

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

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.

Basic finned box

You will start by modeling the box shown in Figure 1. It is assumed that there is no heat dissipation inside the box, so you can focus on the outside walls. Later in the exercise, you will add a circuit board and model internal convection and radiation (see Figure 5 on page 7). *Before beginning the exercise, make sure that the error beep on your computer is turned on (not muted).*

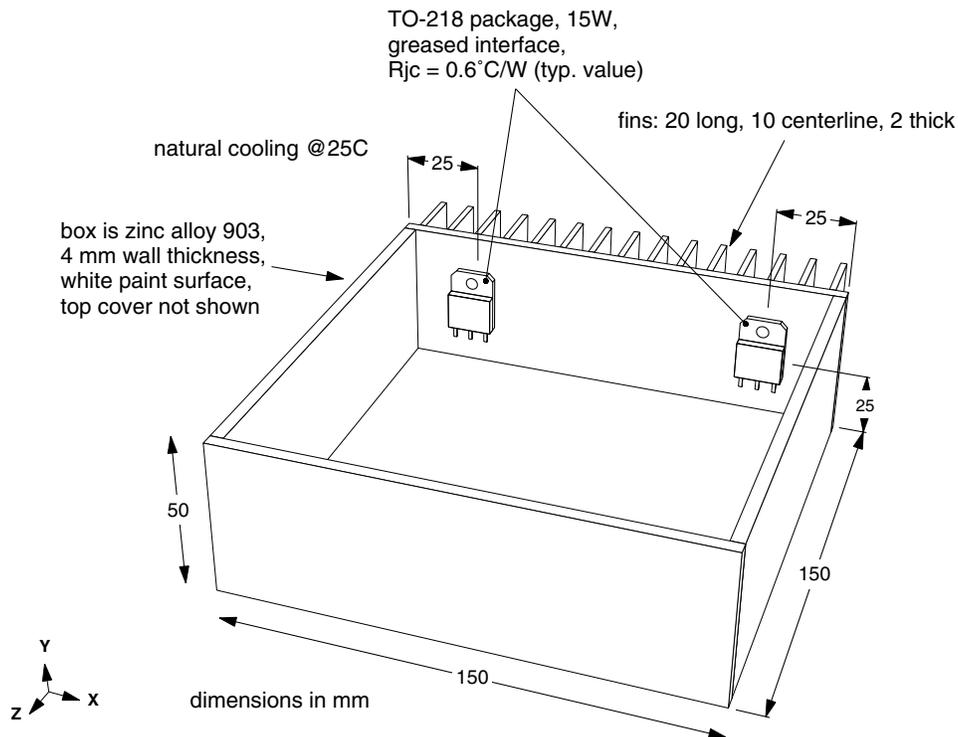


Figure 1: Finned box with power transistors (top cover removed)

Create the box with these commands:

**<F12 Root Menu> → Model → Assembly → Box → Multi-Plate
→ "150,50,150" (do not type the quotes) → (0,0,0) → Uniform
→ "4" (do not type the quotes) → Mainly Horz
→ Other Metal → Zn SAE 903 → Paint/White → Paint/White**

Sauna will create a multi-plate box. The box is composed of 6 plate assemblies and each plate has a mesh of nodes and resistors. Thick dashed lines show the boundary of each assembly.

It is useful to see the model in a perspective view. For commonly used commands, there are seven rows of menu buttons at the lower left of the Sauna window. The right button in the top row is for switching to a perspective view. Switch the view with:

click 

You can also view the model in a shaded view:

click 

Return to wireframe mode by clicking the shade mode button a second time:

click 

Adding fins and heat sources

In a moment you will add heat sources and fins to the back wall. It will be easier if only the back wall is visible. Sauna provides several visibility commands to temporarily turn off parts of the model. As a first step, let's see how Sauna has assigned labels to the six assemblies which make up the box. Obtain an assembly list report:

<F7 Info> → Assemblies → List/Supers → Screen

As shown in the report, Sauna has assigned logical labels ("Back", "Bottom", "Front", etc.) to the assemblies. You can use these labels to turn off assemblies.

Click the Done button to clear the report, then isolate the back wall:

**<F12 Root Menu> → Visibility → Isolate → Assembly → Assy Only → Plate
→ Select 1 → "back" (do not type the quotes) → USE**

Just the back wall will be visible. Resize the window with:

click 

The window will appear as shown in Figure 2 on the next page. As indicated in the figure, Sauna uses different line patterns to draw the plate outline. A short dash is used to indicate the "component side" (front face) of the back wall. A long dashed line shows the "secondary side" (back face).

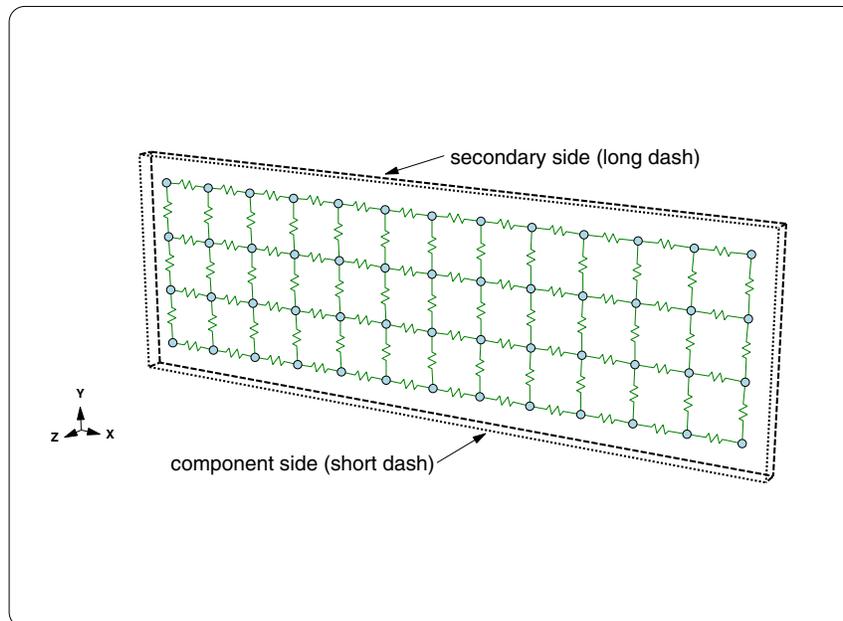


Figure 2: Component and secondary side for back wall

Since the fins are on the back of the box, you will add fins to the secondary side:

**<F12 Root Menu> → Edit → Plate/Board → Plate Props → New Fins
→ "20,10,2" → Secondary → All In Wind → USE**

Additional dashed lines will be added to show the outline of the fins. To clearly see each fin, switch to shade mode:

click 

Return to wireframe mode:

click 

Place the first heat source:

**<F12 Root Menu> → Model → Heat Input → Basic Source → "15" → "Left" → TO-218
→ Typical → Standard → Greased → One → *click edge (dashed line) of plate*
→ Coords/Trap → "25,25"**

The first heat source will be created. Add the second heat source with:

**Basic Source → "15" → "Right" → TO-218 → Typical → Standard → Greased → One
→ *click edge (dashed line) of plate* → Coords/Trap → "125,25"**

The second heat source will be created. If you look at the screen, there are only two case-to-sink resistors for each power transistor. For best accuracy, you should have at least 3 case-to-sink resistors. Remesh the plate to obtain 4 case-to-sink resistors:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Both Axes → Finer 2x
→ All In Wind → USE**

After remeshing, there will be 4 case-to-sink resistors for each heat source.

