Abstract

Present-day interest in the thermal analysis of electronic circuit boards arises mainly because of the failure of such components as a result of thermal fatigue. A thermal/structural ANSYS model was integrated in this study to enable the predictions of the temperature and stress distribution of vertically clamped parallel circuit boards that include series of symmetrically mounted heated electronic modules (chips). The board was modeled as a thin plate containing heated flush rectangular areas representing the heat generating modules. The ANSYS model was required to incorporate the effects of mixed convection on surfaces, heat generation in the modules, and conduction inside the board. Appropriate convection heat transfer coefficients and boundary conditions resulted in a temperature distribution in the board and chips. Then structural analyses were performed on the same finite element mesh with structural elements capable of handling orthotropic material properties. The stress fields were obtained and compared for the two models possessing different fiber orientations.

Introduction

Current interest in the thermal analysis of electronic circuit boards arises mainly because of the failure of such components as a result of thermal fatigue. The cooling of the component boards in many applications is the result of the forced or mixed convection air flow. Many studies in the area of convection cooling of boards have been done in the literature but few studies have related the flow field to the structural integrity of the boards. The initial phase of evaluation of stresses involves the understanding of mixed convection flows that removes heat from the boards. A detailed description of steady state and transient mixed convection studies can be found in [1, 2, and 3]. In the latter, Khalilollahi and Sammakia investigate the effects of a weak pressure gradient aiding the buoyancy-induced flow adjacent to discrete heated components. That study resulted in the prediction of flow parameters using a two-dimensional numerical model. It was found that the aiding pressure field results in improved transport, as expected. However, under some circumstances significant temperature overshoots were observed during the transient. Such overshoots could be a concern since component temperatures may exceed their allowed design temperatures during the transient. The methodology of the work mentioned above and follow-up work is important in this study since it would facilitate the calculation of heat transfer coefficients under different flow and thermal conditions.

Modeling Issues

Generally, an unconstrained, linear isotropic material subject to a temperature field for which \( \nabla^2 T = 0 \) may deform but will remain unstressed [4, 5]. As a result, in all other cases one expects thermal stresses as a result of non-uniform temperature distribution.

A major concern in such analysis when applied to a composite circuit board, on which series of modules are mounted, is the initial need to estimate correctly the mixed convection heat transfer on the board surface. Areas between modules can be thermally affected by the board conduction, heat originated by the modules and components, and convection cooling by air. The convection coefficients used in this study as mentioned were based on the previous experimental and computational methodology that predicted cooling by aiding or opposing flow over discrete uniform heat flux components. The simulated electronic modules were mounted on vertical parallel boards made of composite material. Each board included heated flush area segments representing chips or modules with finite thermal capacity. The convection field was composed of an upward buoyancy induced flow generated by discrete heated components, and an opposed weak down-flow resulting from an adverse pressure field. This circumstance may arise in electronic packaging and in regions where cooling is intended to rely on natural convection. However, an adverse pressure field may occur due to forced convection cooling in other parts of the system. This resulted in estimating the Nusselt number (Nu) of a module as a function of module’s Reynolds
Physical/F.E. Model

Our physical model for the printed composite board is assumed to have dimensions of length L, width w, and thickness t (Figure 1), made of a common composite material. The board contains a number of modules. Each module is represented by a rectangular area with heat generation. In particular the board is made of an E-glass epoxy [6] with the assumed properties as follow: Ex=60.72 Gpa, Ey=24.8 Gpa, density=2570 kg/m³, Poisson’s ratio=.26, CTE= 0.5 µm/m-C, and thermal conductivity=1.3 W/m-K. The board has dimensions: L=20 cm, w=7.5 cm and t=0.03 cm. Four modules (chips) each generating 1 watt were mounted on the board as shown in Figure 1. Each module has an area of 1.5cmX4cm. The entire surface is under mixed convection cooling that creates discrete and spatially averaged coefficients of heat transfers for a typical air flow Reynolds No of 200 to 1400, as presented in Figures 2 and 3. The thermal finite element model representing the circuit board was rendered in ANSYS using shell quad elements. The 3-D finite element shell model first provided temperature distribution using the convection thermal boundaries and element #57 in ANSYS. Then a subsequent change to a structural/static model with element #63 lead to the determination of thermally induced stresses. Structural boundaries include a vertically clamped boundary representing the inserted left (pinned) edge of the board and three other unconstrained edges. The present model uses average heat transfer coefficients that are based on the 2-D flow models as mentioned above. Figure 2 also summarizes the structural boundary conditions for our F.E. model. Future studies are planned to solve the three-dimensional flow/thermal fields in order to render more accurate heat transfer coefficients for the board’s surface.
Results of FE Analysis

The ANSYS modeling were done to render findings about the location and the values of maximum stresses important to the structural failure of the board. Figure 4 indicates the temperature distribution in the board for a sample run when Re = 200. As expected, the temperature of the downstream chip and the area associated with it is
larger than the lower (upstream) one. For the range of Reynolds numbers used in this study (200-1400) it was observe that the difference between the maximum temperatures of the upper/lower chips ranged from 4.5-11%. It is to be noted that the reference temperature (air) is assumed to be 0°C at which the board and modules are stress free. Also Figure 5 shows the max temperature (occurring in the upper chip) for the range of applied Reynolds numbers. As observed, the temperatures are less sensitive to Re, as flow (and Re) intensifies.

\begin{itemize}
\item \textbf{Figure 4} - Sample Temperature Distribution (Re=200)
\item \textbf{Figure 5} - Maximum Temperature Variation with Reynolds Number
\end{itemize}
The location and value of max shear intensity stress (Sint) responsible for a potential crack or damage to the board was found as shown in Figure 6. By observing Figure 6, and as confirmed by the plot of normal stress (Sx) in Figure 7, two conclusions can be reached:

1. The location of maximum stress is at the top edge of the board directly above the chips, which perhaps can be somewhat unusual.

2. An appreciably lower stress was found at the chip areas and around their close vicinities.

\[
\begin{align*}
S_X &= \text{(avg)} \\
N_X &= .946E-04 \\
S_{11} &= -.367E+07 \\
S_{12} &= .700E+07
\end{align*}
\]

Figure 6 - Maximum Stress Intensity (Sint) Distribution (Re=200)
Figure 7 - Normal Stress (Sx) Distribution (Re=200)

Figure 8 represents the differences of stress fields between a board with horizontally oriented fibers and that made with vertical fibers. The differences seem to be significant, (average for the range is about 90%) which indicate the importance of optimal alignment of material fibers in this application.

Figure 8 - Comparison Graph for Maximum Stress Intensity

Finally it was discerned that interestingly the overall displacements is minimally sensitive to the fiber orientation as evident in Figure 9. Some minute differences tend to appear around the flows with higher Re range.
Conclusion

The present work has shown that ANSYS can be used successfully to render important stress evaluation for design or failure analysis in conjunction with a proper methodology that can predict coolant flow heat transfer characteristics. The methodology in finding coefficients of heat transfer was based on the two-dimensional solution of the airflow and thermal field in previous work. The results indicated that:

1. Upstream chips have temperatures in the range of 4.5-11% higher than the downstream units.
2. Chip temperature is less sensitive to Reynolds No. as Re increases.
3. Failure stresses may happen at the top (trailing) edge of the board.
4. The stresses around the chip areas are appreciably lower than the maximum stresses found on the top/bottom edges.
5. Significant value differences were seen for maximum shear intensity stress between the composite board with horizontally aligned fibers and the board with vertically arranged fibers.
6. Displacement field was found to be almost insensitive to the fiber alignment but sensitive to the Reynolds No.

References: