

Sauna™ Quick Tour #4: Modeling LED Boards

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 website.

Overview of Quick Tour #4

You will be modeling a variety of LED board configurations. You will start with low power PLCC LED's on a single layer board, as shown on the left in Figure 1. For the low power LED's, you will experiment with different trace sizes and you will see the impact of a multilayer board. After that, you will add a high power OSLO Square LED to the board. Since the OSLO Square LED dissipates much more heat, you will try using oversize pads, then a multilayer board, and finally a heat sink configuration. The final configuration is shown on the right of Figure 1.

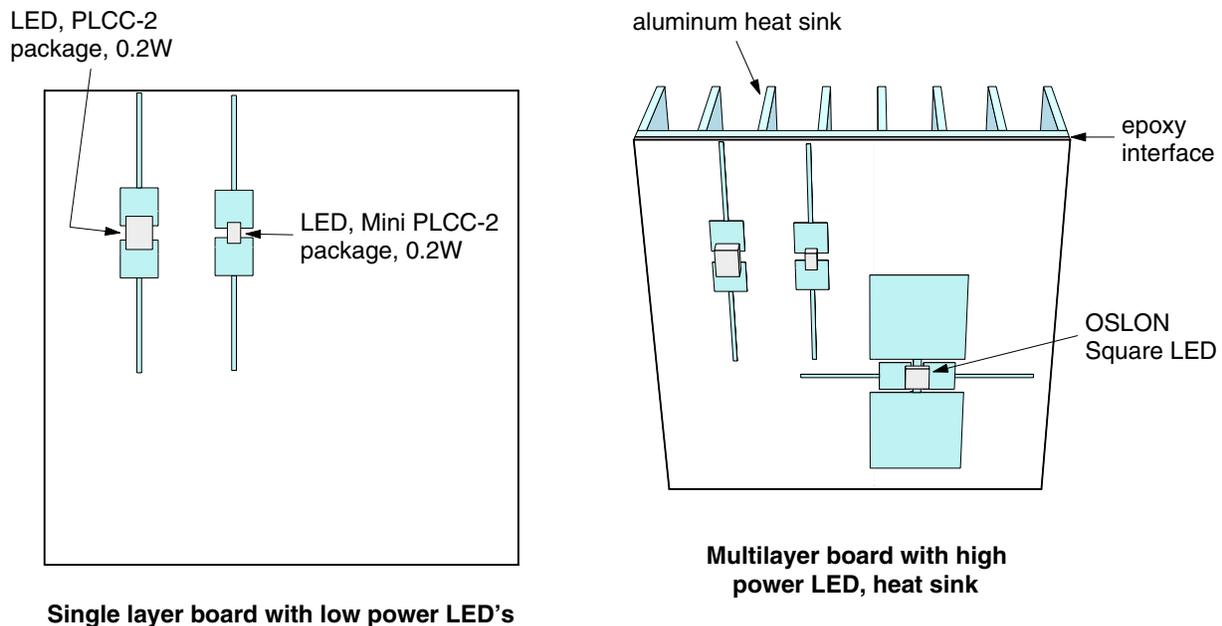


Figure 1: LED board models

Modeling low power PLCC LED devices

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

You will now model the left board in Figure 1. Begin by creating a 50 mm x 50 mm board:

**<F12 Root Menu> → Model → Assembly → Circuit Brd → hit <Enter> to skip → Rectangle
→ Vertical XY → "50,50" (do not type the quotes) → .062"/1.57mm → (0,0,0) → FR4
→ Zer/Lam Only → .025"/0.64mm → One oz. → None**

The board will be created. You will see an array of nodes and resistors inside thick dashed lines. The dashed lines represent the boundary of the board assembly.

Next, add pads for a PLCC-2 package device:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Board Side
→ click on edge of board (dashed line) → Component → One oz.
→ Pad Library → LED's**

You will reach the LED's menu:



Sauna includes roughly 40 standard LED packages, covering a wide range of automotive, industrial and consumer lighting applications. (You can also work with non-standard packages of your own creation.) Continue with:

**PLCC → PLCC-2 → Large-16 mm² → 10 mm/0.4" → 0.50mm/20 mil
→ 0 Degrees → Coords/Trap → "10,35" (do not type the quotes)**

Pads and traces for a PLCC-2 package will be created. Now create pads for the smaller PLCC device:

**Mini PLCC-2 → Large-16 mm² → 10 mm/0.4" → 0.50mm/20 mil → 0 Degrees
→ Coords/Trap → "20,35"**

A second set of pads and traces will be created.

Creating the first heat source

Now you will create the left LED heat source with 200mW of heat dissipation. Figure 2 on the next page shows the internal construction of a typical PLCC-2 package device.

Begin with:

**<F12 Root Menu> → Model → Heat Input → Enhanced Src → LED's → ".2" → "S1" → PLCC
→ PLCC-2 → Typ (0.1 mm)**

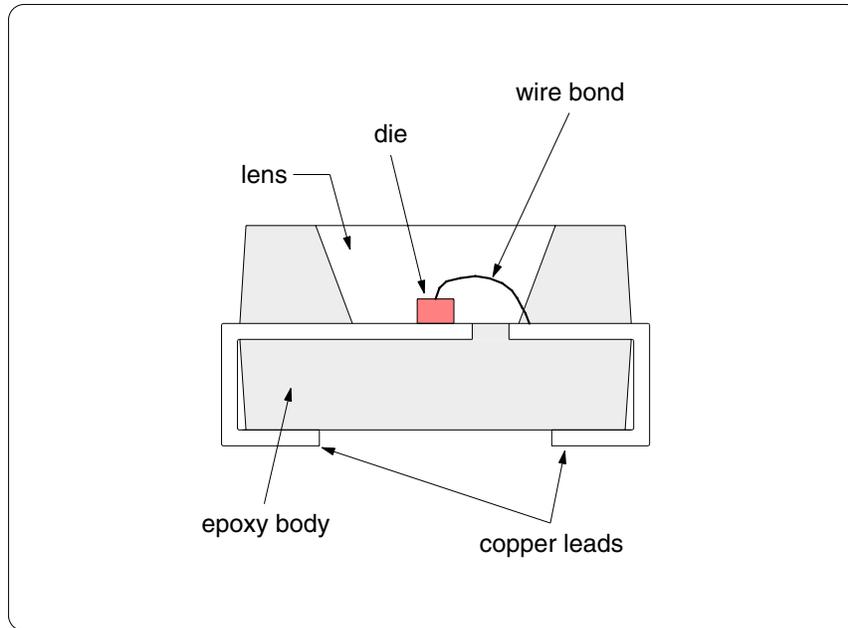


Figure 2: Internal construction of a PLCC LED package

You will reach the R_Lead_Pad (junction-to-lead-pad resistance) menu:

```

R LEAD PAD
>1 Typ/Library
 2 Enter Resis
 3 Enter R para
 4 No Lead Conn

```

The first menu selection is "Typ/Library". With this option you will use $R_{\text{lead-pad}}$ values from Sauna's library which are appropriate for a typical PLCC-2 package device. This is more accurate than the generic "Typ-1500 C/W" which is an option elsewhere within Sauna.

Note that LED's generally have a "thermal lead" (usually the cathode) which has a lower thermal resistance than the non-thermal lead. If you examine Figure 2 above, you will see that the die sits on the left lead, which provides lower thermal resistance, while there is only a wire bond connection to the right lead (more on this later).

Finish creating the heat source:

Typ/Library → Ref Point → click on reference point at center of left pads

The heat source will be created. It's very informative to get a report for the heat source:

<F7 Info> → Trap → Node → Heat Source → click on red heat source node

On the 2nd page of the report, 4th line, you will see that the diepad dimensions are "0 x 0 x 0 (small die assumed)". This means that the diepad, while not actually 0 x 0, is small relative to the size of the body. So heat flow is dominated by conduction between the junction and the leads. All of the PLCC devices use this "small die" assumption, except for the PLCC-4/slug package.

Next, look at the last section of the report "Detailed R-To-Lead-Pad Values (pre-rotation)". You will see that $R_{\text{lead-pad-1}} = 180^{\circ}\text{C}/\text{W}$ and $R_{\text{lead-pad-2}} = 1000^{\circ}\text{C}/\text{W}$. With the typical resistance values, the lower resistance is assigned to lead pad #1, which is the lower pad (before any rotation). These are the typical values provided by Sauna for the PLCC-2 package. You can change these values later if you wish.

Click the Done button to clear the report.

Adjusting the node and resistor mesh

In a moment you will perform a preliminary calculation. But first, the current mesh of nodes and resistors is too coarse for small LED devices. As you will discover with the Sauna User Manual exercises, a 1 mm mesh is appropriate for PLCC LED packages. Remesh with:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Both Axes → Enter Value → "1"
→ All In Wind → USE**

The node mesh will be regenerated at a higher density.

To reduce screen clutter, switch to abbreviated node and resistor symbols:

<F6 Setup> → Display → Symbols → Abbreviated

Adding float resistors and a fixed node

At this point, you have defined the circuit board traces 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 ΔT and convection conditions.

Create the fixed node and float resistors:

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

The fixed node and float resistors will be created.

To better see the new nodes and resistors, you will switch to a right view. For commonly used commands, there are a series of buttons at the lower-left of the Sauna window. The rightmost button in the second row is for switching to a right orthogonal view. Change the view with:

click

The screen will be as shown in Figure 3 on the next page.

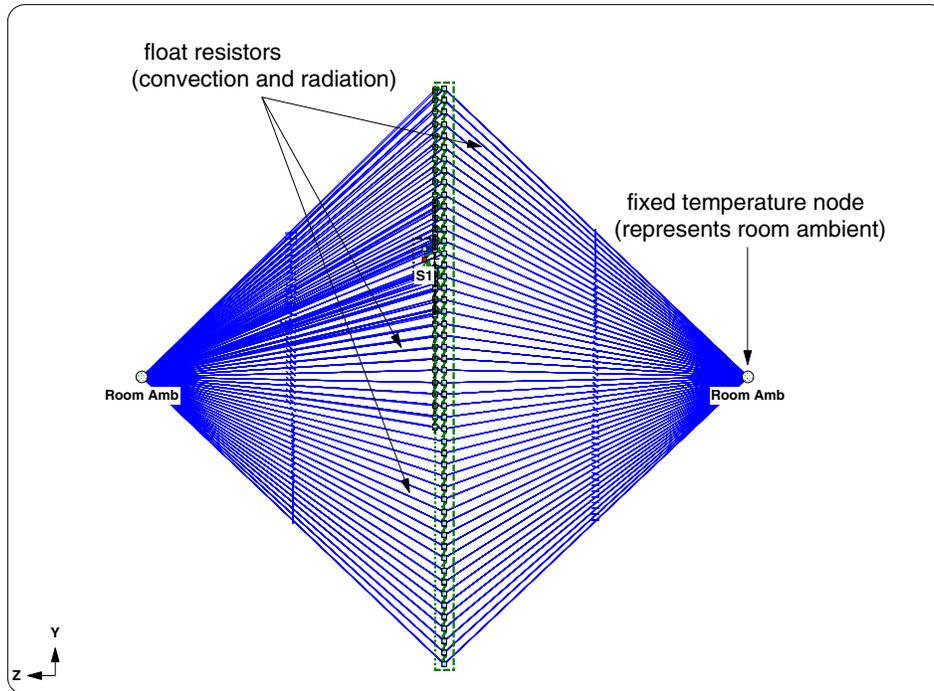


Figure 3: Fixed node and float resistors

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:

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

The calculation only takes a couple of seconds. You should obtain $T_{SI-junct} = 106.57^{\circ}\text{C}$. So with a dissipation of 0.2 watts, $R_{ja} = (106.57 - 25) / 0.2 = 407.85^{\circ}\text{C}/\text{W}$, which is close to the expected typical value of $400^{\circ}\text{C}/\text{W}$ for a PLCC-2 package device with 16 mm^2 pads. Note that the junction temperature would be a few degrees cooler if the device was centered on the board and traces were longer.