Adding float resistors and a fixed node

At this point, you have defined the box and heat input. However, the current model is invalid. No heat flow path has been established between the heat sink 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.

Begin by restoring visibility:

click  → click 

When working with the back wall, you saw that the component side faced in and the secondary side faced out. As it turns out, Sauna uses the same rule for all of the assemblies in the box. So you can create ambients for all six plates at the same time:

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

Sauna will indicate that "6 fixed temp nodes and 544 float resistors created". To get a clearer view of the model, switch to a top view:

click 

The model will be as shown in Figure 3:

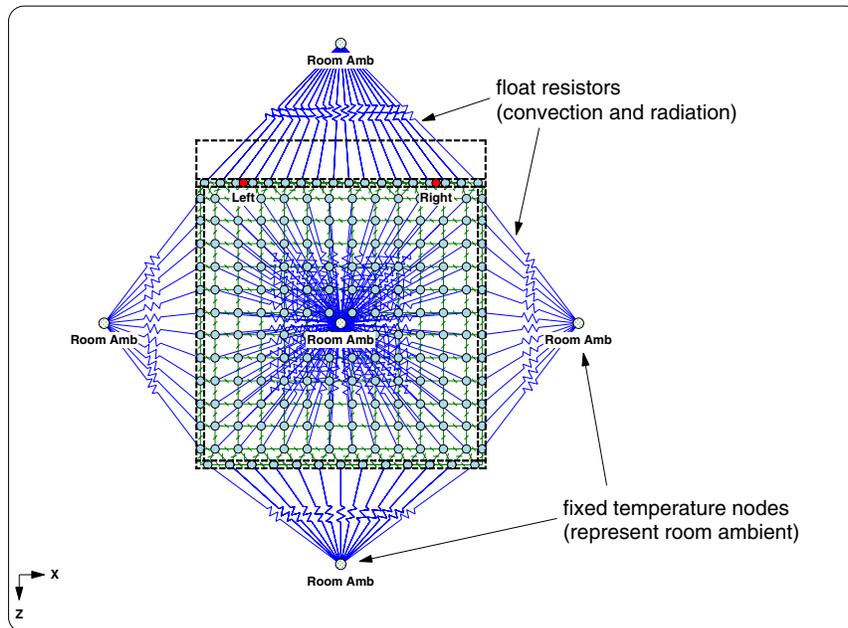


Figure 3: Box model with float resistors and fixed nodes

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

click 

Switch to a perspective view and calculate temperatures:

click 

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

The temperature calculation will only take a few seconds. When complete, you will see two temperatures for each heat source (the left and right heat sources should be identical). The upper temperature, 77.79°C, is the junction temperature. The lower temperature, 68.79°C, is the case temperature ($T_{\text{junct}} = T_{\text{case}} + q \cdot R_{\text{junct-to-case}}$).

It's also interesting to view temperature contours:

click 

What-if #1: horizontal fin channels

In a product development environment, you're frequently interested in evaluating design alternatives. With Sauna, you can easily change wall thickness, material properties, fin dimensions, cooling mode, etc. You will now perform two "what-if" modifications.

When you added fins to the back of the box, Sauna placed the fins in the default orientation, so the fin channels are vertical. Although vertical fin channels are best, it's also interesting to check the performance with horizontal fins.

Change the fin orientation with:

**<F12 Root Menu> → Edit → Plate/Board → Plate Props → Fin Props → Orientation → XY + YZ
→ Horizontal → Secondary → Select 1 → "back" → USE**

Sauna will briefly flash the screen in shade mode to show the new fin orientation. To see the new fin configuration more clearly, switch to shade mode in a right view:

click  → *click* **Right**

Turn off shade mode and return to a perspective view:

click  → *click* 

Calculate temperatures:

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

For both heat sources, you should obtain $T_{\text{junct}} = 82.24^\circ\text{C}$. This is an increase of only 4.5°C. In this situation, horizontal fin channels would probably be acceptable.

It's easy to return to the previous fin configuration. Just use the Undo command:

<F12 Root Menu> → Edit → Undo → *click Yes button*

After using Undo, you should see $T_{\text{junct}} = 77.79^\circ\text{C}$. This is the temperature from the previous calculation.

What-if #2: adding a shelf

Many boxes have internal walls and partitions. As the second what-if you will add a horizontal shelf to the box (refer to Figure 4). Since these types of box constructions are common, Sauna has a "Plate In Box" command to simplify the process.

Add the shelf:

**<F12 Root Menu> → Model → Assembly → Plate In Box → "Shelf" → Calc/No Gap → "10"
→ Horizontal → Minimum → "10" → Other Metal → Zn SAE 903 → Paint/White → Join
→ Zero Resis → All In Wind → USE**

The shelf will be created. To clearly see the shelf, turn off the front wall:

**<F12 Root Menu> → Visibility → Turn Off → Assembly
→ Click Surf → *click anywhere on front wall***

The model will be as shown in Figure 4:

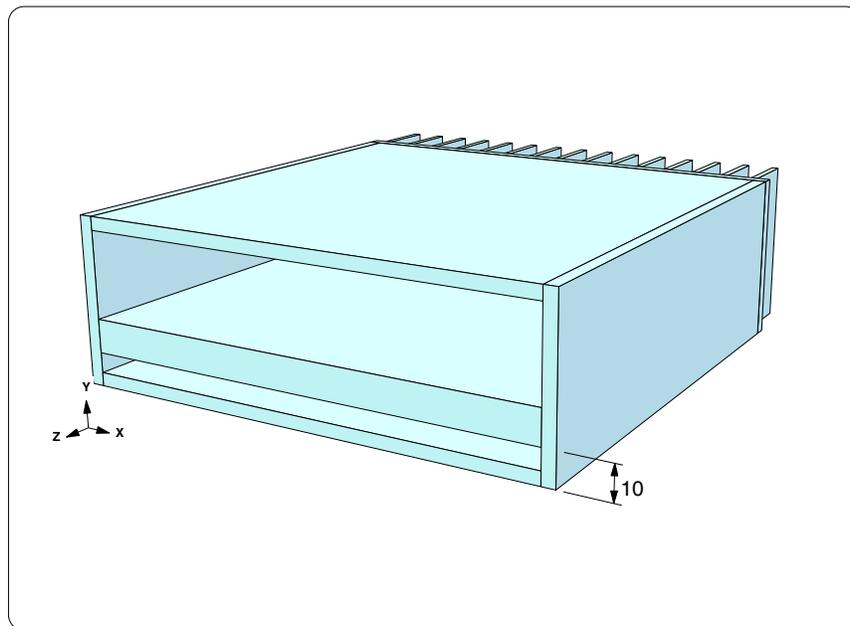


Figure 4: Box with shelf

Turn off shade mode and restore visibility:

click  → **click** 

Calculate temperatures

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

You should obtain $T_{\text{junct}} = 74.06^{\circ}\text{C}$ for both heat sources. This is a reduction of around 4°C . The extra metal in the shelf reduces temperature gradients around the heat sources.

