

# Sauna™ Quick Tour #2: Circuit Board Modeling

## About Sauna Quick Tours

The Sauna Quick Tours let you preview the features provided by the Sauna thermal modeling program. You will create actual models, just like an experienced Sauna user. The documentation explains all modeling steps, so no background with Sauna is required. However, as the name implies, the Sauna Quick Tours are not detailed introductions to the software. Many important topics are skipped over or discussed briefly. To become a skilled user, you need to work the tutorial exercises found in the Sauna User Manuals. Many of these exercise are also available to prospective users on the [thermalsoftware.com](http://thermalsoftware.com) website.

## Overview of Quick Tour #2

You will be starting with the simple model shown in Figure 1. This is just a DPAK-package device on a single layer board. With no heatsinking provisions and 1W of dissipation, you can expect that the device will be rather warm.

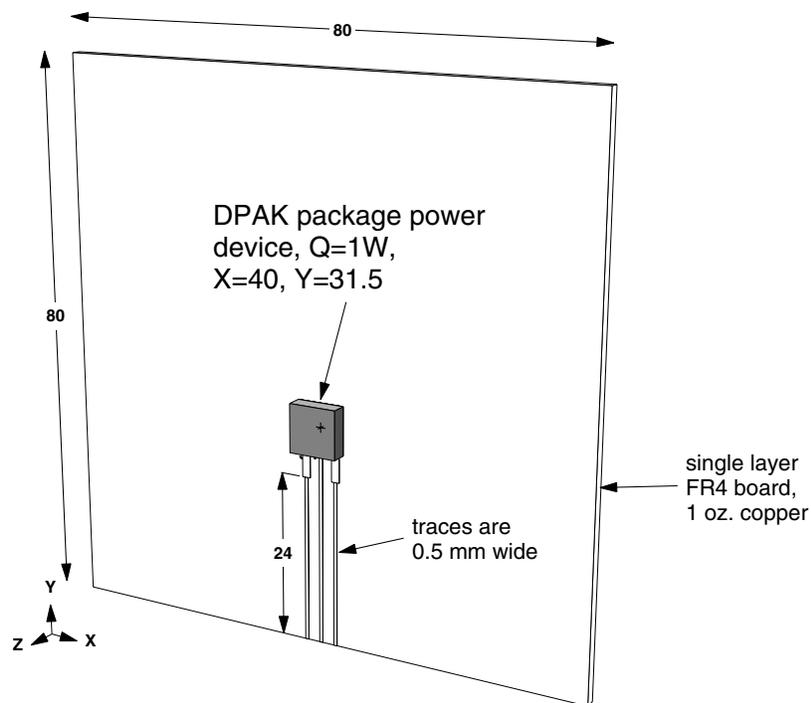


Figure 1: DPAK package device on single layer board

After creating the basic model, you will investigate the three variations shown in Figure 2. These types of what-ifs are very easy to perform with Sauna.

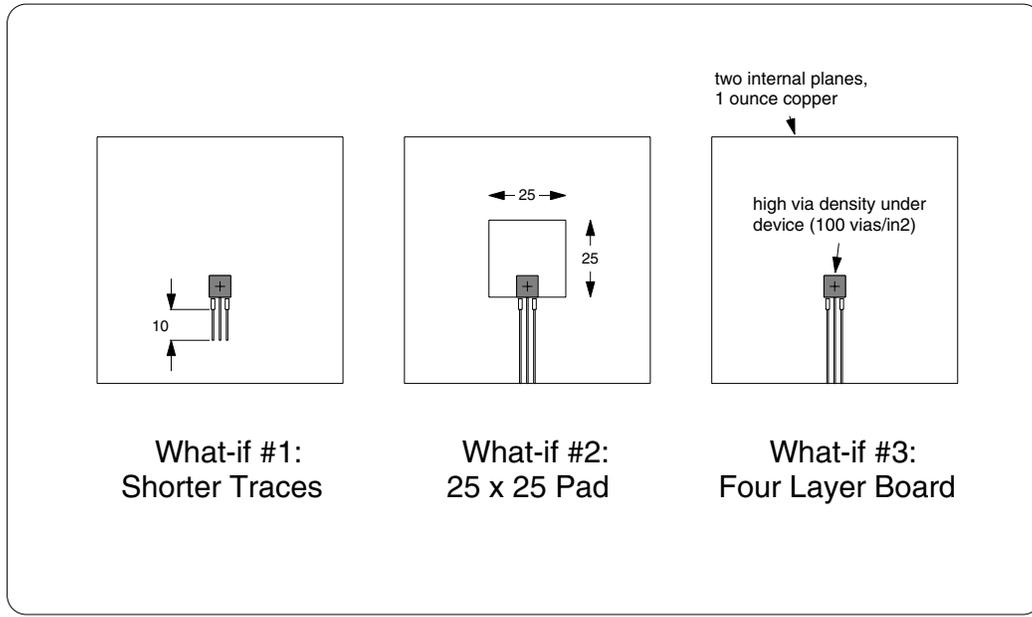


Figure 2: What-if's for DPAK-only board

After the DPAK what-if's, you will add components with different package styles, as shown in Figure 3. Sauna includes a library of over 150 packages, so a number of different component types can be handled.

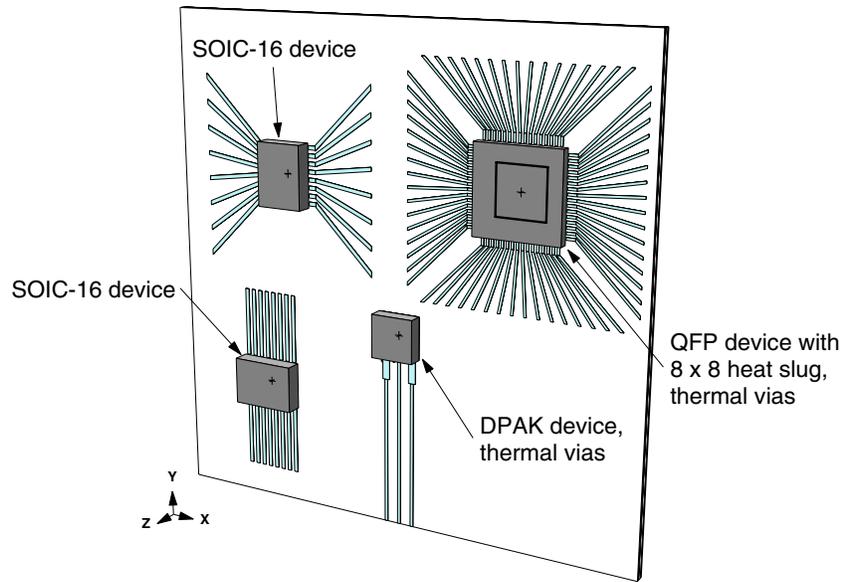


Figure 3: Four layer board with multiple component types, modeled in 2nd half of exercise

You will surely notice that the components are not interconnected with traces. For thermal modeling, the first 10 or 15 mm of traces are most significant, particularly when working with multilayer boards. While it is certainly possible to connect all traces between components, it is only really necessary when components are closely spaced. (More on these effects later.)

