

TUTORIAL

How to Use LTspice's SOAtherm Models in PSIM

June 2018

1. Overview

SOAtherm models in LTspice¹ includes a collection of MOSFET thermal models that predict the maximum MOSFET die temperature over time. The thermal model reports the MOSFET die without influencing the electrical behavior of the MOSFET model.

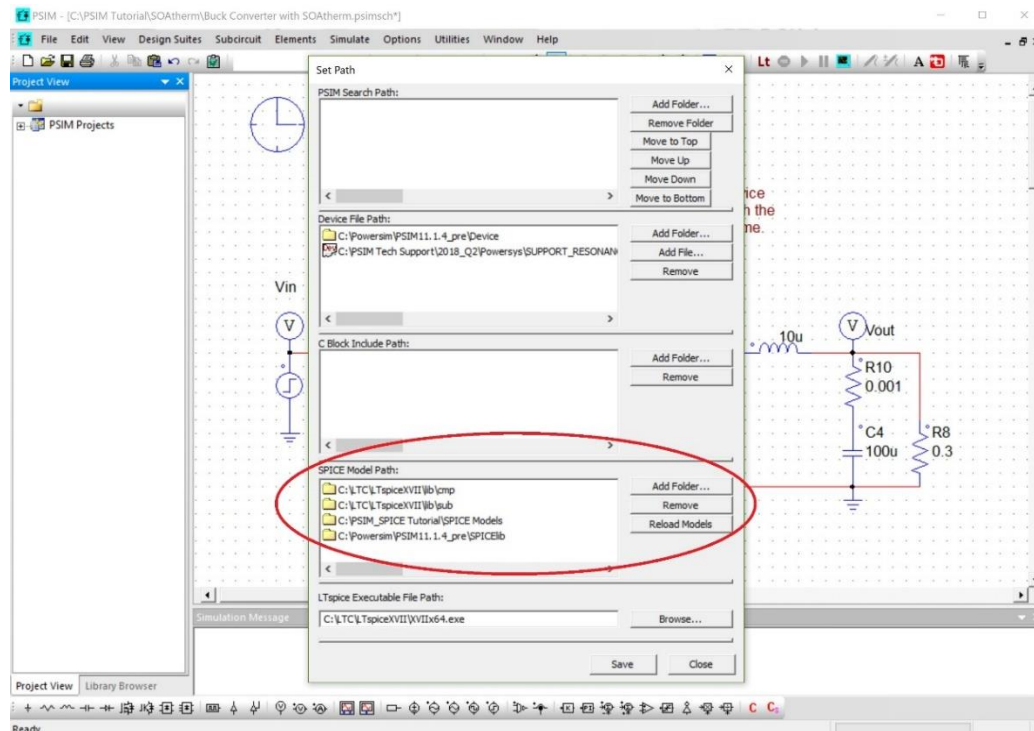
In PSIM schematic, one can use the MOSFET models and the corresponding SOAtherm thermal models to simulate the MOSFET thermal behaviour and safe operating area (SOA).

2. Preparing for Using SOAtherm Models

To include the SOAtherm models into PSIM's SPICE model search path, use the **Set Path** function in the **Option** menu as below.

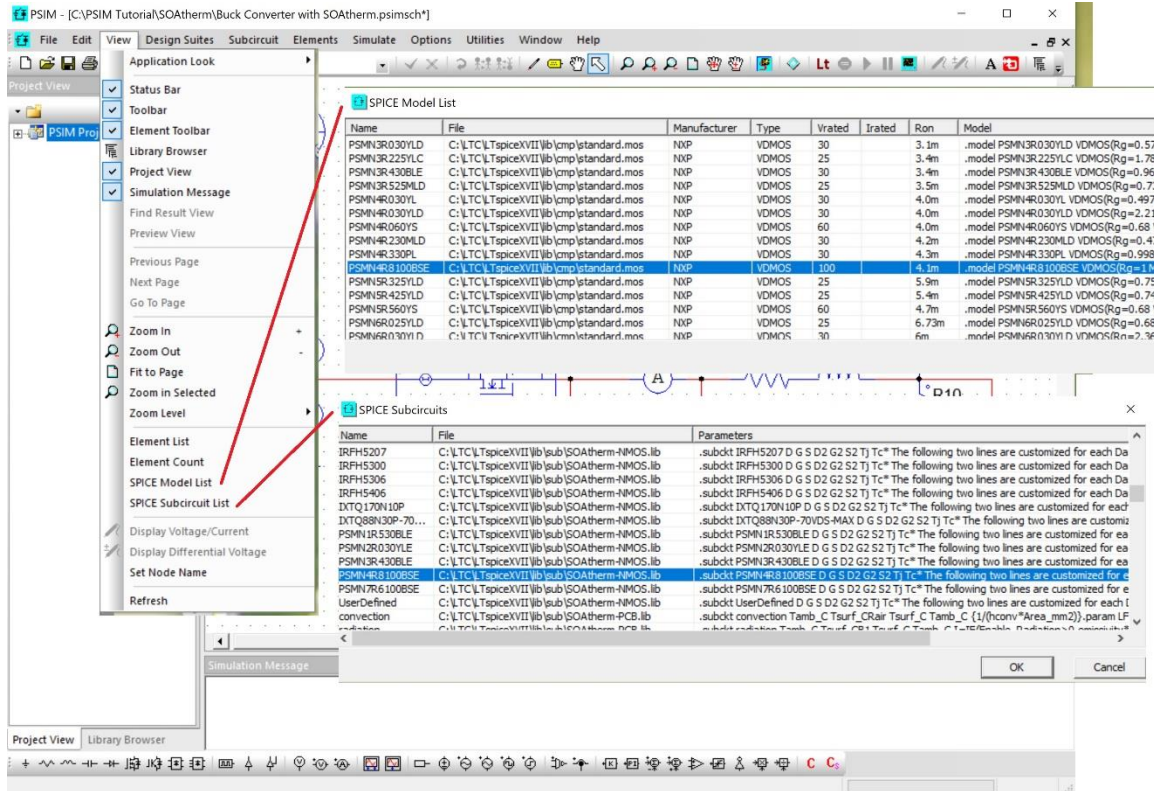
- In PSIM, select **Options >> Set Path**. Click on the button **Add Folder** in the section **SPICE Model Path**, then browse and find the LTspice library path for MOSFET models (for example, "c:\LTspiceXVII\lib\cmp"). Add the folder to the path.
- Do the same to find and add the path for SOAtherm models (for example, "c:\LTspiceXVII\lib\sub").
- In PSIM, select **Options >> Set Path**, and the search path dialog is shown as below.
- Click on **Reload Models** to refresh the search path.
- Click on **Save** and then **Close** to exit the dialog.

