varepsilon_H = \begin{cases} \dfrac{\nu_t}{\sigma_H} & \text{if } B \le z \le H \\ 0 & \text{if } z < B \end{cases} \tag{13}$$

where ν_t is the eddy viscosity, and σ_H is the turbulent Prandtl number. We use $\sigma_H = 0.9$ for all data sets except that of Incropera et al., for which $\sigma_H = 1$. The exception is to be consistent with the numerical predictions of Incropera et al, who used $\sigma_H = 1$. In either case changing the value of σ_H from 0.9 to 1.0 results in changes in Nu of 5 percent or less.

The functional dependence of ν_t on z is based on models derived from experiments by van Driest (van Driest, 1956), Reichardt (Burmeister, 1983, p. 415) and Laufer (Laufer, 1954) for fully-developed pipe flow. We found it necessary to adapt these models for the low Reynolds number range of the turbulent flow encountered in the experimental data sets we simulated. The eddy viscosity concept is useful when the turbulence can be characterized by single length and velocity scales (Tennekes and Lumley, 1972), as is the case for fully-developed flow in ducts. One can certainly argue that we are pushing the limits of applicability for such a simple model of turbulence. For dense arrays of components, however, it seems reasonable to assume that the flow in the gap above the components is similar to flow in a very rough duct.

In the region near the wall the van Driest eddy viscosity model is (van Driest, 1956)

$$\nu_t = \kappa^2 s^2\left[1-\exp\left(-\frac{s^+}{26}\right)\right]^2\left|\frac{\partial u}{\partial s}\right| \tag{14}$$

From measurements in fully developed pipe flow Reichardt (Burmeister, 1983, p. 415) derived a simple formula for the eddy viscosity that is not dependent on the velocity gradient.

$$\frac{\nu_t}{\nu} = \frac{Re^*}{15}\eta(2-\eta)\left[1+2(1-\eta)^2\right] \tag{15}$$

where

$$Re^* = \frac{u^* D/2}{\nu} \tag{16}$$

$$\eta = \frac{y}{D/2} \tag{17}$$

y is interpretted as the distance from the wall and $D/2$ is the pipe radius. Laufer's data (Laufer, 1954) are more widely cited than Reichardt's, but Laufer did not provide a convenient functional relationship for ν_t. By curve fitting Laufer's data we obtain a formula similar to Reichardt's

$$\frac{\nu_t}{\nu} = \frac{Re^*}{14.3}\eta^{1.55}(2-\eta)^{1.79}\left[1+6(1-\eta)^{3.62}\right] \tag{18}$$

Including an ad hoc multiplying factor of van Driest flavor we get

$$\frac{\nu_t}{\nu} = \frac{Re^*}{14.3}\eta^{1.55}(2-\eta)^{1.79}\left[1+6(1-\eta)^{3.62}\right]\left[1-\exp\left(-\frac{\eta Re^*}{26}\right)\right] \quad (19)$$

Addition of the wall damping term is necessary to obtain good results for low Reynolds number turbulent flow. To use equation (19) for flow in parallel plate channels use

$$Re^* = \frac{u^*(H-B)/2}{\nu} \quad (20)$$

$$\eta = \frac{s}{(H-B)/2} \quad (21)$$

NUMERICAL METHOD

The model is implemented in a computer code based on the control-volume finite-difference formulation. An implicit system of linear equations are obtained by integrating over the control volume following standard practices. Since the flow field is prescribed, only the equation (1) needs to be solved, which results in significant savings of computer time. The coefficients of the discrete energy equation are obtained with the Power-Law upwinding scheme. The details of the method may be found in Chapter 5 of Patankar's book (Patankar, 1980). The well-known false-diffusion problems associated with this first order scheme are not present in our calculations because the fluid velocity vectors are exactly aligned with the x-direction grid lines everywhere in the flow field. After solving for the temperature field in the fluid, the model computes the heat transfer coefficient h and the Nusselt number Nu.

In general the numerical solution depends on the resolution of the finite-difference grid. As the number of control volumes increases the variables of interest (h and Nu) approach asymptotes. Since the temperature field varies rapidly near the heat transfer surface, it is more effective to use finer grids in the region near the heat transfer surface and coarser grids elsewhere. All results presented below are obtained by refining the grid until the variables of interest do not change significantly.