Activate temperature contours:

click 

With the contours turned on, you can clearly see the lower pad is hotter. This is what you would expect, since the thermal resistance is lower to this pad.

Turn off contours:

click 

Adding the second LED

Before adding the second LED, you will need to remove the float resistors and ambient nodes. The easiest way to do this is with Undo:

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

Add the second heat source with the same 200mW of dissipated power:

<F12 Root Menu> → Model → Heat Input → Enhanced Src → LED's → ".2" → "S2" → PLCC
→ Mini PLCC-2 → Typ (0.1 mm) → Typ/Library → Ref Point
→ *click on reference point at center of right pads*

The heat source will be added. As before, create float resistors:

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

click 

Calculate temperatures:

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

You should obtain $T_{S1-junct} = 110.16^{\circ}\text{C}$ and $T_{S2-junct} = 128.39^{\circ}\text{C}$. The mini PLCC-2 device is hotter, as expected for a smaller package size.

What-if #1: impact of traces

It will be interesting to see how much the traces help to cool the devices. In a moment, you will delete the four 10 mm traces so that the model appears as shown in Figure 4. But prior to deleting, you need to remove the float resistors and ambients. As before, use Undo:

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

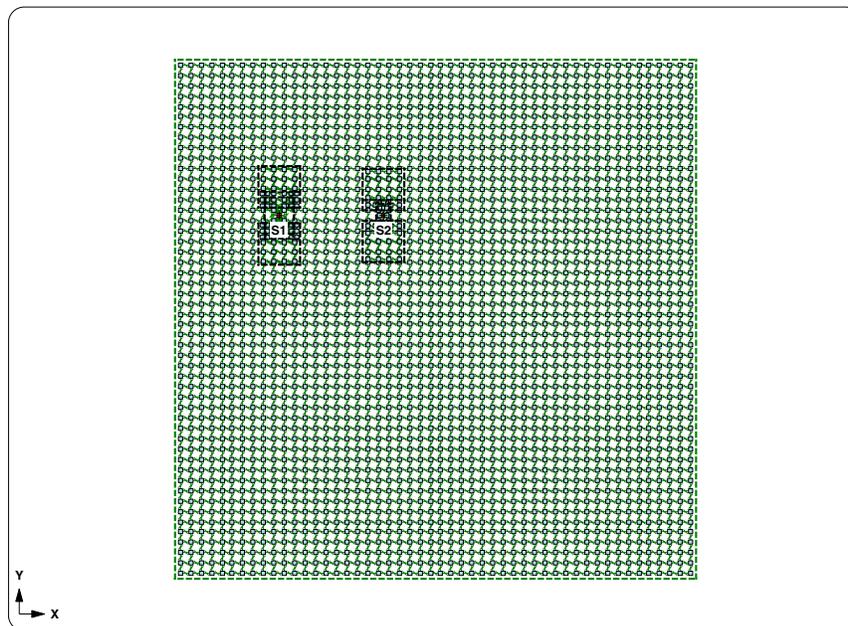


Figure 4: Model after deleting traces

Now you can delete the two traces above the components and two traces below the components.

**<F12 Root Menu> → Delete → Assembly → Plate → Select 1 → *click trace above left device*
→ Select 1 → *click trace above right device* → Select 1 → *click trace below left device*
→ Select 1 → *click trace below right device* → USE**

<F1 Window> → Refresh

After deleting, the model should match Figure 4 on the previous page. Add the float resistors and calculate temperatures:

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

click 

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

You should obtain $T_{S1-junct} = 120.85^{\circ}\text{C}$ and $T_{S2-junct} = 141.87^{\circ}\text{C}$. Temperature have increased by 12.6% and 13.0% respectively. Clearly the traces help to cool the components and need to be included in the thermal model.

Use Undo to return to the previous model configuration:

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

Undo → *click Yes button to undo delete of traces*

Once again you will see traces above and below the components.

What-if #2: adding an internal plane

While traces have an impact, the addition of an internal ground or power plane is even more significant. This is a good thing for most designers since, unless you're designing a very simple product, your board will have internal planes. The internal planes improve the heat spreading ability of the board, which reduces LED temperatures.

Add a single copper plane at the center of the board:

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

The board will be subdivided. Verify that the internal plane was added:

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

The report should indicate that an internal plane ("Copp>2") of one oz. copper was added. Clear the report, then add float resistors:

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

click 

Calculate temperatures:

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

You should obtain $T_{S1-junct} = 82.31^{\circ}\text{C}$ and $T_{S2-junct} = 96.00^{\circ}\text{C}$. This is a change of -32.70% and -31.33%, which is quite significant. Internal planes have a major impact on temperatures, even without thermal vias.

Return to the previous model condition:

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

Undo → *click Yes button to undo add copper layer*

Once again the model has a single copper layer.

High power ceramic LED package

Thus far you have been working with low power LED's which will only dissipate a few hundred milliwatts. But other package styles can produce much more light and will thus dissipate significantly more wattage.

A typical ceramic package is shown in Figure 5. As compared with the plastic PLCC package shown in Figure 3 on page 3, the biggest change is the ceramic body. The ceramic body has much better thermal conductivity than the epoxy used for PLCC-style packages. A variety of ceramic materials are used, including alumina (lowest thermal conductivity, least expensive), aluminum nitride and silicon nitride (best thermal conductivity, most expensive). Because of the good thermal conductivity of the ceramic, the $R_{\text{junct-to-thermal-pad}}$ is rather low, generally in the range of 2-20°C/W.