The dialog window is shown below after the paths are added.



¹ LTspice is copyright by Linear Technology Co., 1998-2018

To check what models and subcircuits are available for SPICE simulation use, select **View >> SPICE Model List** or **View >> SPICE Subcircuit List**, as shown below.



3. Running PSIM Simulation

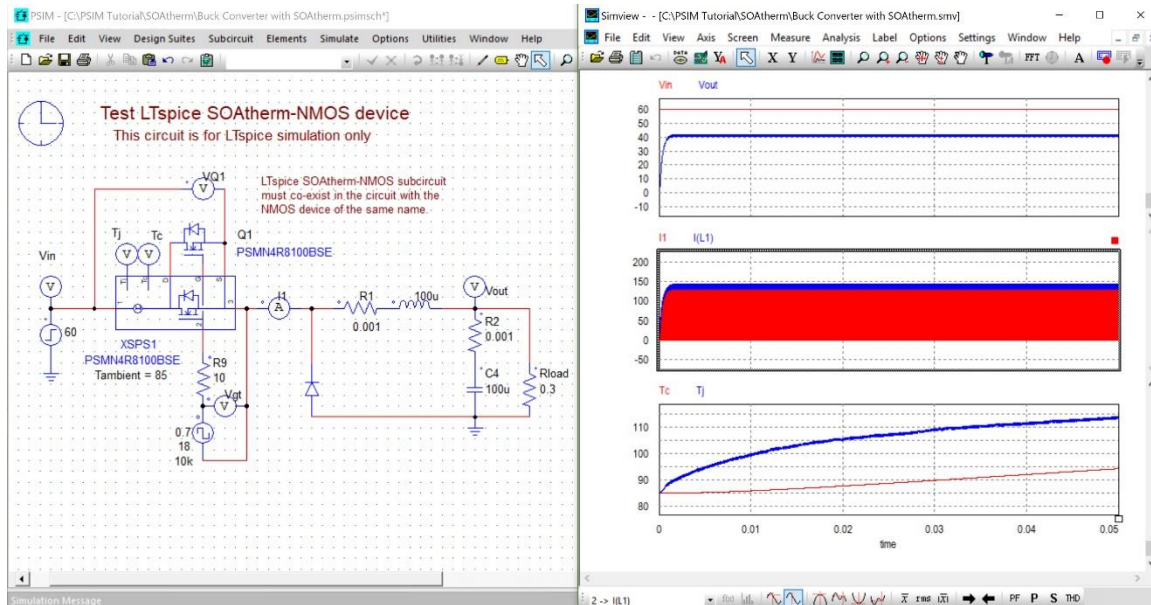
To illustrate the basic procedure of using SOAtherm models in PSIM, we will use the example “Buck converter with SOAtherm (LT).psimsch” in the PSIM folder “examples\SPICE\dc-dc”.

To keep the original example unchanged, we will create a new folder called “C:\PSIM Tutorials\SOAtherm”, and use this folder as the working folder in this tutorial. Copy the example file “Buck converter with SOAtherm (LT).psimsch” into the new folder.

In PSIM, load the file “C:\PSIM Tutorial SOAtherm\Buck Converter with SOAtherm (LT).psimsch”, as shown below. In the circuit, the diode D1 is an ideal model for simplicity. In the position of the transistor device, there is a subcircuit and a MOSFET device both named “PSMN4R8100BSE”.

The MOSFET is used to simulate the switching device’s electrical characteristics while the subcircuit represents the thermal characteristics.