In addition to grid independence, the computed solutions were required to satisfy an overall energy balance, and the linearized coefficient equations were solved to a tight tolerance. In all cases the energy balances close to within 5×10^{-5} W. Since the total heater power ranged from 12 W to about 200 W an energy balance can said to be exactly met. The discrete energy equation was solved until its normalized residual was less than 1×10^{-6} for all results presented below. Such tight tolerances would be unnecessary for use of the program as a design tool.

HEAT TRANSFER PREDICTIONS

The model was tested by simulating experiments performed by other researchers on arrays of heaters in large aspect ratio ducts. In the experiments the heaters were either flush with the bottom wall of the duct or protruding into the duct. Individual heaters were powered, and the power input, the fluid inlet temperature, and the temperatures of the block in the array were measured. From this data the heat transfer coefficient and Nusselt number were calculated. The definitions of heat transfer coefficient, Nusselt number and Reynolds number were not the same in all the experiments considered. The appropriate definitions are provided with each comparison between simulation and experiment.

For each experiment we use the model to solve the energy equation subject to different assumptions about the velocity profile. In the cases where the heater blocks protrude into the duct the blocks were also part of the computational domain. The heaters were fabricated from aluminum or copper and the fluid was either air, water or FC-77. The large differences in thermal conductivity between the solid and fluid guaranteed that the heaters were virtually isothermal.

The experimentalists reduced their data by making corrections for variable thermophysical properties and conduction through the base of the duct. In our calculations the thermophysical properties were defined to be constant. Corrections for conduction losses in our model were unnecessary because it is possible to compute the heat transfer rate through any surface within the domain.

Flush Heaters

Mahaney et al. (Mahaney et al., 1990) studied low Reynolds number convection from a 4×3 array of flush mounted heaters. In the model the velocity field was prescribed with equation (2) with $B = 0$. No turbulence model was used since the flow was assumed to be laminar for all Reynolds numbers tested. Figure 3 is a comparison of the simulation with the experimental results of Mahaney et al. The Reynolds number and Nusselt number for a heater were defined as follows

$$Re = \frac{UL_x}{\nu} \qquad Nu = \frac{QL_x}{A(T_h - T_b)k} \quad (22)$$

where L_x is the length of the heater ($L_x = L_y$ for this experiment), T_b is the bulk mixed temperature of the fluid immediately upstream of the heater, and T_h is the temperature of the heater. The ordinate of the graph is the row averaged Nusselt number divided by the row-averaged $Pr^{1/3}$ where Pr is evaluated at T_h.

The data in Figure 3 show that the numerical simulation (solid curves) agree quite well with the experimental data (dashed curves). The largest deviation from the experiment data is less than 14 percent for the last row of heaters at $Re = 600$. At this low Re mixed convection is important and this effect obviously cannot be accounted for with the velocity profile of equation (2). At $Re = 2000$ the largest deviation was 8 percent for the last row of heaters.

Incropera et al. (Incropera et al., 1986) studied turbulent flow using the same configuration as Mahaney et al. Incropera et al. also used a numerical model to predict the heat transfer. The definition of Reynolds number and Nusselt number for this experiment was

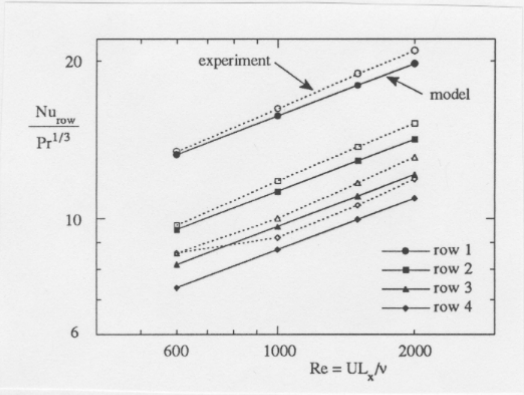

Figure 3 Comparison of simulation results with experimental data of Mahaney et al. for $B = 0$, $S_X/L_X = 0.25$

$$Re = \frac{UD_h}{\nu} \qquad Nu = \frac{QL_x}{A(T_h - T_{in})k} \tag{23}$$

where D_h is the duct hydraulic diameter and T_{in} is temperature of the fluid at the inlet. Our computational model of this experiment used the same grid as for the experiment of Mahaney et al.

Figure 4 shows the comparison of the simulation results with the experimental data of Incropera et al. for turbulent flow. The numerical simulation (solid symbols) agree quite well with the experimental data (dashed curves). Using the curve fit formula given by Incropera et al. the largest deviation from the experiment data is about 14 percent and occurs at $Re = 14000$.