## Creating a circuit board assembly

Before starting the exercise, make sure that the error beep on your computer is turned on (not muted).

You will begin by creating a circuit board assembly which represents the laminate (dielectric) layer:

**<F12 Root Menu> → Model → Assembly → Circuit Brd → hit <Enter> to skip → Rectangle  
→ Vertical XY → "80,80" (do not type the quotes) → .062"/1.57mm → (0,0,0) → FR4**

You will reach the Copp Layers menu:

```
COPP LAYERS
>1 Zer/Lam Only
2 One
3 Two
4 Three
5 Four
6 Six
7 Eight
8 Enter Layers
```

Since you want to create a laminate only layer, you will be using "Zer/Lam Only", which stands for "zero copper layer/laminate only". The other selections are used with the planar board modeling method, which is used for quick simulations. Complete creation of the board assembly with:

**Zer/Lam Only → .025"/0.64mm → One oz. → None**

Sauna will create an array of node and resistors, framed by thick dashed lines which represent the assembly limits.

## Adding traces and pads

Begin creating the traces and pads with:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Board Side  
→ click dashed outline of board → Component → One oz. → Pad Library**

You will reach the Pad Library menu:

```
PAD LIBRARY
1 DPAK's
2 TDSON/TOLL
3 Single/Dual
4 Quad
5 Diodes
6 LED's
7 Resistor
```

Sauna has over 200 different package styles in the library, so you do not need to define individual rectangles for DPAK pads. You just need to pick the component type and the placement point. To see an overview of the different library categories, click the Picture button which is found just underneath the Pad Library menu (to the right of the Menu Ref button):

**click**

A window will pop up showing the different package categories, as shown on the next page.

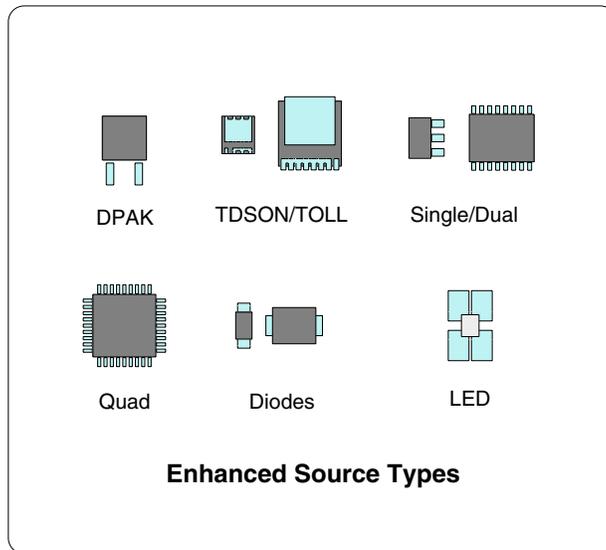


Figure 4: Package categories

Click the Done button to clear the picture, then continue with:

**DPAK's → DPAK → 2 → Same As Body → Enter → "24" (do not type the quotes)  
 → 0.50mm/20mil → Yes → 0 Degrees → Coords/Trap → "40,31.5" (do not type quotes)**

The pads and traces will be created as shown in Figure 5, all with a single string of commands.

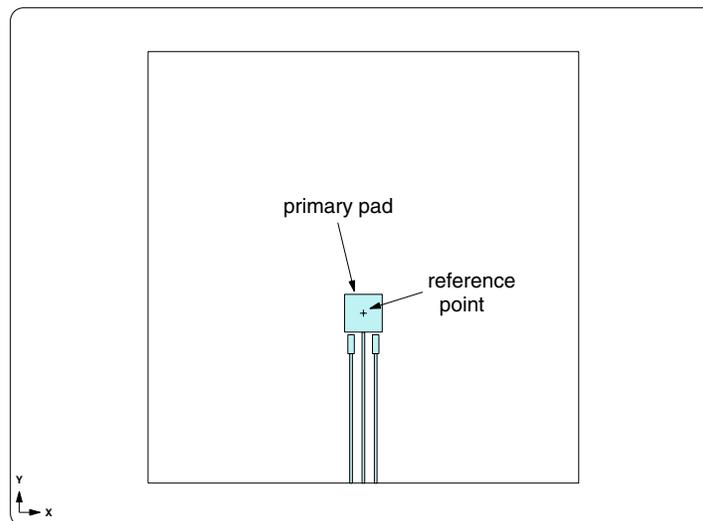


Figure 5: DPAK traces and pads

Note that there is a reference point at the center of the primary pad, which will be used to position the heat source.

### Adding the heat source

Now you are ready to add an "enhanced" heat source. An enhanced heat source allows for heat transfer through the heat slug, as well as through individual leads and the top of the component.

Begin with:

<F12 Root Menu> → Model → Heat Input → Enhanced Src → DPAK's → "1" → "S1"  
→ DPAK → 2 → Typical

You will reach the R\_Lead\_Pad menu:

```
R LEAD PAD
>1 Typ-1500 C/W
2 Enter Resis
3 Enter R_para
4 No Lead Conn
```

The R\_Lead\_Pad menu is used to specify the resistance between the enhanced source junction and the lead pads. The problem with  $R_{\text{junct-to-lead-pad}}$  is that it is generally not on the datasheet and is rather difficult to obtain. But Thermal Solutions has determined that  $1500^{\circ}\text{C}/\text{W}$  is a reasonable typical value to use, primarily based on published data from Infineon and Amkor.

Finish creating the enhanced source:

**Typ-1500 C/W → Ref Point → *click reference point***

The enhanced heat source will be created. If you look closely at the screen, you will see that there is an additional assembly. This is the body assembly associated with the heat source.