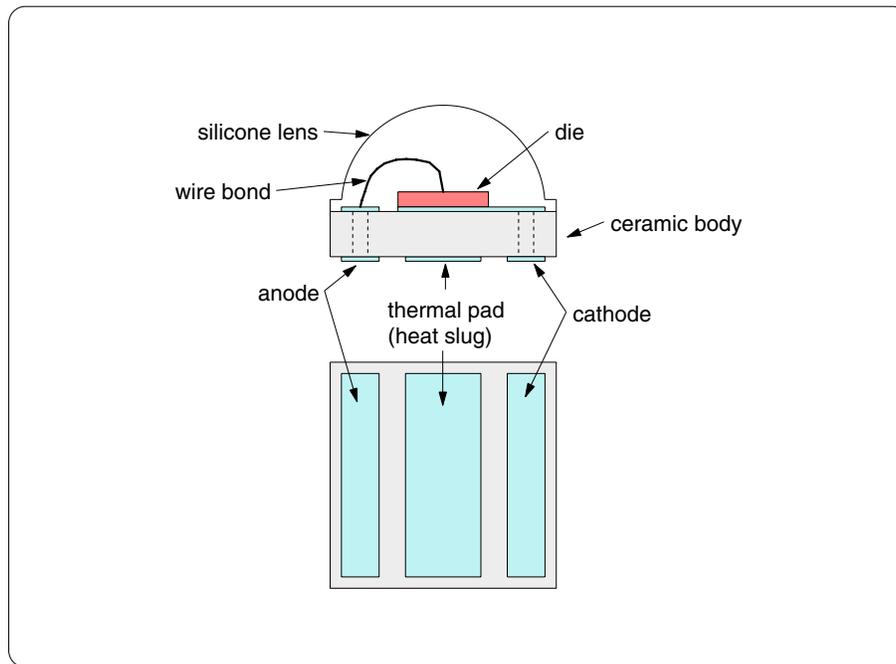


Figure 5: Ceramic LED package

Two comments should be made about the center thermal pad shown in Figure 5. First, Sauna refers to the thermal pad as a "heat slug", borrowing the term used for plastic power packages. LED datasheets, on the other hand, use the term "thermal pad". Second, not all ceramic packages use a large center thermal pad. Instead, the cathode (or possibly anode) is oversized, which provides largely the same benefit as a center pad. Either package style can be handled easily with Sauna.

Adding an OSOLON Square LED to the board

In this section, as shown in Figure 6, you will add a high power OSOLON Square LED device to the board. Notice that the wattage dissipated is 2W, which is 10 times greater than the 200 mW used for the PLCC devices. You will start with the minimal pads shown in Figure 6.

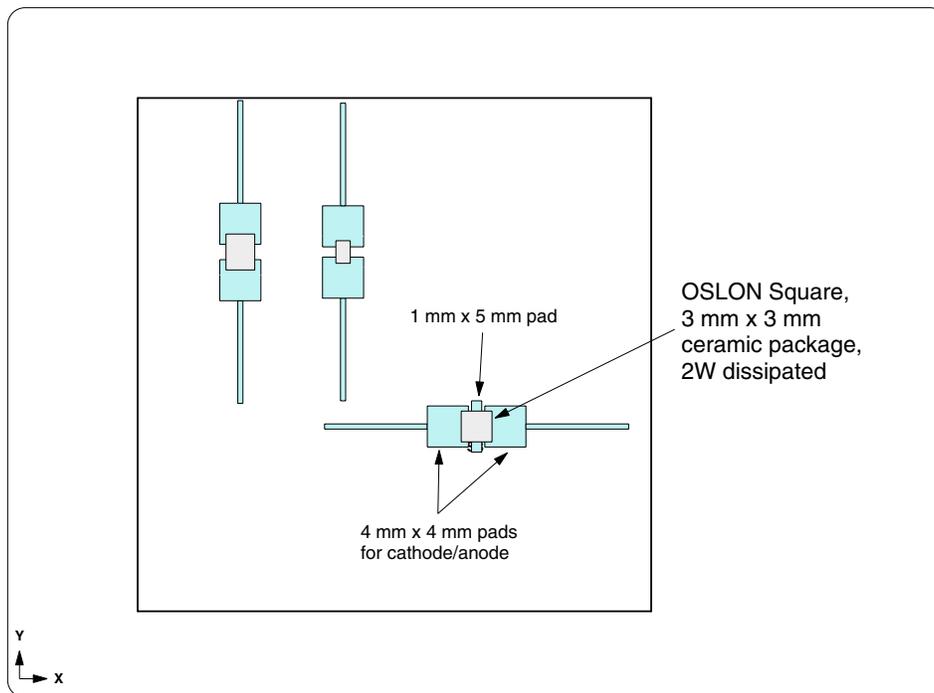


Figure 6: Board with OSOLON Square LED

Begin creating the pads for the OSOLON Square LED:

<F12 Root Menu> → Model → Assembly → Trace/Pad → Trap Trc/Pad
 → *click edge of any trace or pad* → Pad Library → LED's
 → Osram → OSOLON Square → Large-16 mm² → 10 mm/0.4" → 0.50mm/20mil

You will reach the Slug Pad menu:

<p>SLUG PAD >1 Default 2 Specify 3 Not Used</p>
--

As mentioned above, Sauna use "slug pad" to refer to the center thermal pad shown in Figure 5. So with this menu you are specifying the size of the copper pad underneath the center of the device. Finish with:

Specify → "5,1" → +90 → Coords/Trap → "33,18"

The pads will be created. To clearly see the pads, switch into shade mode.

click 

Notice that there is a reference point at the center of the pads, which you will click when the heat source is created. The reference point is a little difficult to see when using wireframe display mode. Also note that the pads for the OSOLON LED are about the same size as the pads for the PLCC devices. So it won't be surprising if the thermal simulation shows that the device is extremely hot.