Protruding Heaters

Garimella and Eibeck (Garimella and Eibeck, 1990) studied forced convection heat transfer from a 6×5 array of block heaters in laminar and turbulent flow. We simulated their base-case experiments, which involved the measurement of Nusselt number as a function of Reynolds number and channel height for a fixed heater size and spacing. At each data point we made two runs, one using a fully-developed velocity profile and another using a uniform velocity profile ($u(z)$ = constant). In either case the fluid was assumed to have non-zero velocity only in the gap between the top of the heater blocks and the upper wall of the channel (cf. Figure 2).

Garimella and Eibeck used different definitions of Re and Nu to achieve a correlation of their results for all channel spacings. Here we present the comparison of their measurements and our predictions with only one definition of Re and Nu. These are

$$Re_H = \frac{UH}{\nu} \qquad Nu = \frac{QB}{A(T_h - T_{ad})k} \tag{24}$$

where T_{ad} is the adiabatic temperature of a heater. Garimella and Eibeck observe that for their experiments the difference between the values of Nu based on T_{ad} and Nu based on T_m were within

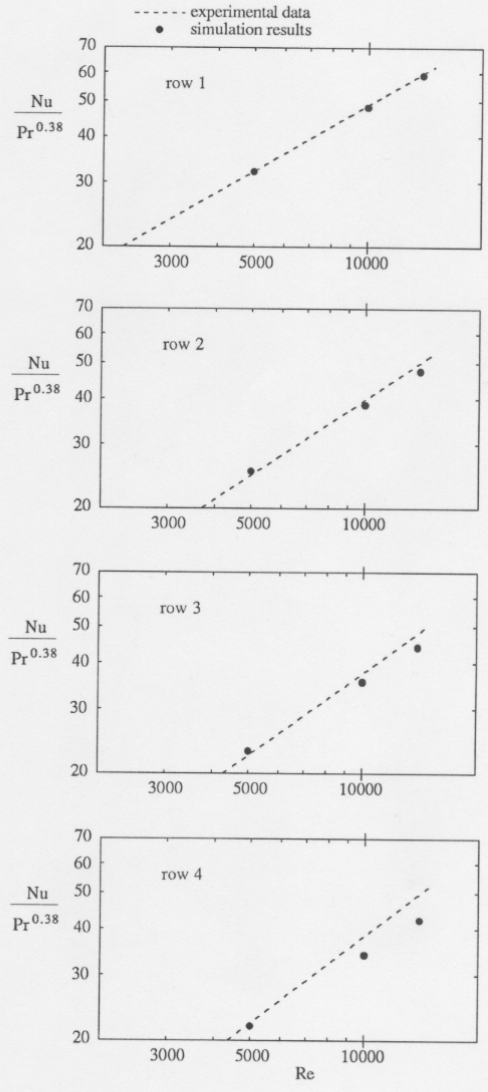

Figure 4 Comparison of simulation results with experimental data of Incropera et al. for $B = 0$, $S_x/L_x = 0.25$