To better see the body assembly, you will switch to a shaded perspective view. For commonly used commands, there are a series of buttons at the lower-left of the Sauna window. The right button in the top row is for switching to a perspective view. Change the view with:

**click** 

Use the right button in the third row to activate shade mode:

**click** 

The screen will be as shown in Figure 6:

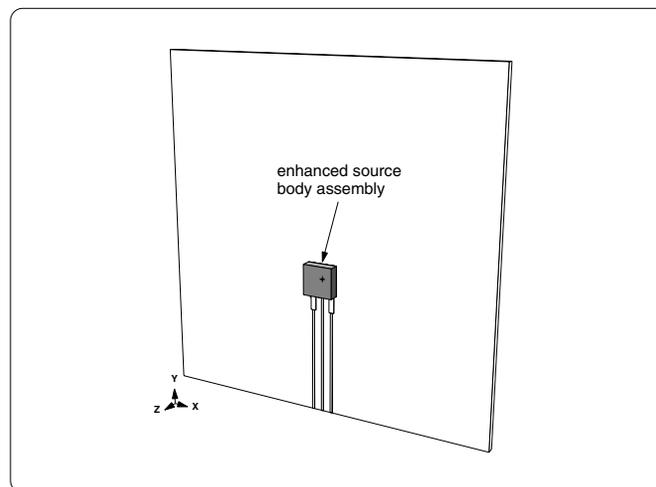


Figure 6: Enhanced heat source with body assembly

Return to a front view and get an Info report for the heat source:

**click** 

<F7 Info> → Trap → Node → Heat Source → ***click red S1 source node***

The report will be displayed on the screen. On the second page of the report, a variety of important information is provided. For example, you can see that  $R_{jc} = 1.80^{\circ}\text{C}/\text{W}$  and  $R_{\text{junct-to-top}} = 37.94^{\circ}\text{C}/\text{W}$ . The last section shows that there are two lead pad connections with  $R_{\text{junct-to-lead-pad}} = 1500^{\circ}\text{C}/\text{W}$ . Clear the report from the screen before continuing.

For best accuracy, you should align the mesh of nodes and resistors to the heat source.:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Align Mesh → Heat Source  
→ 4 Node Conn → *click on red heat source node* → All In Wind → USE**

The node and resistor mesh will be regenerated to match the dimensions of the DPAK source.

### Adding float resistors and a fixed node

At this point, you have defined the circuit board stackup and the heat input. However, the current model is invalid. No heat flow path has been established between the circuit board and the ambient temperature environment. A "fixed temperature" node is required to represent the room environment. The name "fixed temperature" comes from the fact that ambient nodes have temperatures which remain fixed throughout the temperature calculation. You also need "float" resistors to represent the convection and radiation. These are called "float resistors" because the resistance values are adjusted according to the  $\Delta T$  and convection conditions.

Create the fixed node and float resistors:

**<F12 Root Menu> → Model → Amb + Float → Isoltd->Fix → "Room Amb"  
→ Enter Later → Both Sides → All In Wind → USE**

The fixed node and float resistors will be created. To better see the new nodes and resistors, switch to a right view:

**click**

The screen will be as shown in Figure 7:

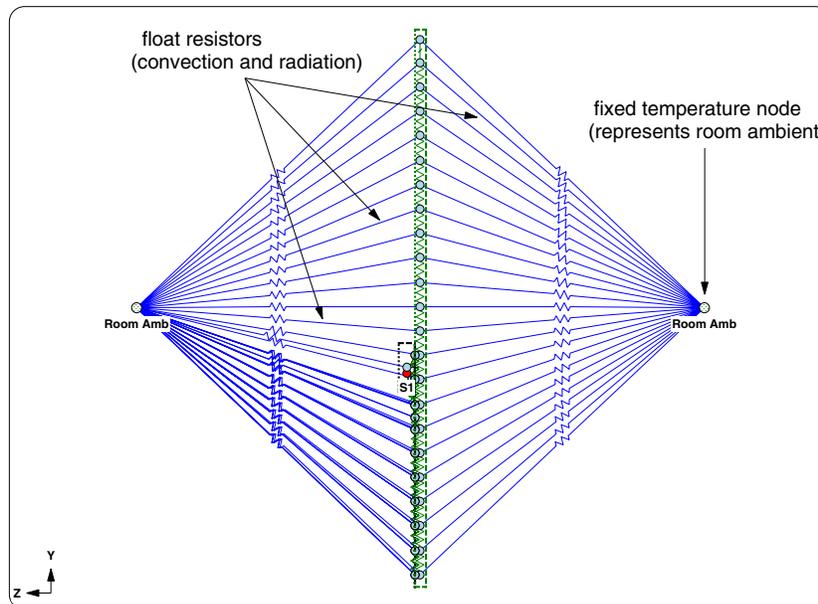


Figure 7: Fixed node and float resistors

Now that the heat flow path to ambient has been created, you can reduce screen clutter by making these nodes and resistors invisible. There's a button for this:

*click* 

Return to a front view:

*click* 

Calculate temperatures with natural cooling:

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

You should obtain  $T_{S1-junction} = 173.98^{\circ}\text{C}$ . Activate contours:

*click* 

With the temperatures contours turned on, you can clearly see that only the center of the board is warm, so the outer parts of the board are not contributing much to the cooling of the component.

### What-if #1: shorter traces

In thermal modeling, there is a rule of thumb that states that **traces longer than 25 mm (1 inch) have no effect on cooling**. Based on the thermal contours, that rule seems valid, because the ends of the traces are rather cool. In this section, you will shorten the traces, as shown on the left of Figure 2 on page 2.

It is easy to change trace length. However, it will be necessary to delete the float resistors and ambient nodes that you created earlier. Sauna can do this automatically, provided that you click the Yes button when modifying the trace length.