Board in box with internal convection and radiation

Now you will add a board to the box, as shown in Figure 5:

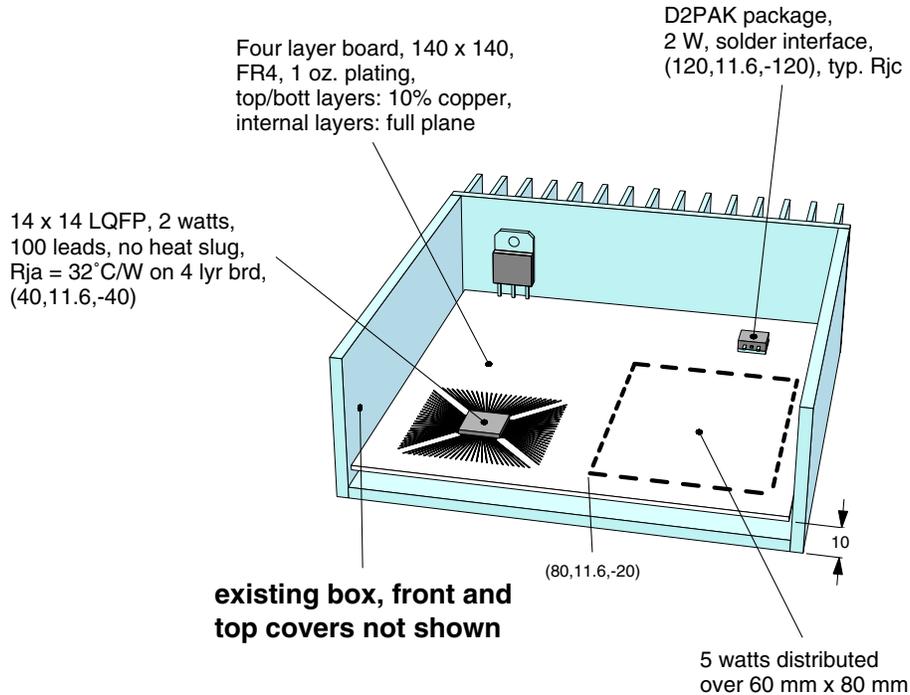


Figure 5: Finned box with circuit board

Begin by deleting the shelf:

<F12 Root Menu> → Delete → Assembly → Plate → Select 1 → *click edge of shelf* → USE

You also need to delete the right heat source:

<F12 Root Menu> → Delete → Node → Heat Source → Any Source
→ Select 1 → *click right red heat source node* → USE

<F1 Window> → Refresh

Planar vs. detailed stackup board models

Sauna provides two different methods for modeling circuit boards: planar and detailed stackup. The planar method is a simplified approach, which has been in use for over a decade. In this type of model, there is no ΔT between the faces of the board. For a multilayer board, you can think of all the layers as being "smashed" together. Also, with the planar approach, you do not model individual traces and pads. Instead, the copper coverage is modified.

As the name implies, the detailed stackup method is more precise. With this method you will create individual traces and pads. You will also modify the via density, so you will be able to

add the vias underneath the DPAK device. If you have already worked through *Sauna Quick Tour #2: Circuit Board Modeling*, you have used the detailed stackup method.

In this exercise you will use both the planar and detailed stackup methods, starting with the planar approach. Both methods are useful, as you will see.

Creating a planar board

You will be using the "Board In Box" command to create the circuit board. The circuit board is 140 mm x 140 mm, which is smaller than the internal dimensions of the box (142 x 142). So, when you reach the Board In Box menu, you will choose an option which allows for a gap between the wall and the board.

Begin with:

<F12 Root Menu> → Model → Assembly → Board In Box → "Main Brd" → Calc/Gap → "1" → Horizontal → .062"/1.6mm → Minimum → "10" → FR4 → Four → One oz.

You will reach the Cu Coverage (copper coverage) menu:

CU COVERAGE	
>1	5%
2	0%
3	10%
4	25%
5	50%
6	75%
7	100%
8	Enter

Since you are creating a planar board, you need to specify the average copper coverage over the four layers of the board. With two full internal planes and 10% coverage on the outer planes, the average copper coverage is $(2 * 1.0) + (2 * 0.1) / 4 = 55\%$. Complete creation of the board with:

Enter → "55" → No Join → All In Wind → USE

The board will be created in the proper position inside the box. Verify that the board has the proper characteristics:

<F7 Info> → Trap → Board → *click edge of board*

You should verify that the board dimensions are 140 mm x 140 mm. In the "-- Copper Layer Properties --" section on the first page, you should see 4 copper layers, 1 oz. plating and 55% copper coverage. Clear the report before continuing.

Using abbreviated symbols

With the newly created board, there are significantly more nodes and resistors on the screen. It is becoming difficult to clearly see the outline of the assemblies. To improve screen clarity, switch to abbreviated symbols:

<F6 Setup> → Display → Symbols → Abbreviated

If desired, you can permanently switch to abbreviated symbols by saving the current display setup (<F6 Setup> → Save Setup).

Adding the D2PAK device

Add the D2PAK heat source to the board:

<F12 Root Menu> → Model → Heat Input → Basic Source → "2" → "S1" → D2PAK → Typical
→ Solder 0.1mm → One → *click edge of board* → Coords/Trap → "120,11.6,-120"

The heat source will be added to the back-right section of the board. Although difficult to see in the current view, there are 4 case-to-sink resistors. So you do not need to remesh the board.

Adding distributed wattage, modeling quad flat pack

Figure 5 (page 7) shows a section of the board with 5 watts of distributed wattage. On a real board, this might correspond to a zone of components with modest dissipation and operating well within the component limits. The only challenge here is to properly select the board nodes.

But there is also the issue of the quad flat pack. Unfortunately, there is no precise way to model this type of quad package with a basic heat source on a planar board. So you will simply apply distributed wattage over an enlarged footprint for the device. When the analysis is complete, you will make a rough estimate of T_{junct} by using R_{ja} and a local internal ambient temperature. In the second half of the exercise, you will perform a more detailed (and accurate) analysis with an enhanced source.

Isolate the circuit board, then switch to a top view:

<F12 Root Menu> → Visibility → Isolate → Check Box → *unselect Visible Plates*
→ *click OK button*

click

Just the board will be visible. As mentioned earlier, there are four case-to-sink resistors for S1.

Before adding the distributed wattage, you will create two reference rectangles. The rectangles are an aid for selecting the board nodes. Rectangles are simply line elements that have no effect on thermal characteristics.

Create a reference rectangle for the 60 mm x 80 mm area of dissipation:

<F12 Root Menu> → Line/Point → Line → New Line → Rectangle → Coords/Trap → "80,11.6,-20"
→ Dx-Dy-Dz → "60,0,-80"

The rectangle will be created. Now you need to define the heat dissipation area for the QFP. The body size is 14 x 14, but the leads will certainly assist in spreading the heat over a wider area. Since the first 10-15 mm of traces are most important, you will distribute the heat over a 36 x 36 area. Add the rectangle with:

Rectangle → Coords/Trap → "22,11.6,-22" → Dx-Dy-Dz → "36,0,-36"

The rectangle for the QFP will be added.