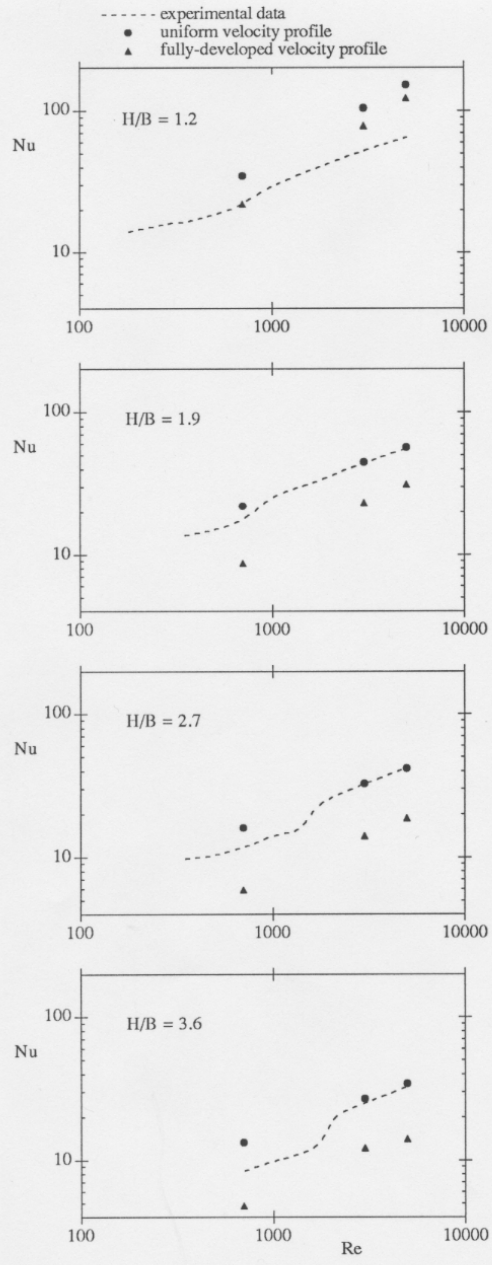

Figure 5 Comparison of simulation results with experimental data of Garimella and Eibeck for $S_x/L_x = 0.87$

the experimental uncertainty. Garimella and Eibeck also observe that the transition from laminar to turbulent flow occurs at different values of Re_H depending on the relative channel height H/B. We used their reported transition Re_H to switch between laminar and turbulent velocity profiles when the fully-developed profiles were used.

In Figure 5 the experimental data are shown as dashed lines and the simulation results for the two velocity profile assumptions are shown with solid symbols. For $H/B = 1.2$ the agreement between the simulation and experiments is poor. Garimella and Eibeck deduce that at this spacing most of the heat transfer is from the sides of the blocks, not the top, because much of the flow goes around the sides of the blocks. Our model assumes just the opposite, i.e., that all flow occurs above the blocks. The disagreement between the simulated Nu and the measured Nu confirms that the assumed velocity profile is inappropriate at this channel spacing.

At higher channel spacings ($H/B = 1.9$, 2.7, 3.6) the simulation results agree with the experiments at higher Re_H when the uniform velocity profile assumption is used. Two issues need to be considered here: the transition from laminar to turbulent flow and the influence on the simulation of the assumed velocity profile. First we consider the influence of the velocity profile assumption at higher Reynolds number. In a companion paper (Garimella and Eibeck, 1991) Garimella and Eibeck presented detailed flow measurements for the same apparatus used for their heat transfer experiments. They observed that beyond the third row an "array shear layer" structure had developed in which the flow above the blocks was nearly uniform and the flow below the top of the blocks was very small. The heat transfer measurements used to obtain the data presented in Figure 5 were averaged over the last three rows of the array, i.e., in the region where the array shear layer had developed. Thus the agreement between the experiments and the simulation is not coincidental. Our assumed velocity profile is a good representation of the flow field measured by Garimella and Eibeck.

Garimella and Eibeck report a significant difference between flow within a column (nominally same value of y) and the flow in the spanwise spaces between columns. Since our velocity profile assumption is uniform across the y-direction it is not a true representation of the velocity profile in the duct. However, since most of the heat transfer for large H/B occurs through the top of the block, the error in the assumed velocity profile in the spanwise spaces between columns of blocks does not affect the computed Nu.

The large difference in computed Nu between uniform and fully-developed flow profiles is not encouraging. In all cases the fully-developed velocity profile under predicts the Nusselt number. To a first order approximation the key difference between the fully-developed and uniform velocity profiles is the z-location of the transition between high and low velocities. For the uniform velocity profile this transition occurs at the top of the block. For the fully-developed velocity profile the transition occurs above the top of the block. Even though the profile is assumed to be turbulent, meaning that the fully-developed

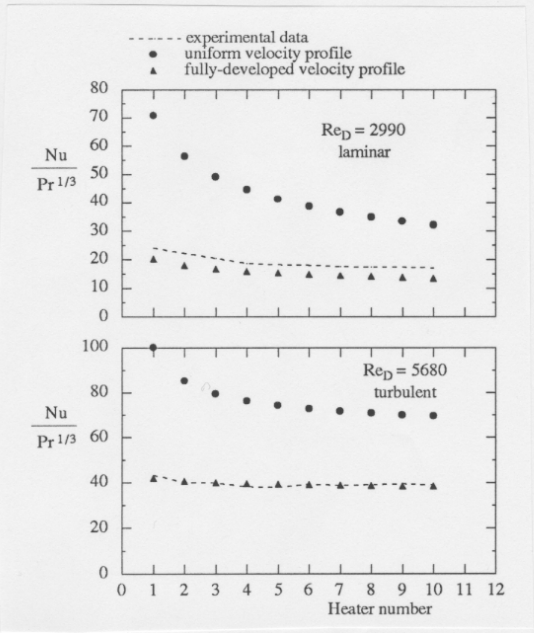

Figure 6 Comparison of simulation results with experimental data of Heindel et al. for $H/B = 3.48$, $S_x/L_x = 0.25$

velocity profile is relatively flat, the data in Figure 5 indicates that this difference is significant.