First, turn off the contours:

*click* 

Begin modifying trace length with:

**<F12 Root Menu> → Edit → Plate/Board → Dimensions → Modify Y → Enter Delta → "-14" → End**

You will reach a grouping menu. Rather than clicking on each individual trace, you can define a region to quickly select all three traces. Figure 8 on the next page shows a typical grouping rectangle. The grouping rectangle does not need to be identical to Figure 8, just reasonably close. You will be digitizing two times to define the rectangle. However, please note that with Sauna digitize means "click and release". It's not hold the mouse button down and drag. With that in mind, finish changing the trace length:

**Select Regn → digitize lower-left corner of grouping rectangle in Figure 8  
→ digitize upper-right corner of rectangle → USE  
→ click Yes to accept delete of float resistors and ambients**

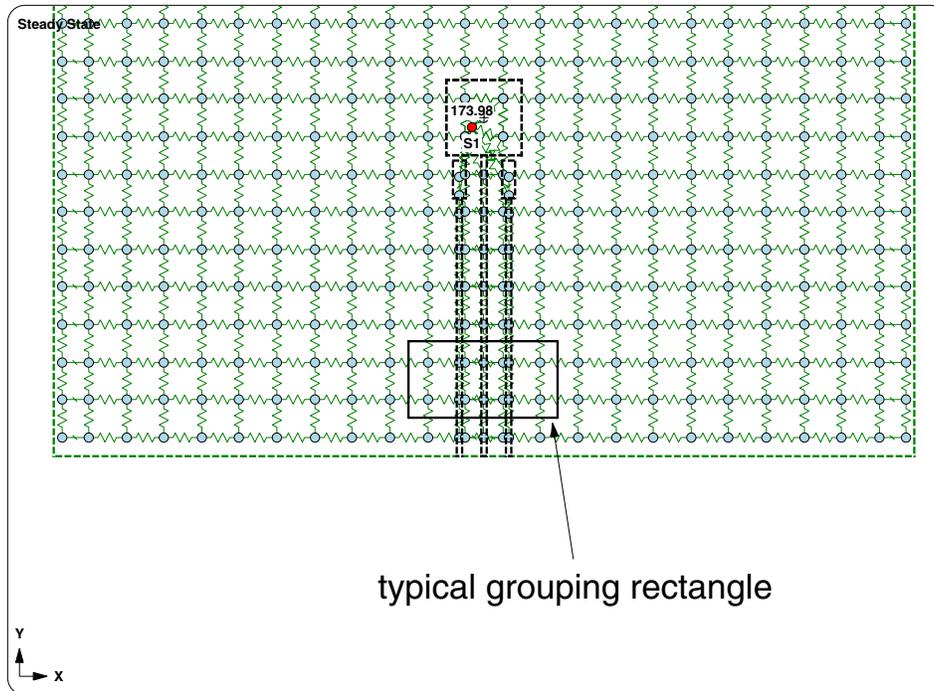


Figure 8: Grouping rectangle for modifying trace length

The traces will be shortened to match what-if #1 in Figure 2. As before, add float resistors:

**<F12 Root Menu> → Model → Amb + Float → Isoldt->Fix → "Room Amb" → Enter Later  
→ Both Sides → All In Wind → USE**

click 

Calculate temperatures:

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

With the shorter traces,  $T_{S1-junct} = 176.20^{\circ}\text{C}$ , a fairly modest increase of  $2.2^{\circ}\text{C}$ . In terms of the  $\Delta T$  above ambient, this is just a 1.5% increase. So for practical thermal modeling, you can use traces which are shorter than 25 mm and still obtain reasonably accurate results.

Use the Undo command to return the traces to the full length:

**<F12 Root Menu> → Edit → Undo → *click Yes button to undo Amb + Float***

**Undo → *click Yes button to undo edit assembly dimensions***

As before, you should see  $T_{S1-junct} = 173.98^{\circ}\text{C}$ .

### What-if #2: using a 25 mm x 25 mm pad

In a moment, you will enlarge the primary pad, so that the model is as shown in Figure 9 on the next page. But before enlarging, it is interesting to obtain a board stackup report:

**<F7 Info> → Assemblies → Brd Stackup → All → Current → Screen**

The report will show that there is one copper layer, identified as "Copp>1", and one laminate layer: "Lamin>1". You can use these layer names to manipulate model visibility. Clear the report, then isolate the copper layer:

**<F12 Root Menu> → Visibility → Isolate → Layer → Copp>1**

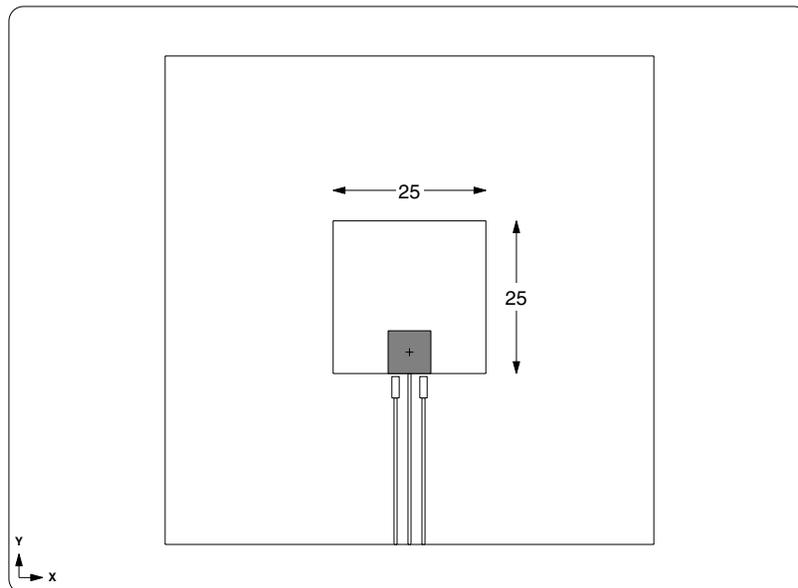
With this change, you will see just the DPAK pads and traces. Now use the Reshape command to resize the primary (large) pad:

**<F12 Root Menu> → Edit → Plate/Board → Dimensions → Reshape → To Rectangle  
→ Ref/Dx-Dy-DZ → *click upper-left corner of primary pad* → "-9,18" → Dx-Dy-Dz  
→ "25,-25" → Vertical XY → Select 1 → *click edge of primary pad* → USE  
→ *click Yes to accept delete of float resistors***

The pad will be enlarged. Turn on the board and switch to shade mode:

**click**  → **click** 

The model will be as shown in Figure 9:



*Figure 9: Model with 25 mm x 25 mm primary pad*

Since you deleted the float resistors and ambients, these need to be recreated:

**<F12 Root Menu> → Model → Amb + Float → Isold->Fix → "Room Amb" → Enter Later  
→ Both Sides → All In Wind → USE**

**click** 

Recalculate temperatures:

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

You should obtain  $T_{S1-junct} = 83.74^{\circ}\text{C}$ . With the large pads, temperatures are substantially cooler.

