Q&A Session from Webinar:
Reducing PCB Re-Spins

Q: What level of accuracy can be obtained from the 3 different thermal models 2resitor vs Delphi vs general model?

A: Generally speaking, a detailed component (where all the geometry is explicitly represented) is the ultimate in modeling techniques. While it incurs a largest solution time penalty (to resolve all of the objects) it can predict junction temperatures to within experimental error in most applications. The Delphi modeling approach is much simpler in terms of object counts and typical achieves results within 5% of the detailed component representation. A 2-Resistor model is simpler still and typically achieves results within 15% of the detailed component representation.

Q: I assume that 0 air flow is also simulated in this modelling?

A: Natural convection and conduction only calculations are fully supported.

Q. Has the effect of high heat source components such as processors been included, vis-a-vis forced convection on adjacent components?

A: Yes, the software is solving a full 3D dimensional problem. Conduction, convection, and radiation are all included in the calculations. The example shown in the seminar where the smaller component was moved downstream of the larger component shows this situation nicely. The downstream component is cooled by warmer air from the upstream component which causes it's junction temperature to increase (or in the terminology of this seminar, the maximum allowable power for that component decreases)

Q: Is there a demo version available to evaluate?

A: A fully functional version of the software is sometimes provided for a limited time. Contact your local Mechanical Analysis Division office for details.

Q: Are you aware of any plans for Altium support?

A: Direct interfacing with Altium is not supported currently. The workaround is to work with IDF files and manually extracted monotone images that show the copper distribution on each layer. A decision on supporting a direct interface in the future has not be made.

Q: How does the SW work with high current traces on the board which heat up itself?

A: Joule heating effects can be inputted into FloTHERM in the form of current, voltage, and resistivity after import of the board geometry. Not supported in FloTHERM PCB.

Q: What MCAD software can FloTHERM interface to?

A: All of them! FloTHERM has the ability to direct import ProE, Solidworks, and Catia files. Neutral file formats such as step, sat, iges, STL, and DXF are supported as well. Virtually all MCAD tools support export of at least one of these file formats.

Q: Can FloTHERM import mechanical housing drawings to automatically define environment settings?

A: Yes. See answer above.

Q: What about interfacing with PADS Layout?

A: Direct interfacing with PADS is not supported currently. The workaround is to work with IDF files and manually extracted monotone images that show the copper distribution on each layer. A decision on supporting a direct interface in the future has not be made.

Q: Which version has this capability?

A: FloTHERM PCB v5.1 or later has the capability of importing EDA layout data and solving for maximum power.

Q: Can layout files done by other EDA tools be imported to FloTHERM PCB? Will the vias be included in the conductivity map computation?

A: Yes, but not as directly as we've seen today. The workaround is to work with IDF files and manually extracted monotone images that show the copper distribution on each layer. Inclusion of the vias works in the same way as the metallic layers. A monotone image is generated showing all vias and a thermal conductivity map is derived from that based on user inputs for resolution, plating and fill materials.

Q: How are filled not filled vias contribution captured in heat transfer?

A: Vias are defined with or without a fill material and the net effect of the vias is included in the thermal conductivity map computation

Q: Can we use it for substrate also?

A: The image processing has a valid use for substrates as well provided one has extracted the image files manually from the EDA tool. The resultant board would then be manipulated in FloTHERM to form one part of the package model.