The discrepancy between the predictions and measurements at lower Reynolds numbers is also due to the velocity profile assumption. At the low Re data points the eddy viscosity is zero and the energy flow is only due to molecular transport down the temperature gradient. In this case the shape of the velocity profile has a more significant affect on heat transport. The fluid in these experiments is water ($Pr \approx 7$) for which the thermal boundary layer thickness on the top of any heater will be thinner than the hydrodynamic boundary layer. The assumption of uniform velocity profile results in an unrealistic enhancement of heat transfer because the fluid velocity adjacent to the heater surface is too high.

We next simulated the experiments of Heindel et al.(Heindel et al., 1992) who studied mixed and forced convection heat transfer from a single column of ten heaters protruding into a duct. The heater block used in these experiments consisted of a square heating element mounted flush with the top of a larger Lexan block, the top of which was also square. The ratio of heater length to block length is $L_h/L_x = 0.25$. These geometrical details along with appropriate thermophysical properties of the heater materials were included in our computational model. Heindel et al. present a large amount of data for two channel heights and different heater powers over a range of Reynolds numbers. We have selected part of their results that show the heater to heater variation of Nusselt number for two Reynolds numbers and a relative channel height $H/B = 3.48$. The Reynolds number and Nusselt number definitions for this data are

$$Re = \frac{UD_H}{V} \qquad Nu = \frac{QL_h}{A(T_s - T_b)k} \tag{25}$$

where D_H is the duct hydraulic diameter, T_s is the temperature of the exposed heater surface, and T_b is the bulk fluid temperature immediately above the center of the heat source. Since Nu is defined in terms of $T_s - T_b$, Nu will decrease in the flow direction because of the wake of upstream heaters.

Figure 6 presents a comparison of the simulation results and the experimental data for forced convection laminar flow ($Re_D = 2990$) and forced convection turbulent flow ($Re_D = 5680$). The distinction between laminar and turbulent flow is made by an experimentally observed transition in overall Nusselt number. The local Nusselt number is scaled with $Pr^{1/3}$ to account for the effect of variable fluid properties in the experiment. The experimental data for both Re_D values show little variation in local Nu. The simulation results obtained by assuming a uniform velocity profile in the gap drastically over predict the local Nu number, whereas simulation results obtained by assuming that the velocity profile is fully-developed results in close agreement with the experimental data. This is the reverse of the situation obtained when we simulated the experiments of Garimella and Eibeck. A possible explanation for this can be found by considering the relative spacing S_x/L_x between the heater blocks.

For the experiments of Garimella and Eibeck considered here $S_x/L_x = 0.87$ and for the experiments of Heindel et al. $S_x/L_x = 0.25$. In addition the heated surface area of the blocks in Heindel's experiment was only one quarter of the length of the block. Since the distance between blocks in Heindel's experiment was relatively short, it is reasonable to think that the flow in the gap above the blocks would behave more like fully-developed flow. In other words one would expect the velocity near the surface of the block to be low. In contrast, in the experiments of Garimella and Eibeck the streamwise spacing of the blocks relative to the heater length was greater. Thus, one would expect that the top surface of the heaters would tend to retard the fluid less than in the case of more closely spaced heaters. This reasoning, though it has the right qualitative sense, is approximate because we have made such a simple prescription for the velocity profile. The actual flow is three-dimensional, as demonstrated by the flow visualization of Garimella and Eibeck (Garimella and Eibeck, 1991).