## Comparing with published results

For the comparison with published data, you will need to calculate the junction-to-ambient thermal resistance ( $R_{ja}$  or  $\theta_{ja}$  or  $R_{\theta-j_a}$ ). The equation is  $R_{ja} = (T_j - T_{ambient}) / Q$ . For the initial calculation with the minimum pads, you obtained  $T_{S1-junct} = 173.98^\circ\text{C}$ , so  $R_{ja} = (173.98 - 25.0) / 1 = 148.98^\circ\text{C}/\text{W}$ . For the 25 mm x 25 mm pads,  $T_{S1-junct} = 83.74^\circ\text{C}$  and  $R_{ja} = 58.75^\circ\text{C}/\text{W}$ .

Infineon Technologies has published a very useful document, "Thermal Resistance Theory and Practice" (available at infineon.com), which provides quite a bit of useful thermal data, including  $R_{ja}$  for DPAK package devices with different size copper pads. The thermal resistance provided by Infineon is quite close to Sauna's simulated values. For standard pads,  $R_{ja-Sauna}$  is within 3.5% of  $R_{ja-Infineon}$ . For the 25 x 25 pad,  $R_{ja-Sauna}$  is within 7.4% of  $R_{ja-Infineon}$ . So there is a good match with published data.

## What-if #3: four layer board

While the analysis of a single layer board is certainly instructive, boards with four or more copper layers are much more common. You will be converting the single layer board to the four layer configuration shown in Figure 10. As you will see, Sauna can easily model these types of boards.

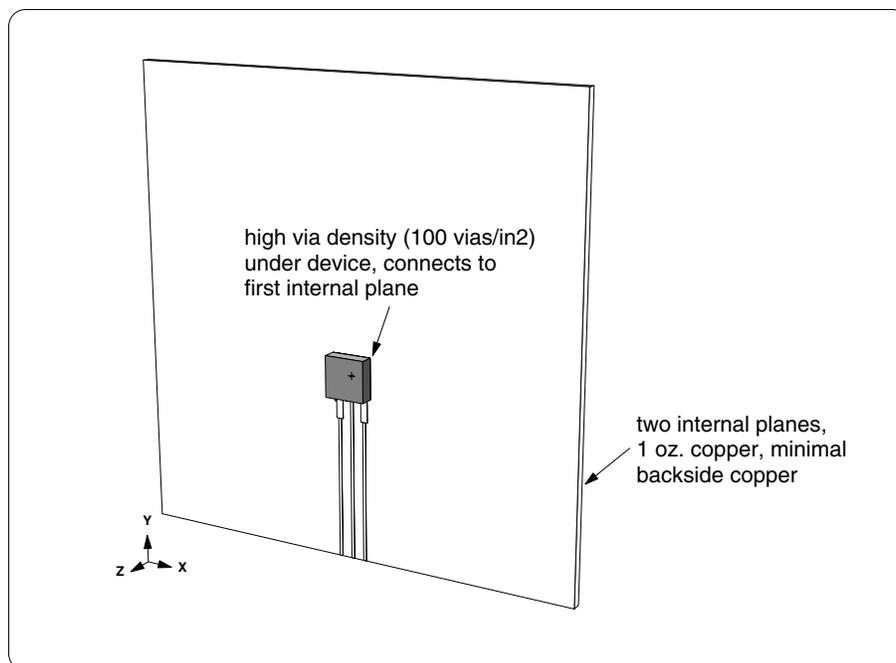


Figure 10: Four layer board

First, use the Undo command to restore the original pads:

**<F12 Root Menu> → Edit → Undo → click Yes button to undo Amb + Float**

**Undo → click Yes button to undo reshape**

**click** 

As before, you should see  $T_{junct} = 173.98^\circ\text{C}$ .

Before converting to a multi-layer stackup, you will need to delete the float resistors:

click 

**<F12 Root Menu> → Delete → Node → Fixed → All In Wind → USE**

Now you can use the Subdivide commands to create the stackup:

**<F12 Root Menu> → Edit → Plate/Board → Subdivide → Stackup → Board → Intern Layer Two → Uniform → One oz. → Typical → All In Wind → USE**

The new circuit board layers will be created. Get an Info report for the new stackup:

**<F7 Info> → Assemblies → Brd Stackup → All → Current → Screen**

The board stackup report shows 3 copper layers and 3 laminate (dielectric) layers. You used the "typical" option, so the laminate thicknesses are 0.396 mm, 0.711 mm and 0.396 mm (.016"/.028"/.016").

Internal planes improve cooling, even when there are no thermal vias. To see this, clear the report, then add float resistors and calculate temperatures:

**<F12 Root Menu> → Model → Amb + Float → Isold->Fix → "Room Amb" → Enter Later → Both Sides → All In Wind → USE**

click 

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

You should obtain  $T_{\text{junct}} = 66.94^{\circ}\text{C}$ , a dramatic reduction. Ground and power planes have an important impact on cooling, even when there are no thermal vias.

## Adding thermal vias

In a moment, you will modify the via density for the front laminate layer. Of course, it's very important that you modify the front laminate layer and not one of the other laminate layers. So you will start by isolating the front laminate layer.

Isolate the layer:

**<F12 Root Menu> → Visibility → Isolate → Layer → Lamin>1**

With this visibility change, you will see only the front laminate board assembly and the DPAK. All other laminate layers are turned off. All copper pads and traces are also turned off. To verify, switch to a top view in shade mode.

click  → click 

The top view shows only the DPAK and the front laminate layer. It is important to note that **Sauna links enhanced sources to the board assemblies, not to the traces and pads.** (If you delete the front laminate layer, the DPAK will be deleted automatically.)

While you just saw that internal planes without vias can improve thermal performance, it's safe to say that temperatures will be even cooler with vias underneath the heat source.

*Note that there is no need to delete float resistors when modifying vias. You must delete the float resistors whenever there is a change to the outer surface of the board stackup. But vias are an internal property of the laminate layers, so nothing changes on the outer surface.*

Switch to wireframe mode and return to a front view:

**click**  **→ click** 

Activate the display of via density:

**<F6 Setup> → Display → Node → Via Density → Vias/In2**

You will see "0i", for 0 vias per inch<sup>2</sup>, displayed across the board. Currently there are no vias.