Now that the two reference rectangles have been created, you can easily select the correct nodes. You will be using "Select Region" for the first time. With "Select Region" you digitize twice to define a grouping rectangle. With Sauna, digitize means "click and release" (which you will do two times). It's not hold the mouse button down and drag. Begin by adding distributed wattage for the QFP (see Figure 6):

**<F12 Root Menu> → Model → Heat Input → Distribute → Board Nodes → "2" → Face Area
→ Select Regn → use digitize point #1 in Figure 6 → use digitize point #2 → USE**

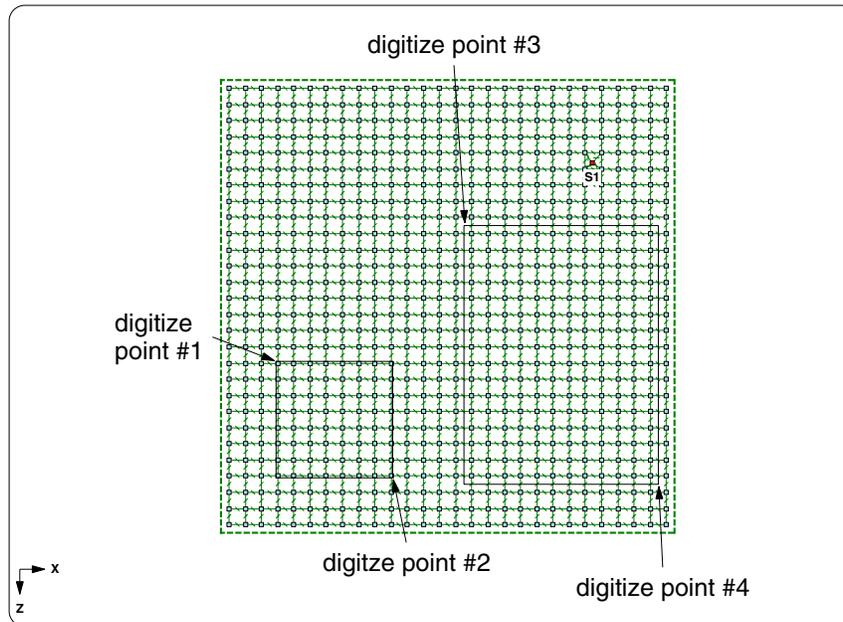


Figure 6: Selecting nodes for distributed wattage

Sauna will indicate that "distributed wattage applied to 64 nodes". If you got a different number of nodes, use **Edit → Undo** and try again.

Now add the distributed wattage for the 60 x 80 rectangle:

**Board Nodes → "5" → Face Area → Select Regn → use digitize point #3
→ use digitize point #4 → USE**

This time you should see the message "distributed wattage applied to 192 nodes". Verify the model wattage:

<F7 Info> → Heat Load → Input → With Supers → Screen

Under the heading "-- Wattage Totals --", the report should indicate that $Q_{total} = 24W$. After verifying the total heat load, clear the report.

The rectangles can now be deleted:

<F12 Root Menu> → Delete → Line/Point → Line → All In Wind → USE

The rectangles will be deleted. Of course, since the rectangles have no impact on thermal characteristics, it is not actually necessary to delete these elements

Before moving on, it should be noted that there is an even quicker way to model the board dissipation. Instead of defining two zones of dissipation, you could simply assign a total of 7 watts to the entire board. For a quick model, this would be fine.

Creating convection and radiation networks

The gap between the board and bottom cover is 6 mm. Many boxes have thin air gaps like this. When the gap is less than 10 mm (0.4"), the air is largely stationary. But a significant amount of heat is transferred through direct conduction and radiation. And, the thinner the gap, the better the heat transfer.

Since thin air gaps are common, Sauna provides "parallel gap resistors". These are easy to use, but you must be careful to select the correct assemblies. As a first step, turn on all assemblies and switch to a front view:

click  → click **Front**

Once again, you will use "Select Region" to define a grouping rectangle. But this time you will select assemblies, not nodes. If the grouping rectangle intersects an assembly edge, the assembly will be selected (added to the group). With Select Region you can select many assemblies simultaneously, without having to click repeatedly on assembly edges. You will be using the grouping rectangle shown in Figure 7. The grouping rectangle does not need to be identical to Figure 7, just reasonably close.

Now create the gap resistors:

**<F12 Root Menu> → Model → Resistor → Float → Para Gap → Cond/Radtn → Convectn OK
→ Horizontal → 100% Area → Entire Assy → Select Regn
→ digitize upper-left corner of grouping rectangle (see Figure 7)
→ digitize lower-right corner of rectangle → USE**

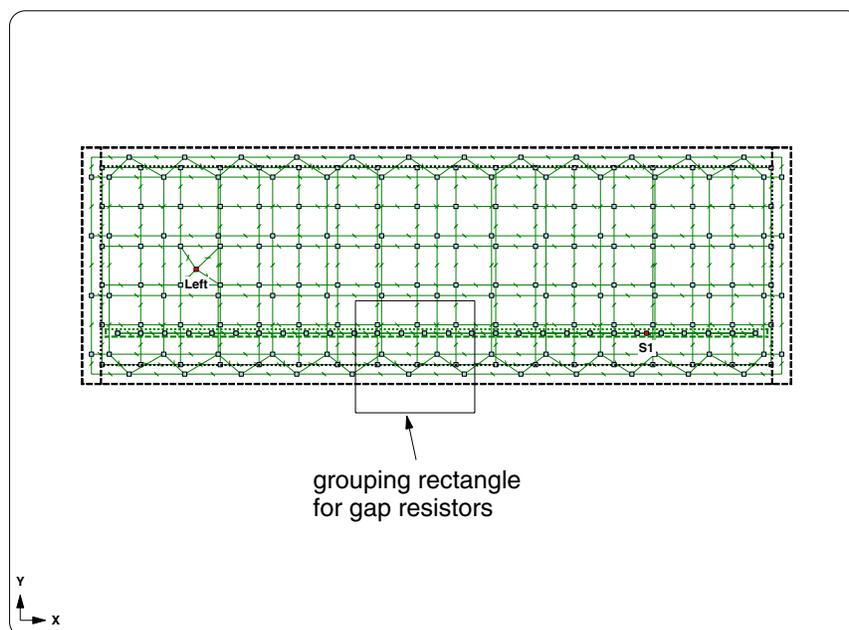


Figure 7: Grouping rectangle for gap resistors

The parallel gap resistors will be created. Sauna will indicate that "1444 gap resistors created; area = 1.96E+4".

Now you will model convection and radiation above the board. The board and top cover are separated by roughly 35 mm. Air will circulate in this region so you can't use gap resistors. You need to create convection and radiation networks. The convection network is comprised of internal ambients and convection resistors. The radiation network is based on a gray radiation analysis, which involve view factors, surface type and ΔT . Sauna calculates the radiation view factors for you automatically. In fact, Sauna calculates all the necessary convection and radiation characteristics automatically. As a user, you only need to define the networks.