The last set of experiments we simulated were those of Anderson and Moffat (Anderson and Moffat, 1992a), which involves flow of air past an 8×6 array of rectangular flatpack heaters. The heat transfer results are presented in dimensional form as the adiabatic heat transfer coefficient as a function of

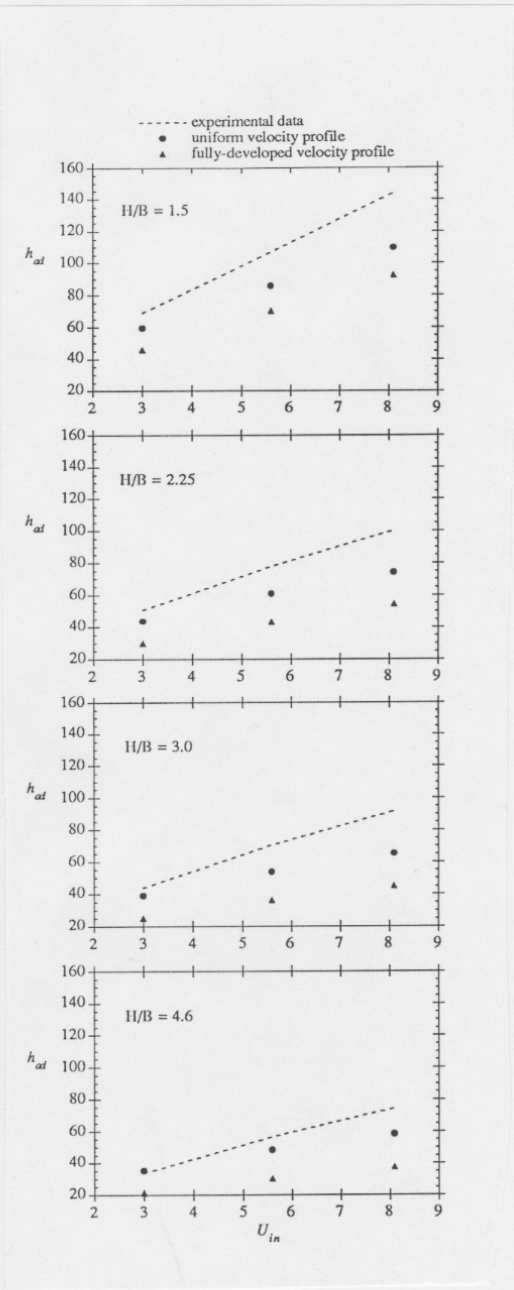

Figure 7 Comparison of simulation results with experimental data of Anderson and Moffat for $S_x/L_x = 0.34$ and $S_y/L_y = 0.27$

inlet velocity, U_{in}. Anderson and Moffat power one heater at a time and measure the temperature of all the heaters in the array. From this they compute an adiabatic heat transfer coefficient for each heater from

$$h_{ad} = \frac{Q}{A(T_h - T_{ad})} \tag{26}$$

where T_h is the measured temperature of the heater when it is powered and T_{ad} is the temperature of the heater when it is not powered, but all other elements in the array are powered. A comparison of our simulation results with some of Anderson and Moffat's experimental data is shown in Figure 7. The data are the average values of adiabatic heat transfer coefficient for the last five rows in the array. At nearly all data points in the comparison the computed heat transfer coefficients are less than the corresponding measured values. The largest error of approximately 50 percent occurs at the highest velocity and smallest channel spacing ($H/B = 1.5$).

We have no firm explanation for the discrepancy between the simulation results and these data sets. Both the velocity profile and the turbulence model are suspect. The geometry of the heated array is similar to those of the other experiments. (cf. Table 1), and the Reynolds number range is also comparable. From this one can rule out that drastically different fluid dynamic effects are present in the Anderson and Moffat experiment. One significant difference is that the coolant used by Anderson and Moffat is air (Pr = 0.7) and the coolant for all other data sets is water (Pr = 7). The data in Figure 7 might be an indication that our turbulence model underpredicts the molecular tranport.

CONCLUSIONS

We have presented a simplified model of convective heat transfer from arrays of heaters that simulate electronic packages on a printed circuit board. The model involves solution of the three-dimensional energy equation in the fluid and the solid heaters (if the heaters protrude into the duct). The model uses an assumed form of the velocity profile in the gap between the top of the heaters and the top wall of the duct. We have developed an eddy-viscosity turbulence model by adapting eddy viscosity data presented by experimentalists. We have tested our model by comparing its predictions to the results of several heat transfer experiments.