Now you need to visualize the exact nodes to be modified. Activate the outlining of diepads:

**<F6 Setup> → Display → Node → Outlines → Enhan Diepad**

With this change you will see a red rectangle which indicate the limits of the diepad. For a DPAK, the outline is the same as the overall part, as it is assumed that the copper slug is the same size as the overall package (in reality, the slug is usually a bit smaller, but Sauna ignores this).

Now you can easily change the via density to "High":

**<F12 Root Menu> → Edit → Plate/Board → Board Props → Vias → Density → High (100)  
→ Node Group → Select Regn → *click upper-left corner of red outline*  
→ *click lower-right corner of red outline* → USE**

The via density will be changed to "100i" for the selected nodes. Also, Sauna indicates that "via density modified for 4 nodes". If you make a mistake, use **Edit → Undo** and try again.

The via density change is complete. Restore visibility and calculate temperatures:

**<F6 Setup> → Display → Use Default**

**click** 

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

With the vias, you should obtain  $T_{\text{junct}} = 50.46^{\circ}\text{C}$ , which is roughly 40% cooler than the no-via model.

Activate temperature contours:

**click** 

With the contours, you can see that the internal planes do a much better job of distributing the heat. So instead of a small hotspot by the component, a larger section of the board is warm, with a dramatic reduction in the peak temperature.

## Adding pads for the multi-component layout

In the remainder of the exercise you will add three components to achieve the layout shown in Figure 11. This will take around ten minutes. The procedure for adding these components is fairly similar to what you have done already. So if your time is limited, you can stop here. Otherwise, please continue on.

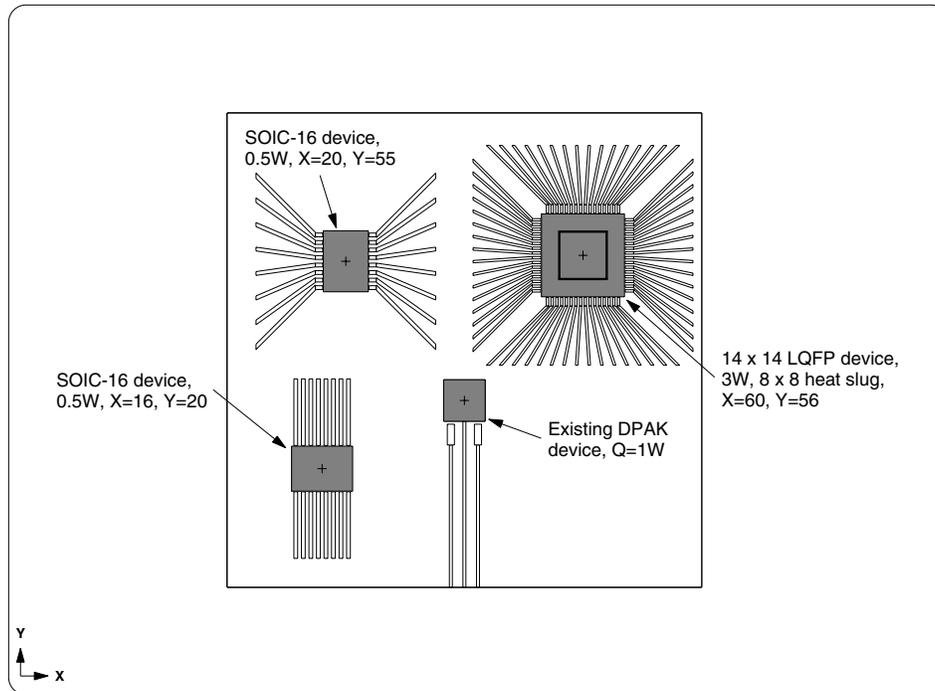


Figure 11: Multi-component board

In Figure 11, notice that the components are not connected to each other through the traces. As you saw earlier, traces which are longer than 10 mm have only a modest effect on cooling. So it follows that if the components are separated by more than 20 mm, there is only limited thermal interaction. Since all of the components in Figure 11 are separated by more than 20 mm, you do not need to connect components for the purposes of a thermal simulation. Obviously, it's a different situation when the components are close together. In this case, you would need to connect at least some of the traces and pads. Of course, Sauna can handle this, but there is additional work involved.

Start by creating the traces for the lower SOIC-16 device:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Trap Trc Pad  
→ click trace or pad on front layer → Pad Library → Single/Dual → SOIC Wide  
→ 16 → Detailed → 10 mm/0.4" → No → 0 Degrees → Coords/Trap → "16,20"  
→ click Yes to accept autodelete of float resistors**

The pads will be created. Next, you will create the pads for the upper SOIC-16 device. This device has flared leads and is rotated at 90 degrees. Create the pads:

**SOIC Wide → 16 → Detail/Flare → 10 mm/0.4" → No → 90 Degrees  
→ Coords/Trap → "20,55"**

Pads for the upper SOIC device will be created.

Now you will add pads for the quad flat pack (QFP) device. This device has an 8 mm x 8 mm heat slug. You can create the pad for the heat slug at the same time as the other traces and pads. Enter:

**<F2 Backup> → Quad → LQFP → 14 x 14 → 64 Leads → Detail/Flare → 10 mm/0.4"  
→ Yes → "8,8" → Coords/Trap → "60,56"**

The QFP pads will be created. You are ready to add the components.

## Adding the SOIC heat sources

The method for creating enhanced heat sources for dual and quad devices is similar to the method for DPAK's. But there are some additional options to consider.

You will begin with the lower SOIC device:

**<F12 Root Menu> → Model → Heat Input → Enhanced Src → Single/Dual → ".5" → "S2"  
→ SOIC Wide → 16 → Middle**

You will reach the Diepad Area menu:

```
DIEPAD AREA
>1 Typical/36%
 2 Typical/25%
 3 Specify
```

With this menu you are specifying the area of the die attach pad (diepad). The area of the diepad has a definite effect on heat transfer to the top of the component, as well as downward into the board. Unfortunately, there is a good chance that this parameter is unavailable, as it's almost never found on a datasheet. You may be successful (maybe) if you ask the component manufacturer, or you could cut open the part with a fine bladed saw. But if you don't have this information, just use the typical value (area of diepad is 36% of the component body area).