Now add the OSOLON Square heat source:

**<F12 Root Menu> → Model → Heat Input → Enhanced Src → LED's → "2" → "S3" → Osram
→ OSOLON Square → Use Library → Typ (0.1 mm) → Typ/Library
→ Ref Point → *click reference point at center of pads***

The heat source will be created. It's very informative to get an Info report:

<F7 Info> → Trap → Node → Heat Source → *click S3 red source node*

Near the top of the second page, the report indicates that the diepad size is 1.0 wide x 2.8 deep. This is the size of the thermal pad (heat slug) on the bottom of the part, not the size of the actual die. The size of the thermal pad is the most important parameter for thermal analysis. The die itself is generally smaller than the thermal pad and perhaps slightly offset. The die size and offset certainly has an impact on thermal performance, but this is taken into account in the $R_{\text{junct-to-thermal-pad}}$ on the datasheet.

A few lines down, the report shows "Upper layer thickness: 0.00 (flip chip, bottom pad)". With the die at the top of the package, this can be considered a flip chip. But, of course, it's cooled from the bottom, as opposed to a flip chip with a heat sink on top, as is common for microprocessors.

Towards the bottom of the page, the "Junction-to-case R" of $3.60^{\circ}\text{C}/\text{W}$ is shown. This is another term borrowed from power mosfets. This is $R_{\text{junct-to-thermal-pad}}$ (or $R_{\text{junct-to-heat-slug}}$). Terminology can get messy at times when there's a wide variety of package styles.

Finally, at the end of the report, note that the $R_{\text{junct-to-lead-pad}}$ is just $10^{\circ}\text{C}/\text{W}$. With a ceramic body, the thermal resistance to the cathode and anode is quite low and this provides another way to cool the part.

Since the OSOLON Square is the most important heat source, it makes sense to align the mesh to this device. After clearing the report, enter:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Align Mesh → Heat Source
→ 4 Node Conn → *click S3 source node* → All In Wind → USE**

The node and resistor mesh will be regenerated.

Add float resistors and calculate temperatures:

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

click 

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

You should obtain $T_{S3-junct} = 394.11^{\circ}\text{C}$. As expected, the OSOLON LED is excessively hot.

Enlarging the pads

Since the OSOLON Square LED is excessively hot, you will try using large pads connected to the center thermal pad, as shown in Figure 7:

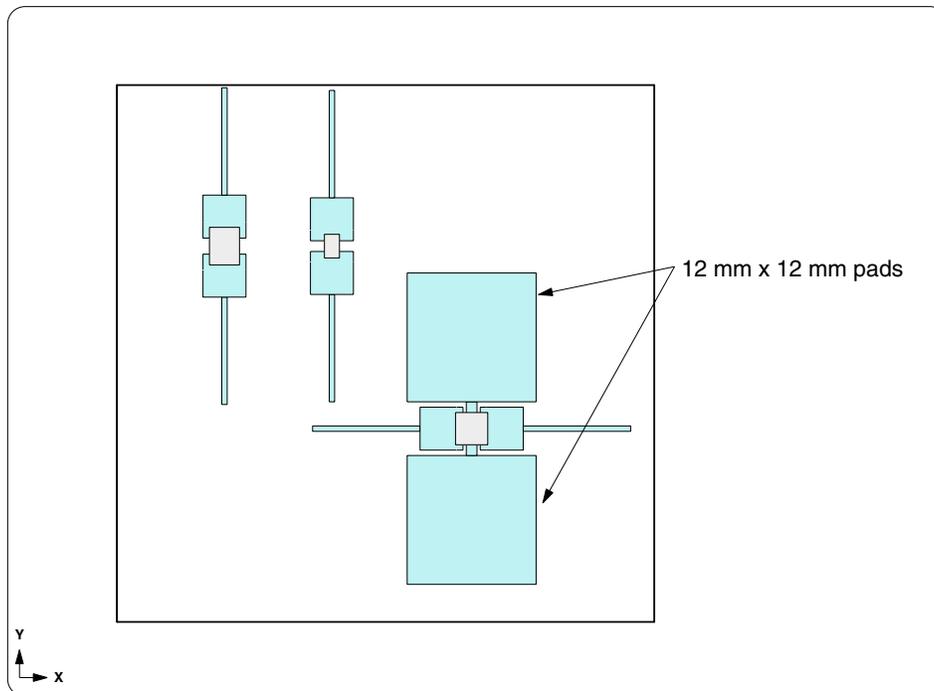


Figure 7: OSOLON Square with larger pads

It's fairly simple to add the 12 x 12 pads. Begin by using Undo to remove the float resistors:

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

Before creating the 12 x 12 pads, you will zoom in to match Figure 8 on the next page. Zooming in is easy, you just use **<F3 Zoom In>** and digitize two times. 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, zoom in:

<F3 Zoom In> → *digitize 1st point to obtain Figure 8* → *digitize 2nd point to obtain Figure 8*

When complete, the window should roughly match Figure 8. If it's not the desired window, just zoom back out with  and try again.