The model has demonstrated limited success. For flush mounted heaters the agreement between the model and experimental data is quite good, with the largest discrepancies being about 15 percent. Since flush heaters are a poor physical model of board-mounted electronic components this success is of minor practical importance. For heated blocks that protrude into the flow the model predictions are not as reliable. Assumptions about the shape of the velocity profile have significant affect on the quantitative accuracy of the model. For the data of Garimella and Eibeck an assumption of a uniform velocity profile gives the best results, whereas an assumption of a fully-developed velocity profile does the best job of simulating the measurements of Heindel et al. We can make a plausible physical argument for the

success of the different velocity profile assumption in simulating these two experiments. Since the quantitative accuracy of the model depends on a user-specified velocity profile the model is less than an ideal design tool. Furthermore, the comparison between the simulation results and the experimental results of Anderson and Moffat are disappointing.

The results demonstrate the need for more sophistication in our model. We are currently studying ways of linking our depth-averaged flow model to the computation of the three-dimensional energy equation. This approach still requires information about the vertical (z-direction) velocity profile and a turbulence model. We are also investigating the formulation of a 2 1/2 dimensional CFD model, the complexity of which would lie somewhere between that of the depth-averaged model and a full three-dimensional CFD model.

ACKNOWLEDGEMENTS

We are grateful for the financial support of Intel Corporation, and the technical advice of Dr. Muralidhar Tirumala of Intel.

REFERENCES

Anderson, A. M. and R. J. Moffat, (1990), "A new type of heat transfer correlation for air cooling of regular arrays of electronic components", in *Thermal Modeling and Design of Electronic Systems and Devices*, vol. HTD-Vol. 153, R. A. Wirtz and G. L. Lehmann (ed.), ASME, New York.

Anderson, A. M. and R. J. Moffat, (1992a), "The adiabatic heat transfer coefficient and the superposition kernel function: Part 1—Data for arrays of flatpacks for different flow conditions", J. Elect. Pkgng., **114**(1): 14-21.

Anderson, A. M. and R. J. Moffat, (1992b), "The adiabatic heat transfer coefficient and the superposition kernel function: Part 2—Modeling flatpack data as a function of channel turbulence", J. Elect. Pkgng., **114**(1): 22-28.

Burmeister, L. C., (1983), *Convective Heat Transfer*, New York, Wiley.

Garimella, S. V. and P. A. Eibeck, (1990), "Heat transfer characteristics of an array of protruding elements in single phase forced convection", Int. J. Heat Mass Transfer, **33**(12): 2659-2669.

Garimella, S. V. and P. A. Eibeck, (1991), "Fluid dynamic characteristics of the flow over an array of large roughness elements", J. Elect. Pkgng., **113**(4): 367-373.

Heindel, T. J., F. P. Incropera and S. Ramadhyani, (1992), "Liquid immersion cooling of a longitudinal array of discrete heat sources in protruding substrates: I—Single-phase convection", J. Elect. Pkgng., **114**(1): 55-62.

Incropera, F. P., J. S. Kerby, D. F. Moffatt and S. Ramadhyani, (1986), "Convection heat transfer from discrete heat sources in a rectangular channel", Int. J. Heat Mass Transfer, **29**(7): 1051-1058.

Laufer, J., (1954), "The structure of turbulence in fully developed pipe flow", National Advisory Committee for Aeronautics, Report no. 1174, 1954.

Mahaney, H. V., F. P. Incropera and S. Ramadhyani, (1990), "Comparison of predicted and measured mixed convection heat transfer from an array of discrete sources in a horizontal rectangular channel", Int. J. Heat Mass Transfer, **33**(6): 1233-1245.

Patankar, S. V., (1980), *Numerical Heat Transfer and Fluid Flow*, Washington D.C., Hemisphere.

Recktenwald, G. W. and P. O. Butler, (1991), "Depth-averaged modeling of convective heat transfer from printed circuit boards", ASME Winter Annual Meeting, Paper No. 91-WA-EEP-36, Atlanta, GA.

Techo, R., R. R. Tickner and R. E. James, (1965), "An accurate equation for the computation of the friction factor for smooth pipes from the Reynolds number", J. Appl. Mech., **32**: 443.

Tennekes, H. and J. L. Lumley, (1972), *A First Course in Turbulence*, Cambridge, MA, MIT Press.

van Driest, E. R., (1956), "On turbulent flow near a wall", J. Aero. Sci., **23**: 1007-1011.

White, F. M., (1986), *Fluid Mechanics*, New York, McGraw-Hill.

Wirtz, R. A. and W. Chen, (1992), "Laminar-transitional convection from repeated ribs in a channel", J. Elect. Pkgng., **114**(1): 29-34.