Finite Element Stress Evaluation Of a Composite Board using 2-D and 3-D Convection Models

Amir Khalilollahii
Pennsylvania State University, The Behrend College

Abstract
Thermal fatigue and high levels of thermal stresses in electronic circuit boards have been of great concerns to designers in a wide range of applications such as avionics and computers. Fluid/thermal/structural models were 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. Initially two convection models, a 2-D and a 3-D model were incorporated to predict the heat transfer coefficients that strongly affect the temperature distribution in the board and chips. Then ANSYS models were used to incorporate the effects of mixed convection on surfaces, heat generation in the modules, and conduction inside the board. Then analyses were performed by ANSYS using structural elements capable of handling orthotropic material properties. The stress fields were obtained and compared for the two models possessing different convection models.

Introduction
Thermal analysis of circuit boards is of much interest 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 remove heat from the boards. A detailed description of steady state and transient mixed convection studies can be found in previous work such as [1 -3]. In one of these studies [3], the effects of a weak pressure gradient aiding the buoyancy-induced flow adjacent to discrete heated components was studied. 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 were important in the previous study [4] where failure stresses for a model of a composite board were analyzed. In that study, under different fiber orientation and flow conditions, the heat transfer coefficients were obtained by a 2D conjugate thermal model followed by 3D ANSYS stress analysis using shell elements. It was found that the maximum failure stresses decrease as the fibers orient with the cross flow direction. The present study is basically an extension of the previous work where a 3D conjugate model for the convective flow was incorporated in the evaluation of heat transfer coefficients.

Modeling Issues
A linear isotropic material subject to a temperature field for which \( \nabla^2 T = 0 \) may deform but will remain unstressed [5,6]. As a result, in all other cases one expects thermal stresses as a result of non-uniform temperature distribution.

The initial need to estimate correctly the mixed convection heat transfer characteristics of the flow and the board is a major concern since the stress components are very sensitive to the temperature gradients. Areas between modules can be thermally affected by the board conduction, heat originated by the modules and components, and convection cooling by air. Certain areas away from the chips also may play an important roll in creating sharp temperature gradients. Thus a more realistic 3D model was deemed to be important in the evaluation of the failure stresses. The convection coefficients used in this were based on the 2D previous computational methodology that predicted cooling by aiding or opposing flow over discrete uniform heat. In
addition, 3D models of conjugate heat transfer in FLUENT were used in predicting the local heat transfer coefficients. The convection field was composed of an upward buoyancy induced flow generated by discrete heated components, and an aiding weak flow resulting from an aiding pressure field. This circumstance may arise in electronic packaging and in regions where cooling is intended to rely on natural convection. However, an aiding or adverse pressure field may occur due to forced convection cooling in other parts of the system.

The simulated electronic modules were mounted on vertical parallel boards made of a common composite material. The material fibers were aligned normal to the flow to produce the least amount of stress [4]. Few rectangular areas were assigned a typical heat generation rate representing chips or modules attached to the board.

The coupling of the models as mentioned above resulted in the evaluation of maximum shear intensity stress for a range of flow intensities (Reynolds numbers). Also this study will compare the results for displacement, temperature, and temperature gradient fields by both 2D and 3D convection models in order to gain more insight into the level of differences in these models.

**Physical/F.E. Model**

The 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 with the assumed properties as follow: Ex=60.72 Gpa, Ey=24.8 Gpa, density=2570 kg/m³, Poisson’s ratio=.26, CTE= 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 100 to 3000. 2D and 3D FLUENT conjugate heat transfer models were incorporated to accurately obtain the local heat transfer coefficients on the board. Different pressures at the bottom and top surface of the flow regime (volume) and the bouncy forces generated the mixed convective flow. As Figure 1 shows the board surface was divided to smaller rectangles for which the heat transfer coefficients were averaged. These discretized and averaged coefficients were input to a series of ANSYS models in which thermal and stress characteristic were predicted.

![Figure 1. Composite board with structural and thermal BC](image-url)
The thermal/structural ANSYS FE model representing the circuit board was rendered using shell quad elements (shell57). Shell models with average convection coefficients from the 2D and 3D models provided temperature distribution. Then a subsequent change to a structural/static model with element #63 resulted in the distribution of stress components and displacements. Structural boundaries include a vertically clamped boundary representing the inserted (or pinned) left edge of the board and three other unconstrained edges (Figure 1). The question of the thermal/structural effects of 2D vs. 3D convection models on the boards with oblique fiber orientations may be of importance and can be addressed in future.

**Results of FE Analysis**

Two sets of ANSYS models, incorporating namely 2D and 3D convection models, were solved and compared to find the differences in (1) the temperature distribution and maximum temperature, (2) failure stress, (3) maximum displacement, (4) and the temperature gradient regarding each set.

Figure 2 indicates the temperature distribution in the board for a sample run, Re ~3000. As expected, the temperature of the downstream chip and the area associated with it is larger than the lower (upstream) one. Also slightly larger blue areas are observed in Figure 2(b) indicating larger temperature gradient in the 2D model. It is noted that the reference temperature (air) is assumed to be 0 C at which the board and modules are stress free.

Figure 3(a and b) indicates the difference in temperature gradients between the two models. 3D model shows slightly lower levels of temperature gradient compared to the 2D model (about 3 percent). Consequently lower levels of stress are expected in the 3D model.

Figure 4 clearly shows this behavior. The maximum stress intensity for 3D model is about 13% lower than the value for the 2D model. The location of max temperature gradients is different between the two models as seen in Figure 3(and b) . However the location of maximum shear intensity is the same for both models as evident in Figure 4.
The max temperature plot (Figure 5) demonstrates the effects of different convection patterns for each model. The trends seen for 2D and 3D model is followed by the behavior of maximum displacement as shown in figure 6. In both plots the curves almost follow each other’s trend with almost a constant difference. However this behavior is not seen for the stress plot, as shown in Figure 7. As we notice the stress levels are almost identical when flow is very close to pure natural convection (Re~100). This difference grows...
and reaches about 13% at Re~3000. The probable cause of this uneven difference can be explained by the fact that the gradients are non-uniform and do not follow the same pattern as temperatures do. This fact is somehow evident in Figure 8, where at the bottom edge a significant difference is observed between the 2D and 3D results for the temperature gradients. This is of importance since on this edge the maximum stress intensity and the potential for cracking can happen.
It is interesting to observe that, (1) the temperature and displacement maxima have been higher in 3D model compared to the 2D. This is opposite to the trend for the failure stress. And (2), failure stresses are appreciably lower at the chip surfaces (areas) and around their close vicinities.
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 and three-dimensional coefficients as found by solution of conjugate heat/flow models using FLUENT. The comparison of the effect of 2D and 3D convection models on the structural parameters revealed some notable differences. Most important of all was the opposite trend seen in the stress intensity vs. Re. 3D models resulted in prediction of lower stress field while this was not the case for the temperature and displacement fields. The study indicated that the failure stresses happen at the bottom edge of the board in both 2D and 3D convection models.

References