Continue with:

**Typical/36% → Typ (0.1 mm)**

Once again, you will reach the R\_Lead\_Pad menu:

```
R LEAD PAD
>1 Typ-1500 C/W
 2 Enter Resis
 3 Enter R_para
 4 No Lead Conn
```

Earlier in the exercise, you used "Typ-1500 C/W" with the DPAK. You also saw that heat transfer through the leads was not particularly significant. However, for components that do not have an external copper pad (heat slug),  $R_{\text{junct-to-lead-pad}}$  is an important thermal parameter. The recommended approach is to obtain  $R_{\text{junct-to-lead-pad}}$  by reverse engineering the  $R_{\text{junct-to-ambient}}$  ( $R_{\text{ja}}$ ,  $\theta_{\text{ja}}$ ) from the component datasheet. This is covered in detail in *Intermediate Exercise 3: More On Circuit Board Modeling* (available at [thermalsoftware.com/eval\\_exercises.htm](http://thermalsoftware.com/eval_exercises.htm)). Sauna has special tools for this task and it should only take a few minutes to obtain

$R_{\text{junct-to-lead-pad}}$ . For now, however, you will simply use the default of "Typ-1500 C/W" which will provide a good first pass result. Continue with:

**Typ-1500 C/W → Ref Point → *click reference point for lower SOIC device***

The component will be created and Sauna will indicate that "Enhanced heat source created, 16 lead connections".

It's interesting to get an Info report for the enhanced source:

**<F7 Info> → Trap → Node → Heat Source → *click S2 heat source***

On the second page, under "-- Junction To Lead Pad Resistance --", there is information on the number of lead pad connections and the resistance values, with  $R_{\text{average-to-pad}} = 1500^{\circ}\text{C}/\text{W}$  and  $R_{\text{parallel-to-pad}} = 93.75^{\circ}\text{C}/\text{W}$ . This is important information to check. (Although you have used uniform  $R_{\text{junct-to-lead-pad}}$  in this exercise, *Sauna let's you use a different resistance for every lead pad*, an important feature for modeling components with heatsinking leads.)

Clear the Info report, then add the upper SOIC component:

**<F12 Root Menu> → Model → Heat Input → Enhanced Src → Single/Dual → ".5" → "S3"  
→ SOIC Wide → 16 → Middle → Typical/36% → Typ (0.1 mm) → Typ-1500 C/W  
→ Ref Point → *click reference point for upper SOIC device***

The upper SOIC component will be created.

## Adding the QFP with heat slug

Begin creating the QFP source:

**Quad → "3" → "S4" → LQFP → 14 x 14 → 64 Leads → Heat Slug**

You will reach the Diepad Area menu:

DIEPAD AREA	
>1	Use Brd Pad
2	Typical/25%
3	Typical/36%
4	Specify

This Diepad Area menu is different from earlier in the exercise. When modeling a heat slug component, the diepad will usually have the same dimensions as the solder pad on the board. Continue with:

**Use Brd Pad → Typical**

You will reach the Gap To Board menu:

GAP TO BOARD	
>1	Typ (0.1 mm)
2	0.02mm/.001"
3	0.05mm/.002"
4	0.15mm/.006"
5	0.20mm/.008"
6	0.25mm/.010"
7	0.50mm/.020"
8	Enter

For the resistors to the heat slug pad, Sauna assumes a solder interface. But there is also a portion of the component with an air gap. This menu is used to specify that air gap. Finish with:

**Typ (0.1 mm) → Typ-1500 C/W → Ref Point → *click QFP ref point***

The QFP heat source will be created.

### Performing a preliminary calculation

You have placed all of the components. However, you still need to add vias for the QFP. But this is a good time to perform a preliminary calculation. First, add float resistors:

**<F12 Root Menu> → Model → Amb + Float → Isoltd->Fix → "Room Amb" → Enter Later  
→ Both Sides → All In Wind → USE**

*click* 

Now calculate temperatures:

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

You should obtain  $T_{S1-DPAK} = 66.72^{\circ}\text{C}$ ,  $T_{S2-SOIC} = 75.61^{\circ}\text{C}$ ,  $T_{S3-SOIC} = 78.20^{\circ}\text{C}$  and  $T_{S4-QFP} = 115.85^{\circ}\text{C}$ .

### Aligning to the QFP and adding thermal vias

At the moment, the circuit board is aligned to the DPAK. Since the QFP is dissipating 3W, and is likely a more expensive component, it makes sense to align to this component. (For aligning and adding vias, there is no need to delete the float resistors because the outer surface of the board stackup will not changed.)

Try aligning to the QFP:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Align Mesh → Heat Source  
→ 4 Node Conn → *click S4-QFP heat source node* → All In Wind → USE**

It won't work. An error screen will be displayed with the message "Lamin>1: non-uniform via density". Sauna is unable to realign a board with non-uniform properties. The via density is not uniform, due to the vias underneath the DPAK.

You will need to set the via density to "none" for the entire board. After clearing the error screen, enter these commands:

**<F12 Root Menu> → Edit → Plate/Board → Board Props → Vias → Density → None  
→ Entire Board → Group**

As you have seen earlier, you can select an assembly by clicking an edge or using a grouping rectangle. But you can also select with an assembly label, which you will do now. Complete the modification with:

**Select 1 → type "Lamin>1" and hit <Enter> (do not type the quotes) → USE**

Sauna will indicate that "via density modified for 1 board". Now you can realign to the QFP source:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Align Mesh → Heat Source  
→ 4 Node Conn → click S4-QFP heat source node → All In Wind → USE**

The mesh for the board stackup will be regenerated. As before, isolate the front layer, activate via display and show diepad outlines:

**<F12 Root Menu> → Visibility → Isolate → Layer → Lamin>1**

**<F6 Setup> → Display → Node → Via Density → Vias/In2**

**<F6 Setup> → Display → Node → Outlines → Enhan Diepad**

Now you can easily change the via density to "High" for both the QFP and DPAK:

**<F12 Root Menu> → Edit → Plate/Board → Board Props → Vias  
→ Density → High (100) → Node Group  
→ Select Regn → click upper-left corner of QFP red outline  
→ click lower-right corner of QFP red outline  
→ Select Regn → click upper-left corner of DPAK red outline  
→ click lower-right corner of DPAK red outline → USE**

The via density will be changed to "100i" for the selected nodes. Also, Sauna indicates that "via density modified for 8 nodes".

Restore the display setup and visibility:

**<F6 Setup> → Display → Use Default**

**click **

Calculate temperatures:

**<F12 Root Menu> → Analyze → Calc Temps → Steady → Natural → "25"**

You should obtain  $T_{S1-DPAK} = 70.56^{\circ}\text{C}$ ,  $T_{S2-SOIC} = 75.96^{\circ}\text{C}$ ,  $T_{S3-SOIC} = 79.15^{\circ}\text{C}$  and  $T_{S4-QFP} = 96.23^{\circ}\text{C}$ .