When you create the convection and radiation networks, you will use "Select Region" with the grouping rectangle shown in Figure 8. Create the networks:

**<F12 Root Menu> → Model → Amb + Float → Horiz Para → Cnv/Rd Ntwk → 4 Ambients
→ 100% Area → 6 Surfaces → Select Regn
→ digitize upper-left corner of grouping rectangle (see Figure 8)
→ digitize lower-right corner of rectangle → USE**

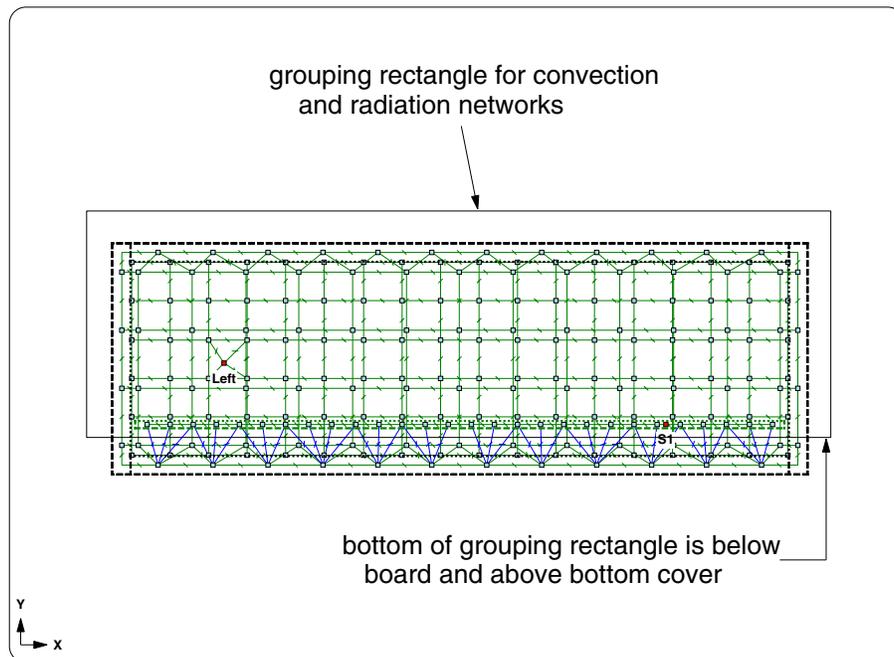


Figure 8: Grouping rectangle for convection and radiation networks

The convection and radiation networks will be created. To better understand the new networks, turn off assemblies in the XY plane:

**<F12 Root Menu> → Visibility → Turn Off → Check Box → click XY Plane (under All Plates)
→ click OK button**

The screen will be as shown in Figure 9 on the next page. Below the board, the gap resistors handle both convection and radiation. Above the board there are separate convection and radiation networks.

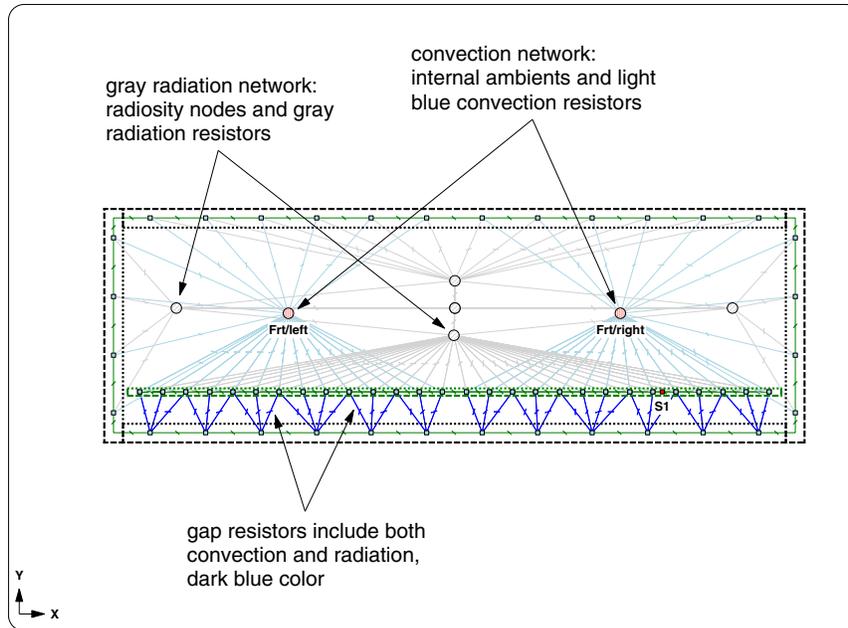


Figure 9: Convection and radiation inside box

Restore visibility and calculate temperatures:

click 

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

You should obtain $T_{\text{Left-junct}} = 74.38^{\circ}\text{C}$ and $T_{\text{S1-junct}} = 93.04^{\circ}\text{C}$.

Estimating the QFP junction temperature, activating contours

Switch to a perspective view

click 

In the perspective view, you can clearly see that $T_{\text{Amb-Frt/left}} = 54.39^{\circ}\text{C}$. You can use the ambient temperature and the R_{ja} (32°C/W) to estimate a junction temperature. Thus, $T_{\text{QFP-junct}} = 54.39 + 2 * 32 = 118.39^{\circ}\text{C}$. This should be considered a preliminary estimate, as discussed in the section "Comparing planar results with stackup results".

It's also interesting to see the contours. Turn on contours:

click 

Turn off these assemblies:

<F12 Root Menu> → Visibility → Turn Off → Assembly → Click Surf → *click anywhere on front wall* → Click Surf → *click anywhere on right wall* → Click Surf → *click anywhere on top cover*

The screen will be as shown in Figure 10 on the next page. This completes the analysis with a planar board. In the remaining sections you will model the board with a detailed stackup.

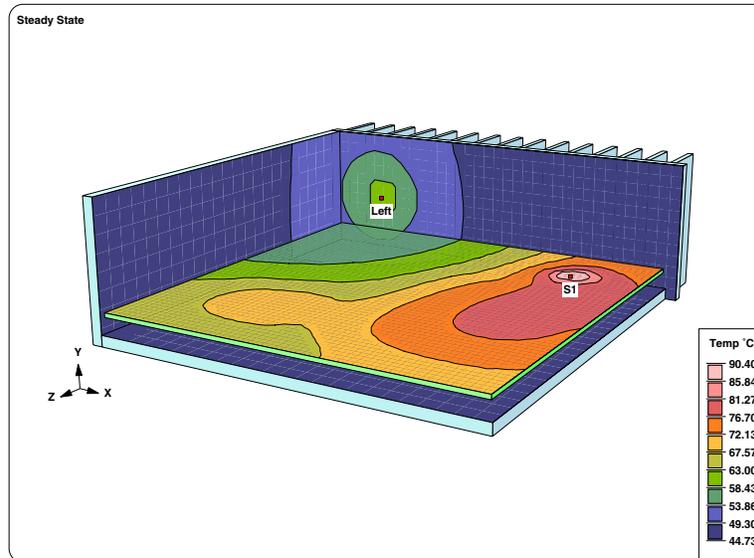


Figure 10: Temperature contours for the planar board model

Preparing to create the detailed stackup

If you worked through *Sauna Quick Tour #2: Circuit Board Modeling*, you are already familiar with the creation of detailed stackups. So it is not essential that you work through the remainder of this Quick Tour.