Run LTspice simulation, and results are as shown below on the right. The voltage and current of the MOSFET device are VQ1 and I1; and the case and junction temperature of the device are Tc and Tj. Waveforms show that it needs more time for Tc and Tj to reach the thermal steady state.



4. Using SOA Models in PSIM Schematic

In the previous section, the MOSFET and the SOAtherm models were already in the PSIM schematic. The following is the procedure of how to put those models in PSIM schematic.

First, in the example circuit, remove the MOSFET model and the SOAtherm model.

To place the MOSFET device into PSIM schematic:

- Find a MOSFET device from the PSIM Element library by selecting **Elements** >> **Power** >> **Switches** >> **MOSFET**. Place it on the schematic.
- Double click on MOSFET to open the parameter dialog window, and select the model level as "SPICE Model".
- Click on the browse button of the "SPICE Model Name" input field. From the SPICE Model List, find and select the device "PSMN4R8100BSE".

To place the SOAtherm model into PSIM schematic:

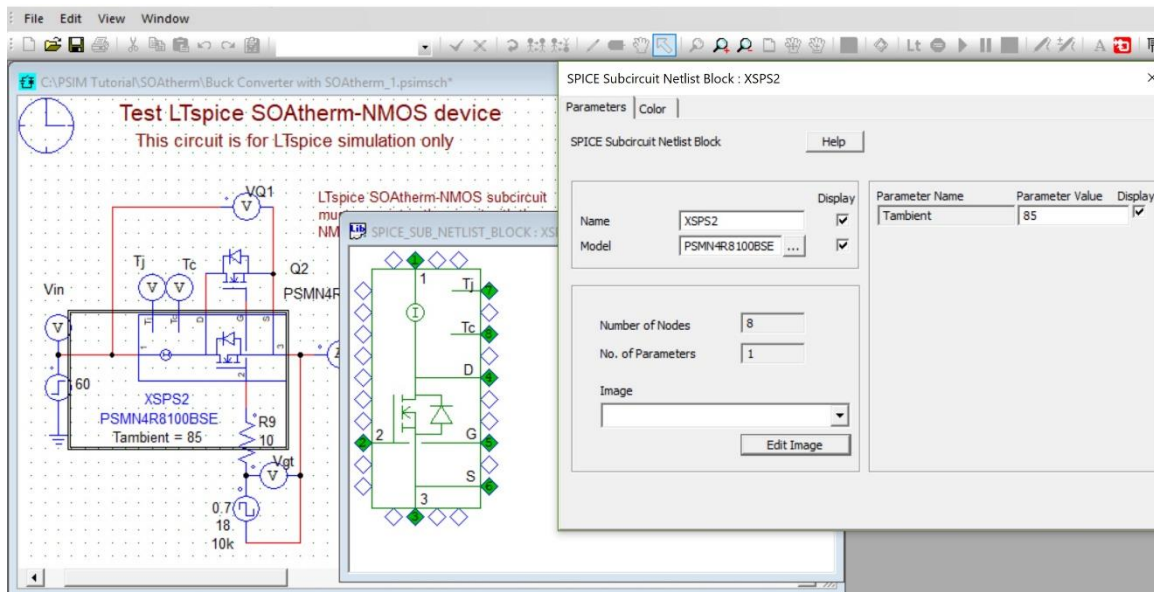
- Select **Elements** >> **SPICE** >> **SPICE Subcircuit Netlist Block** and place a SPICE subcircuit block into the schematic.
- Double click on this block to open the parameter dialog window.
- Click on the browse button of the "Model" input field. From the library list, select "SOAtherm-NMOS.lib". Then find and select the subcircuit "PSMN4R8100BSE".
- The number of nodes is automatically updated.
- The number of parameters can be modified. For this subcircuit "PSMN4R8100BSE", there are many parameters. To find out the parameter definitions, use a text editor and open the file "SOAtherm-NMOS.lib", and find the device "PSMN4R8100BSE". But not all parameters need to be defined. One can choose how many and which parameters to define for each particular case. In this example, the only parameter of concern is "Tambient". Therefore, the number of parameter is "1".
- In the parameter list, enter the parameter name as "Tambient" and the value as "85".

Now edit the image of this subcircuit by clicking on **Edit Image**. One can draw a custom image for this subcircuit. Note that the node sequence must be the same as defined in the model netlist in “SOAtherm-NMOS.lib”. Please note the following:

- The node sequence of this subcircuit model is: “D G S D2 G2 S2 Tj Tc”.
- The first 3 node “D G S” are for the connections of Drain, Gate, and Source of the SOAtherm device.
- The next 3 nodes “D2 G2 S2” are for the regular MOSFET device PSMN4R8100BSE’s Drain, Gate, and Source.
- The last 2 nodes “Tj Tc” are junction and case temperature output in voltage level.

Alternatively, you can select the pre-built image for this model. From the **Image** pull-down menu, select “MOSFET_D_G_S_D2_G2_S2_Tj_Tc”.

Below is the image for this SOAtherm model:



Now, save this PSIM file and run LTspice simulation.