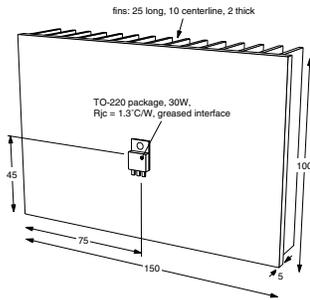
This concludes Quick Tour #2. You should delete the model:

**<F12 Root Menu> → Delete → Everything → click Yes button**

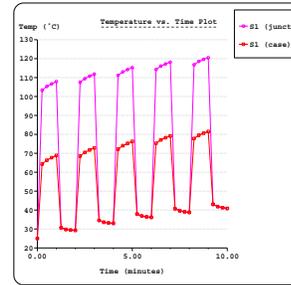
There are three other Sauna Quick Tours, as described on the next two pages.

## Sauna Quick Tour #1: Heat Sinks, Duty Cycles And Stackup Models

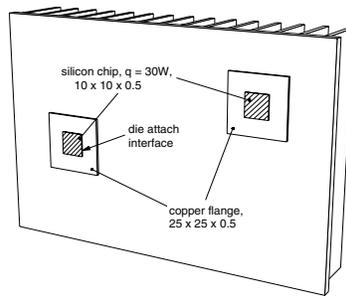
In the first Sauna Quick Tour, you will model these configurations:



*Heat sink with standard device*



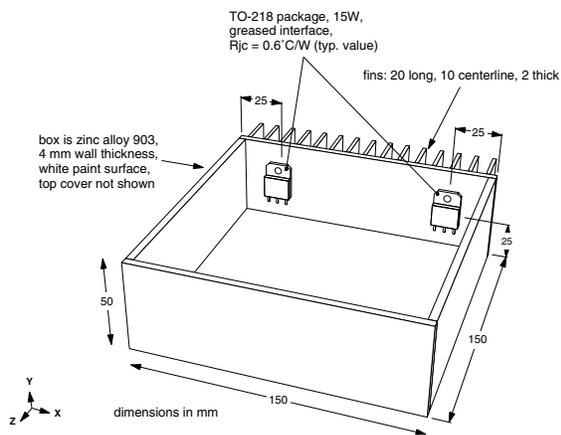
*Duty cycle transient*



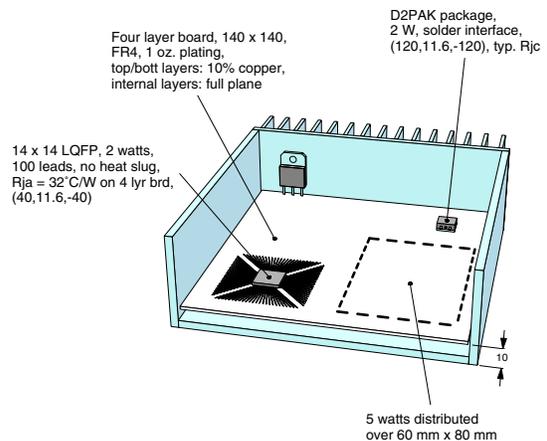
*Heat sink with custom stackups*

## Sauna Quick Tour #3: Basic Finned Box, Box With Board

With Sauna you can do a quick analysis of a box, or you can model in great detail (including internal convection and gray radiation analysis with automatic view factor calculation). These boxes are modeled:



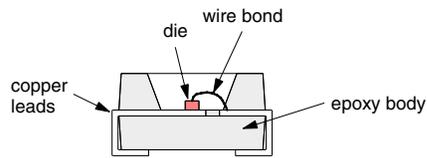
*Basic finned box*



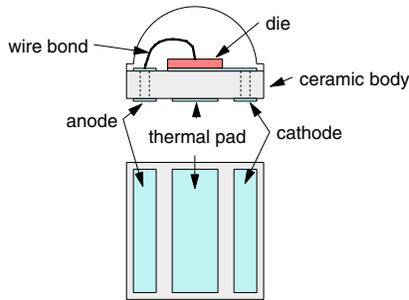
*Box with internal board*

## Sauna Quick Tour #4: Modeling LED Boards

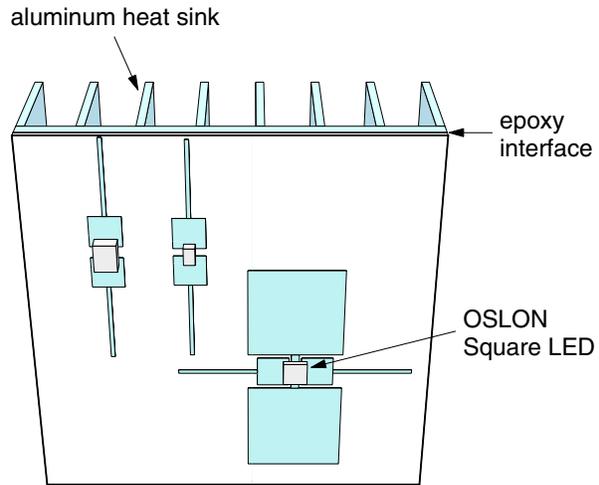
You can model boards with both low power and high power LED's. Sauna's library includes 40 standard LED packages, covering a wide range of automotive, industrial and consumer lighting applications (user-defined packages are also allowed). It's easy to experiment with different pad sizes, add or remove internal copper planes, test different via densities and simulate boards which are bonded to heat sinks.



PLCC package



ceramic package



Multilayer board with high power LED, heat sink

## Wrapping up and disclaimers

You have just seen that Sauna can quickly model board stackups with a variety of component types. You can also quickly handle a variety of what-if modifications: changes in via density, component placement, pad size, trace width, and more. And, even with many advanced features, Sauna is moderately priced and there is outstanding technical support.

Remember that the Sauna Quick Tours only provide an overview of Sauna's capabilities. **Important details have been omitted.** If you want to create accurate Sauna models, you must work through the Introductory Exercises in the Sauna User Manuals. Many of the exercises are available online ([www.thermalsoftware.com/eval\\_exercises.htm](http://www.thermalsoftware.com/eval_exercises.htm)). *You can even work through the exercises with the Sauna Evaluation Package.*

Thank you for your interest in the Sauna thermal modeling package. ◆

6 Jul 17