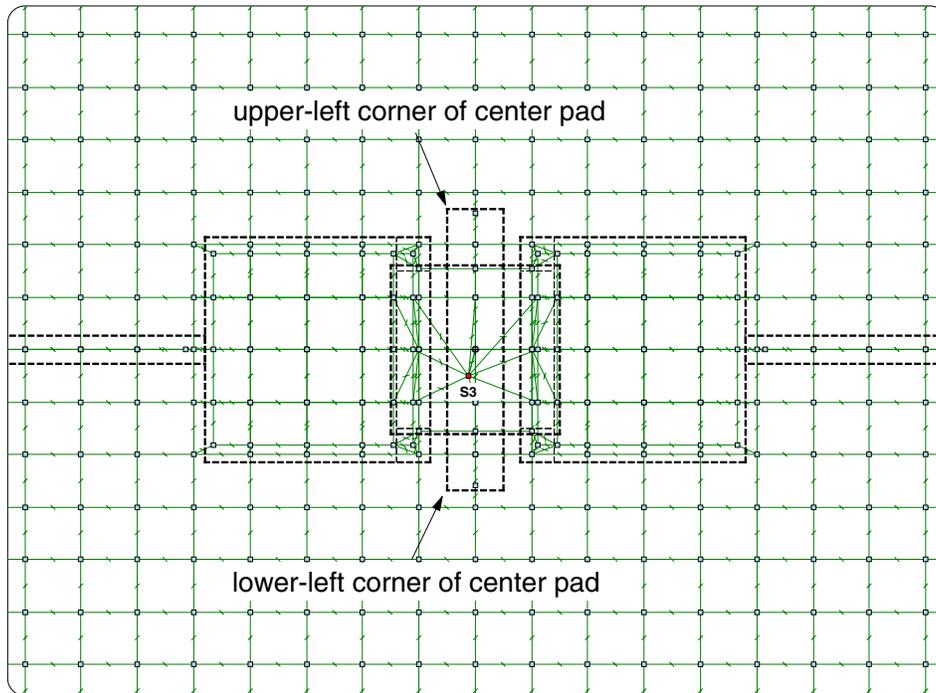


Figure 8: Reference points for adding 12 x 12 pads

Now you can create the upper pad:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Trap Trc/Pad
 → click edge of any trace or pad → Pad/Rect → Ref/Dx
 → click upper-left corner of center pad (see Figure 8)
 → "-5.5" → Dx-Dy-Dz → "12,12"**

The upper 12 x 12 pad will be created. Now create the lower pad:

**Pad/Rect → Ref/Dx → click lower-left corner of center pad (see Figure 8)
 → "-5.5" → Dx-Dy-Dz → "12,-12"**

The lower 12 x 12 pad will be created. Zoom out, then create float resistors:

click 

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

click 

Now you can calculate temperatures:

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

You should obtain $T_{S3-junct} = 234.53^{\circ}\text{C}$. With the larger pads, temperatures have decreased by roughly 160°C . A nice improvement, but the device is still too hot. Activate contours:

click 

The contours show a sharp temperature gradient along the center thermal pad. This suggests that there would be a benefit from making the center pad wider and reducing the gap between the 12 x 12 pad and cathode/anode pads. But you won't try that in this exercise. Instead you will add an internal copper plane in the next section.

Adding an internal copper plane

As before, use Undo to remove the float resistors:

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

Now add the internal plane:

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

The internal layer will be created. Add float resistors and calculate temperatures:

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

click 

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

You should obtain $T_{S3-junct} = 149.46^{\circ}\text{C}$, a significant reduction.

Adding thermal vias

The OSLO LED temperature was reduced significantly by adding an internal copper plane, even without any thermal vias. But it's safe to say that there will be further improvement when the thermal vias are included.

When adding vias, there is no need to remove the float resistors, since the outer surface of the board is not changed in any way. But it will be helpful to zoom in.

Zoom in to match Figure 9 on the next page:

<F3 Zoom In> → *digitize 1st point to obtain Figure 9* → *digitize 2nd point to obtain Figure 9*

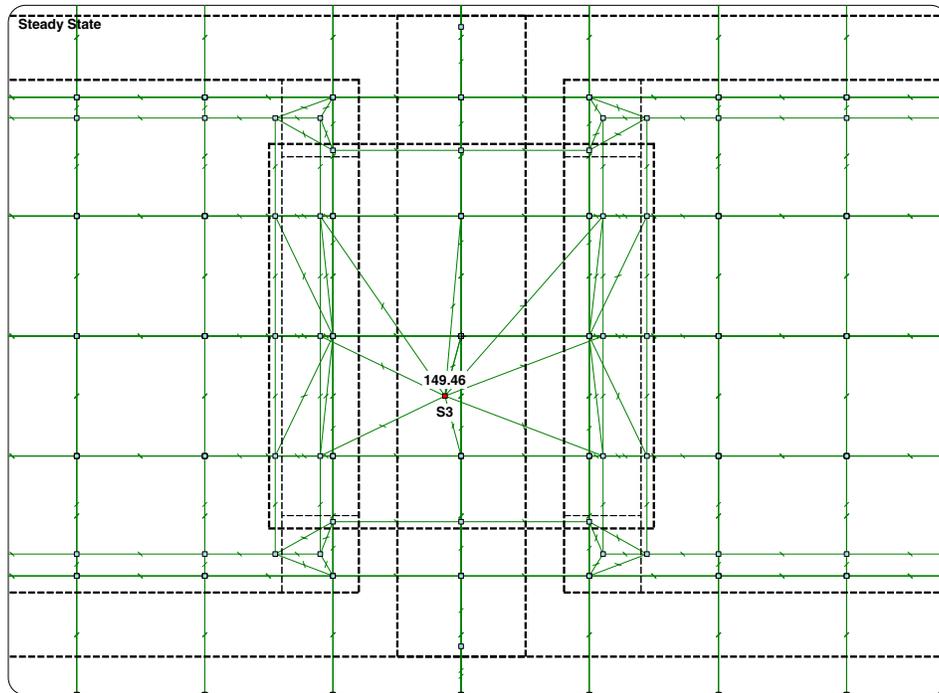


Figure 9: Zoom in to match this window

You will be modifying the via density for the front laminate layer. Of course, it's very important that you modify the front laminate layer and not some other layer. So you will be isolating the frontmost laminate layer.

Isolate the front laminate layer now:

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

With this visibility change, you will see only the front laminate board nodes and the OSLO Square heat source. All copper pads and traces are turned off.

It's important to note that Sauna links enhanced heat sources to the board assemblies, not to the traces and pads. (If you delete the front laminate layer, the LED heat sources will also be deleted.)

Next, activate the display of via density:

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

You will see "0i", for 0 vias per inch², displayed for all board nodes. The board does not have any thermal vias so the via density is 0.0.

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 indicates the limits of the diepad. (For a ceramic LED, this is the outline of the thermal pad on the underside of the package.) You will be modifying the board nodes which are inside the red rectangle.

Switch to "High+" via density for the target nodes:

**<F12 Root Menu> → Edit → Plate/Board → Board Props → Vias → Density → High+ (250)
→ Node Group → Select Regn → *click upper-left corner of red rectangle*
→ *click lower-right corner of red rectangle* → USE**

Sauna will indicate that “via density modified for 3 nodes”. Also, you should see “250i” for the three nodes inside the red rectangle.