While it is possible to convert a planar board into a stackup, it's generally simpler to delete the planar board and start over.

Restore assembly visibility, then delete the planar board:

click  → click 

<F12 Root Menu> → Delete → Assembly → Board → All In Wind → USE

Delete the gray radiation network and internal ambients:

<F12 Root Menu> → Delete → Special Del → Gray Netwk → All Netwks

<F12 Root Menu> → Delete → Node → General → Internal Amb → All In Wind → USE

Now you are ready to create the board stackup.

Using in board in box to create the first laminate layer

Start by using the Board In Box command to create the initial laminate layer:

<F12 Root Menu> → Model → Assembly → Board In Box → hit <Enter> to skip → Calc/Gap → "1"
 → Horizontal → .062"/1.5mm → Minimum → "10" → FR4 → Zer/Lam Only → .025"/0.64mm
 → One oz. → None → No Join → All In Wind → USE

The board will be created. Since you did not directly specify a label, the board label will be "Lamin>1". (This is the recommended approach, do not use an arbitrary label in this situation.)

Isolate the board and switch to a top view:

**<F12 Root Menu> → Visibility → Isolate → Check Box → *unselect Visible Plates*
→ *click OK button***

click

Adding traces and pads

Add the trace pattern for the D2PAK:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Board Side → *click edge of board*
→ Component → One oz. → Pad Library → DPAK's → D2PAK → 2 → Same As Body
→ Pad Only → 0 Degrees → Coords/Trap → "120,0,-120"**

The D2PAK pads will be added. Note that you used placement coordinates of "120,0,-120", so the Y-coordinate is 0, which is not actually on the board. In fact, the Y-coordinate is ignored because Sauna will always snap the trace pattern onto the proper side of the board. Now add pads for the QFP, using a 15 mm flare pattern:

**<F2 Backup> → Quad → LQFP → 14 x 14 → 100 Leads → Detail/Flare → 15 mm/0.6" → No
→ Coords/Trap → "40,0,-40"**

The trace pattern will be added to the board.

There is one more pad to create. With the planar board, you assigned wattage directly to the board nodes. But with a board stackup, it makes more sense to assign the wattage to a pad. Create a 60 x 80 pad:

**<F12 Root Menu> → Model → Assembly → Trace/Pad → Trap Trc/Pad → *click a trace or pad*
→ Pad/Rect → Coords/Trap → "80,0,-20" → Dx-Dy-Dz → "60,0,-80"**

Since the 60 x 80 pad represents an area of components and traces, you will reduce the copper coverage, to better match the effective thermal conductivity of this zone. Modify the copper coverage to 25%:

**<F12 Root Menu> → Edit → Plate/Board → Plate Props → Trace/Pad → Coverage → 25%
→ Entire Assy → Select 1 → *click edge of 60 x 80 pad* → USE**

Sauna will inform you that "copper coverage modified for 1 trace/pad".

Adding heat sources

Begin by adding the D2PAK device:

**<F12 Root Menu> → Model → Heat Input → Enhanced Src → DPAK's → "2" → "S1"
→ D2PAK → 2 → Typical → Typ-1500 C/W → Ref Point → *click D2PAK ref point***

The D2PAK heat source will be added. Next, begin creating the QFP source:

Quad → "2" → "S2" → LQFP → 14 x 14 → 100 Leads → Middle → Typical/36% → Typ (0.1 mm)

You will reach the R_lead_pad menu:

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

Up until now, you have always used "Typ-1500 C/W". But as stated in the board exercise, $R_{\text{junct-to-lead-pad}}$ is an important parameter, particularly for components without heat slugs. The recommended way to obtain $R_{\text{junct-to-lead-pad}}$ is to "reverse engineer" the $R_{\text{junct-to-amb}}$, taking into account the vendor's test board. This method, which provides an excellent match with published data, is explained in *Intermediate Exercise 3: More On Circuit Board Modeling* (available on the Sauna evaluation webpage). In the intermediate exercise, an analysis is performed for a 14 x 14, 100 lead QFP with an R_{ja} of 32°C/W, the same as shown in Figure 5. A value of $R_{\text{junct-to-lead-pad}} = 2000^{\circ}\text{C/W}$ is obtained. You will use this value. Continue with:

Enter Resis → "2000" → Ref Point → *click QFP ref point*

The QFP device will be added. Switch to shade mode:

click 

In shade mode, you can clearly see the enhanced sources which, unlike basic sources, includes a 3D body assembly.

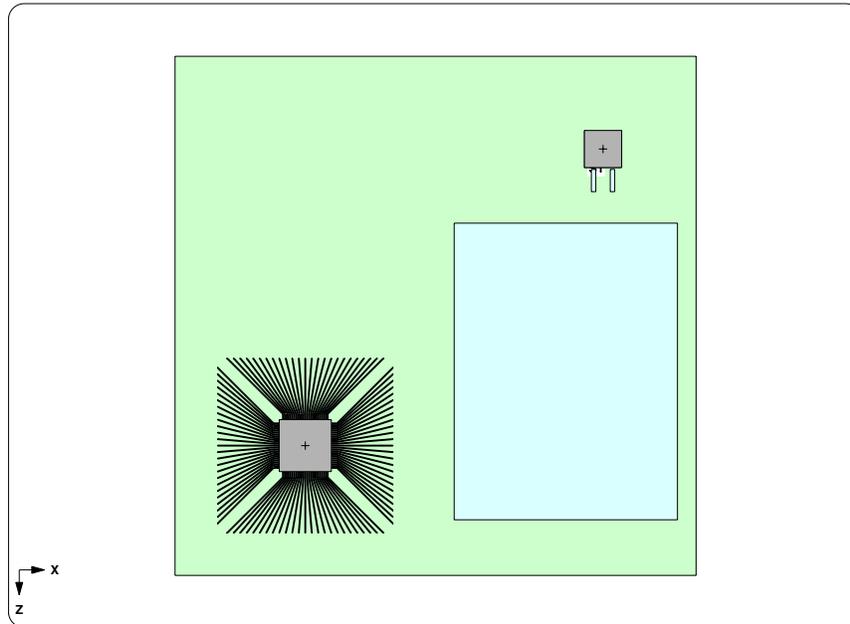


Figure 11: Enhanced sources on board

Finally, you need to add the distributed wattage:

**<F12 Root Menu> → Model → Heat Input → Distribute → Plate → "5"
→ Select 1 → *click edge of 60 x 80 pad* → USE**

Distributed wattage will be applied.

You should verify that the total wattage is correct:

<F7 Info> → Heat Load → Input → With Supers → Screen

With all the traces, the report will extend over several pages. But at the end, you should see 24 watts of total dissipation. Clear the report before continuing.