Restore the display setup and zoom out:

<F6 Setup> → Display → Use Default

Symbols → Abbreviated

***click*  → *click* **

It's useful to obtain a board stackup report in inch units:

<F7 Info> → Assemblies → Brd Stackup → All → Inch → Screen

The report shows that there are two copper layers and two laminate layers. The front copper layer ("Copp>1") has a total area of 0.65 in² (419 mm²) while the second layer ("Copp>2") is full plane (3.88 in² = 2500 mm²). The front laminate layer has via density varying between 0-250 vias/in², while the second laminate layer does not have vias.

Since the float resistors were never deleted, the modification is complete. Clear the report and calculate temperatures:

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

You should obtain $T_{S3-junct} = 123.65^{\circ}\text{C}$. While this is less than the $T_{junct-max}$ of 135°C for an OSLON Square device, the current simulation is at 25°C . If the design maximum ambient is 50°C , the T_{junct} would be roughly 25°C warmer and would exceed 135°C . For a real design project, you could probably reach this goal by using thicker copper or adding additional vias or adding internal layers. But since this is an exercise, you will reach your goal by adding a heat sink to the back of the board.

Adding a heat sink on the back of the board

You will be creating the heat sink configuration shown in Figure 10 on the next page.

Once again, you need to remove the float resistors. However, you can't use Undo as before because the float resistors were created prior to modifying via density (you need to keep the vias under the device). But it is easy to manually delete the fixed nodes and float resistors.

First, turn on fixed nodes:

***click* **

Now delete the fixed nodes, which automatically causes the float resistors to be deleted:

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

The fixed nodes and float resistors will be deleted.

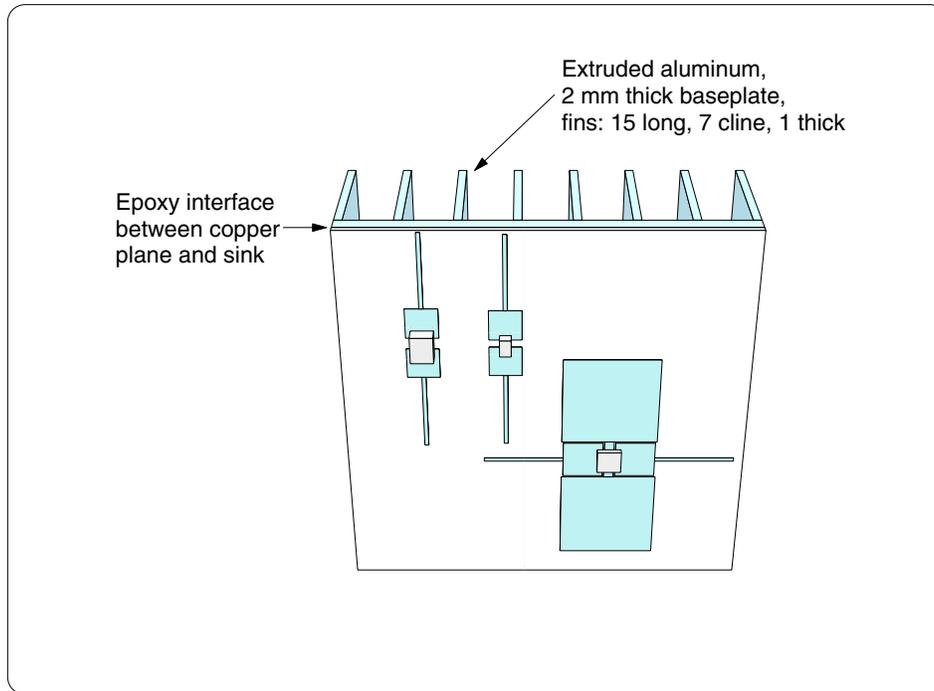


Figure 10: Board with heat sink

As shown in the figure, the heat sink will be bonded to the internal copper plane that you created a few moments ago. So you need to delete the back laminate layer.

Earlier in the exercise, you selected a trace with **Select 1** → *click trace*. This is a useful approach, but Sauna provides an alternative method for situations where a click could be misinterpreted. Every trace, pad, board or other assembly has a unique label and you can use these labels for selection. At the moment, you need to delete the back laminate layer. The label of the back laminate assembly is "Lamin>2". So when it's time to select, rather than clicking the back laminate assembly, you can just type "Lamin>2" (don't actually type the quotes) and hit <Enter>.

With this in mind, switch to a right view and delete the back laminate assembly:

click Right

<F12 Root Menu> → Delete → Assembly → Board → Select 1
→ "lamin>2" (*do not type the quotes*) → USE

The back laminate assembly will be deleted.

Next, to create the backplane of the heat sink, you will take advantage of Sauna's very useful "Surface Assembly" command. The label of the copper plane assembly is "copp>2>1" and you will use this label when creating the surface assembly.

Create the surface assembly:

<F12 Root Menu> → Model → Assembly → Surface → Neg Z (Back) → Plate → Enter Thick → "2"
→ Enter Label → "sink" → Auto → Specify → Aluminum → Extruded → Anodized
→ Select 1 → "copp>2>1" → USE

The surface assembly will be created.

Activate shade mode to clearly see the stackup of assemblies:

click 

Get a report for the newly created plate:

<F7 Info> → Trap → Plate → "sink"

On the 2nd page of the report, note that the stack join interface is "Zero resistance". This is the default for a surface assembly but it's not appropriate for this model.

Clear the report, then change the interface type:

<F12 Root Menu> → Edit → Join Intrfc → Stack → Ordinary

You will reach the Stack Interf menu:

```
STACK INTERF
>1 Zero Resis
2 Die Attach
3 Flat/Dry
4 Flat/Grease
5 Solder 0.1mm
6 Air Gap
7 Unit Area R
8 Total Resis
```

Obtain a description of the interface types by clicking the Menu Ref button (just below the menu):

click 

A description of the different interface types will be displayed. You will be using "Die Attach". As shown on the Menu Ref page, this interface assuming a conductive epoxy with a thickness of 0.05 mm (.002"). The unit interface resistance is 38.5°C-mm²/W. That could be hard to achieve for a heat sink to board interface. But it's a good starting point.