Since the D2PAK is a heat slug component, you should align the board stackup to this source:

**<F12 Root Menu> → Edit → Plate/Board → Remesh/Align → Align Mesh → Heat Source
→ 4 Node Conn → *click S1 heat source* → All In Wind → USE**

Sauna will inform you that "aligned nodes for 205 assemblies". However, everything will look the same. That's because the D2PAK placement point already matched the mesh. So, in this case, the alignment was not actually necessary.

Adding layers to the stackup

You are ready to create the internal board layers:

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

The internal layers will be created. You should obtain a board stackup report:

<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. Notice that copper coverage for the Copp>1 layer is less than 100%, because of the 25% copper coverage on the large pad. The Copp>1 layer has copper area = 5,385.22 mm², less than full-plane. On the other hand, the internal layers, Copp>2 and Copp>3, have 100% copper coverage and are full-plane (area = 19,600 mm²). Clear the report before continuing.

Modifying via density

All that remains is to add vias under the D2PAK device. The vias will extend from the D2PAK pad on the surface to the first internal copper plane. So you need to modify the first laminate layer, "Lamin>1", and not the other laminate layers. The first step is to isolate "Lamin>1":

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

With this visibility change, you will see only the top laminate assembly and the heat sources. All other laminate layers are turned off, as are all traces and pads. The heat sources are visible because Sauna links heat sources to board assemblies, not traces and pads.

While the correct layer is visible, you still need to take care to modify only the nodes underneath the "S1" D2PAK. Start by activating the display of via density.

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

You will see "0i", for 0 vias per inch², displayed across the board. Currently, this layer has zero vias. To identify the exact nodes to be modified, turn on diepad outlines:

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

Now you will see red outlines for both the S1 and S2 sources (the outline of the S2 source is rather difficult to see without zooming in). Now you can change the via density:

**<F12 Root Menu> → Edit → Plate/Board → Board Props → Vias → Density → High (100)
→ Node Group → Select Regn → *click back-left corner of D2PAK red outline*
→ *click front-right corner of D2PAK red outline* → USE**

Sauna should indicate that 4 nodes were modified. For these 4 nodes, “100i” is now displayed.

Restore the default display setup, turn on model elements and switch to a front view:

<F6 Setup> → Display → Use Default

Symbols → Abbreviated

click  **→ click** 

One last time, check the board stackup:

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

The report should show that the Lamin>1 layer has vias, while the other layers do not. You should clear the report before continuing.

Completing the model and calculating temperatures

You will follow the same procedure as before for adding ambients and float resistors, beginning with the gap resistors:

**<F12 Root Menu> → Model → Resistor → Float → Para Gap → Cond/Radtn → Convectn OK
→ Horizontal → 100% Area → Entire Assy → Select Regn
→ *use grouping rectangle from Figure 12* → USE**

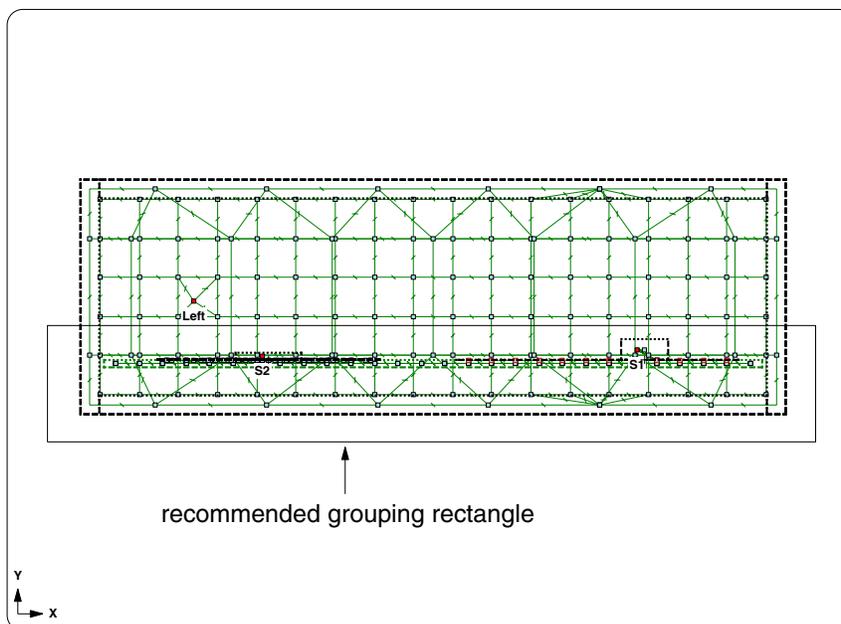


Figure 12: Recommended grouping rectangle for gap resistors

The gap resistors will be created and Sauna will inform you that “1444 gap resistors created; area = 1.96E+4”. Now create convection and radiation networks above the board:

**<F12 Root Menu> → Model → Amb + Float → Horiz Para → Cnv/Rd Ntwk → 4 Ambients
→ 100% Area → 6 Surfaces → Select Regn → *grouping rectangle from Fig. 8 (page 12)* → USE**

Convection and radiation networks will be created. Calculate temperatures:

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

The calculation will take longer, due to the larger model size with additional layers. When complete, you should obtain $T_{\text{Left-junct}} = 74.38^{\circ}\text{C}$, $T_{\text{S1-junct}} = 100.63^{\circ}\text{C}$ and $T_{\text{S2-junct}} = 114.03^{\circ}\text{C}$.

Comparing planar results with stackup results

So now you have modeled the box two different ways. With the planar board approach, it only took a few minutes to define both the board and complete convection and radiation networks. The detailed stackup method took somewhat longer, but the task was still completed in minutes, not hours. But how does the accuracy of the two methods compare?

Table 1 compares the junction temperatures for the detailed stackup and planar models. Some interesting conclusions can be drawn. First, for a wall mounted device, such as the TO-218 device, the details of board modeling are not important. There will be little, if any, impact.

Heat Source	Stackup T_{junct}	Planar T_{junct}	ΔT	Percentage Change
Left (wall mounted TO-218)	74.38	74.38	0.00	0.0%
S1 (D2PAK)	100.63	93.04	-7.59	-10.0%
S2 (QFP)	114.03	118.39	+4.36	+4.9%

Table 1: Detailed stackup and planar results

Second, for a device with a large heat slug and few leads, like the D2PAK, the planar assumption is optimistic because, effectively, the planar model provides infinite vias to all layers. Although the planar model is optimistic, temperatures were still within 10% for the D2PAK device. So, for the "DPAK class" of component, the planar model is adequate.

However, it's not so easy to make a conclusion about the non-heat slug QFP, or any component that relies heavily on cooling through the leads. For the problem just analyzed, the temperatures are fairly close. But you shouldn't count on this. For these types of components, the planar model provides a "first pass" temperature, with an error band of $\pm 20\%$ or so.

Finally, there's this question: what about modeling components with small heat slugs on a planar board? For example, you might be modeling a 10 x 10, 64 lead QFP with a 6 x 6 heat slug. In this case, experience has shown that you can model with a basic source which is slightly larger (approx. 20%) than the actual heat slug.

Of course, in all cases, the most accurate results come from the detailed stackup method. And the graphics will look much better in your Powerpoint presentations!

Other types of box models

The current box model is really quite simple. You can do much more with Sauna. You could have multiple boards, including partial boards. "Box in box" models can be handled without great difficulty. Figure 13 shows a more complex box model. The box can be easily modeled by an experienced Sauna user.

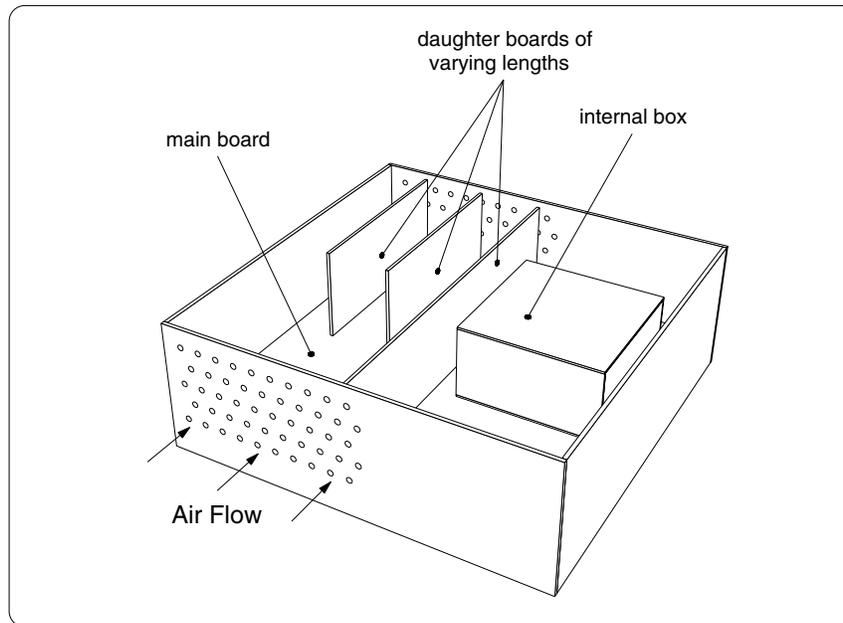


Figure 13: More complex box model, easily modeled by experienced user

Wrapping up and disclaimers

This exercise is complete. You should delete the current model:

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

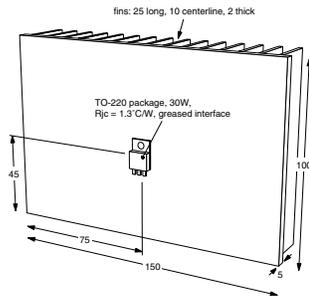
Remember that the Sauna Quick Tours only provide an overview of Sauna's capabilities. **Important details have been omitted.** If you want to create accurate Sauna models, you must work through the Introductory Exercises in the Sauna User Manuals. Many of the exercises are available online (www.thermalsoftware.com/eval_exercises.htm).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

If you have questions, be sure to contact Technical Support (support@thermalsoftware.com, 734-761-1956).

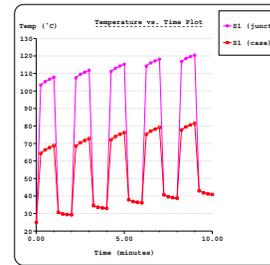
There are three other Sauna Quick Tours, as described on the next two pages.

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

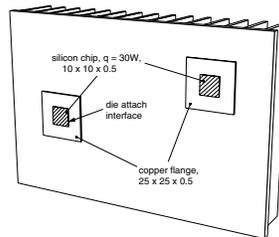
In the first Sauna Quick Tour, you will model these configurations:



Heat sink with standard device



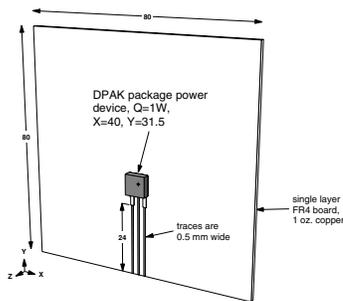
Duty cycle transient



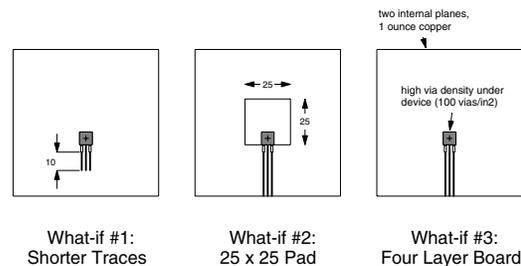
Heat sink with custom stackups

Sauna Quick Tour #2: Circuit Board Modeling

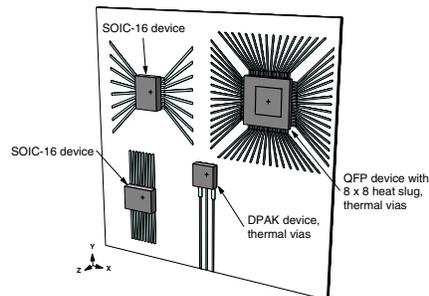
In the second Sauna Quick Tour, you will see how Sauna can be used for both quick models and detailed studies of circuit boards:



Simple DPAK device model



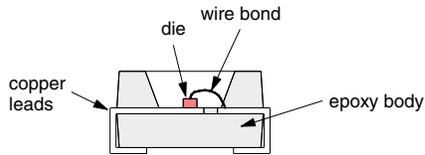
DPAK device what-ifs



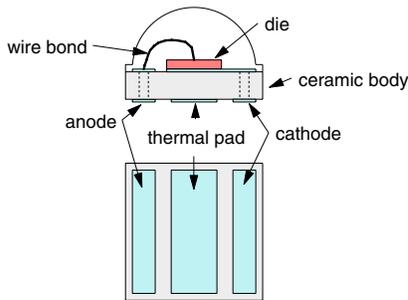
Four layer board with multiple components

Sauna Quick Tour #4: Modeling LED Boards

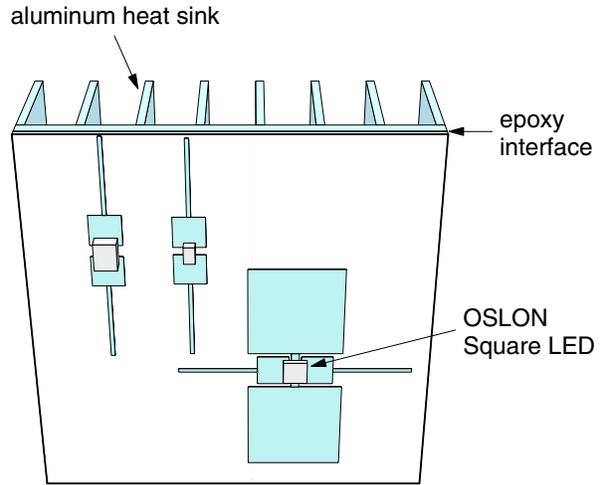
You can model boards with both low power and high power LED's. Sauna's library includes 40 standard LED packages, covering a wide range of automotive, industrial and consumer lighting applications (user-defined packages are also allowed). It's easy to experiment with different pad sizes, add or remove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

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

30 Jun 17