After clearing the Menu Ref screen, finish editing the interface:

Die Attach → Grp To Model → Select 1 → "sink" → USE

Sauna will indicate that "1 stack join modified".

The next step is to add fins to the aluminum plate:

**<F12 Root Menu> → Edit → Plate/Board → Plate Props → New Fins → "15,7,1"
→ Secondary → Select 1 → "sink" → USE**

The fins will be added. Switch to shade mode and change the view:

click  → *click*  → *click*  → *click*  → *click* 

The model should match Figure 10.

Return to a front view:

click 

Add float resistors and calculate temperatures:

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

click 

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

You should obtain $T_{S3-junct} = 82.48^{\circ}\text{C}$, a very significant reduction. Clearly this is an effective way to cool LED's, but the cost will be significant. It's best to fully investigate changes to the board's copper layers and vias before committing to the heat sink approach.

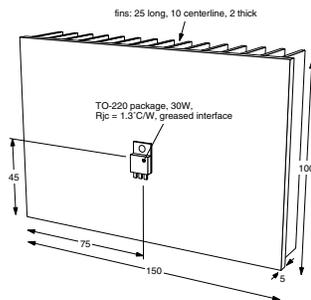
This concludes Quick Tour #4. You should delete the model:

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

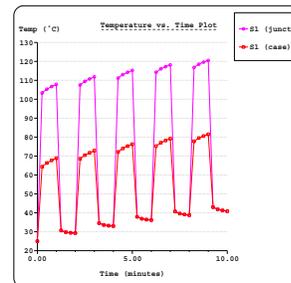
There are three other Sauna Quick Tours, as described below.

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

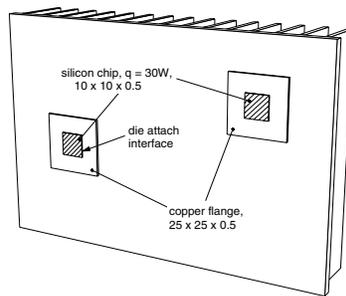
In the first Sauna Quick Tour, you will model these configurations:



Heat sink with standard device



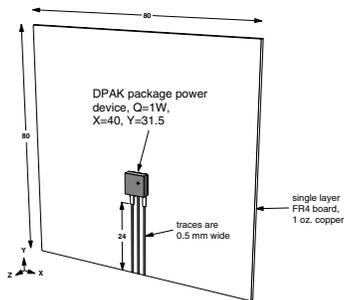
Duty cycle transient



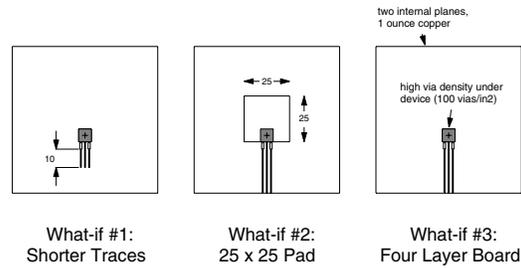
Heat sink with custom stackups

Sauna Quick Tour #2: Circuit Board Modeling

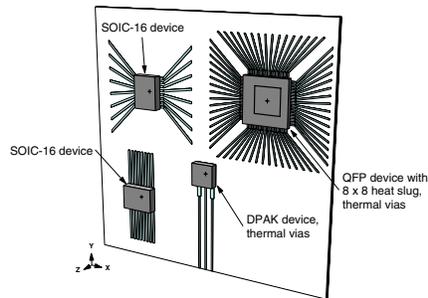
In the second Sauna Quick Tour, you will see how Sauna can be used for both quick models and detailed studies of circuit boards:



Simple DPAK device model



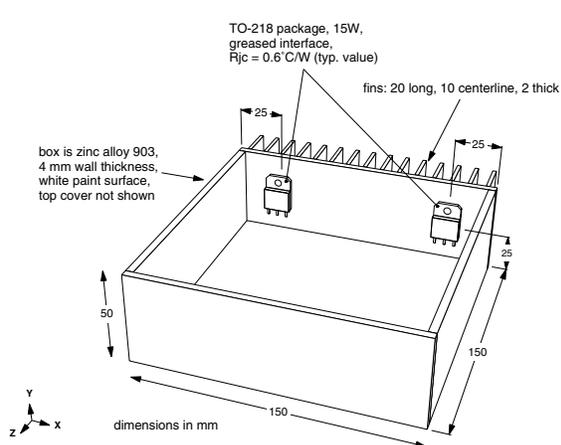
DPAK device what-ifs



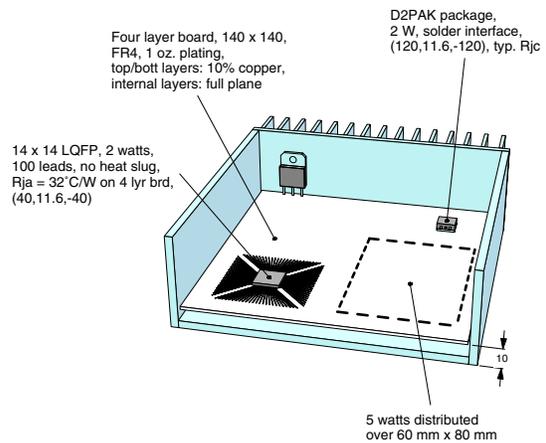
Four layer board with multiple components

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

Wrapping up and disclaimers

You have just seen that Sauna can quickly model a variety of LED board configurations. Even with many advanced features, Sauna is moderately priced and includes expert technical support.

Please 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). *You can even work through the exercises with the Sauna Evaluation Package.*

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

20 Jul 17