

What's New with LTspice IV?

Gabino Alonso



New Blog Article: "Modeling Safe Operating Area Behavior of N-Channel MOSFETs" by Dan Eddleman
www.linear.com/solutions/5239

twitter — Follow @LTspice at www.twitter.com/LTspice
f — Like us at facebook.com/LTspice

BLOG BY ENGINEERS, FOR ENGINEERS

Check out the LTspice blog (www.linear.com/solutions/LTspice) for tech news, insider tips and interesting points of view regarding LTspice.

Modeling Safe Operating Area Behavior of N-Channel MOSFETs by Dan Eddleman

www.linear.com/solutions/5239—Verifying that a Hot Swap design does not exceed MOSFET's safe operating area (SOA) is challenging. Fortunately, thermal behavior and SOA can now be modeled in LTspice.

The new SOAtherm-NMOS symbol included in LTspice contains a collection of MOSFET thermal models that can be used to verify that the maximum die temperature is not exceeded. The SOAtherm provides MOSFETs' junction and case temperatures in °C (represented as voltage in waveform viewer) and does not

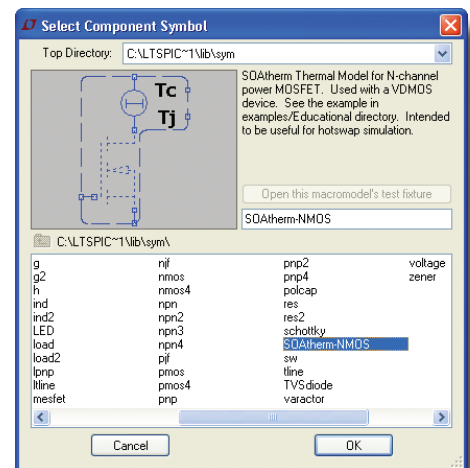
influence the electrical behavior of the circuit simulation. Even though using the SOAtherm-NMOS symbol/models is as easy as placing the symbol on top of the NMOS in an LTspice schematic and editing the component attributes, this blog provides a step-by-step tutorial and highlights some design considerations.

SELECTED DEMO CIRCUITS

For a complete list of example simulations utilizing Linear Technology's devices, please visit www.linear.com/democircuits.

Buck Switching Regulators

- **LT3840:** High efficiency synchronous buck converter (4.5V–60V to 3.3V at 20A) www.linear.com/LT3840
- **LT8620:** 5V 2MHz buck converter (5.5V–65V to 5V at 2A) www.linear.com/LT8620
- **LTC3607:** Dual monolithic synchronous buck regulator (4.5V–15V to 1.8V, 600mA & 3.3V, 600mA) www.linear.com/LTC3607
- **LTC3622:** Dual monolithic synchronous buck regulator (5V–17V to 3.3V, 1A & 5V, 1A) www.linear.com/LTC3622
- **LTC3875 & LTC3874:** High efficiency, 4-phase buck supply with sub-milliohm DCR sensing (4.5V–14V to 1V, 120A) www.linear.com/LTC3875
- **LTM4630:** High efficiency 6-phase 80A buck regulator (11V–13V to 0.95V, 80A) www.linear.com/LTM4630



Easily model the thermal behavior of MOSFETs in LTspice using the SOAtherm NMOS symbol.

Boost Switching Regulators

- **LTC3769:** High voltage 60V synchronous boost controller (6V–55V to 48V, 1A) www.linear.com/LTC3769

Flyback, Forward and Isolated Controllers

- **LTM8058:** 2kV isolated flyback converter with LDO post regulator (4.3V–29V to 5.7V, 120mA & 5V, 120mA) www.linear.com/LTM8058

LED Drivers

- **LT3796-1 & LTC1541:** SEPIC LED driver with 100:1 analog dimming (8V–20V to 35V string, 1A) www.linear.com/LT3796
- **LT3797:** Triple LED boost controller (2.7V–40V to 3x 50V LED strings, 1A) www.linear.com/LT3797

Wireless Power

- **LTC4120:** Wireless power receiver with 800mA buck battery charger www.linear.com/LTC4120

What is LTspice IV?

LTspice® IV is a high performance SPICE simulator, schematic capture and waveform viewer designed to speed the process of power supply design. LTspice IV adds enhancements and models to SPICE, significantly reducing simulation time compared to typical SPICE simulators, allowing one to view waveforms for most switching regulators in minutes compared to hours for other SPICE simulators.

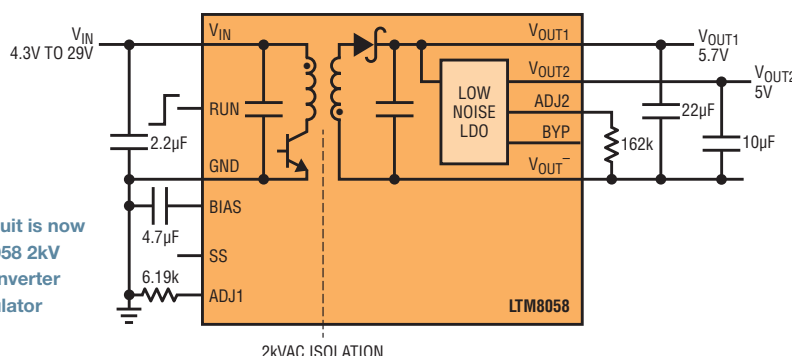
LTspice IV is available free from Linear Technology at www.linear.com/LTspice. Included in the download is a complete working version of LTspice IV, macro models for Linear Technology's power products, over 200 op amp models, as well as models for resistors, transistors and MOSFETs.

SELECTED MODELS

To search the LTspice library for a particular device model, choose Component from the Edit menu or press F2. LTspice is updated often with new models, so be sure to keep your installation of LTspice current by choosing Sync Release from the Tools menu.

- **LTC3807:** Low IQ, synchronous step-down controller with 24V output voltage capability www.linear.com/LTC3807
- **LTM4634:** Triple output 5A/5A/4A step-down DC/DC μ Module regulator www.linear.com/LTM4634

Demonstration circuit is now available for LTM8058 2kV isolated flyback converter with LDO post regulator



Buck-Boost Switching Regulators

- **LTC3114-1:** 40V, 1A synchronous buck-boost DC/DC converter with programmable output current www.linear.com/LTC3114-1

Flyback, Forward and Isolated Controllers

- **LT8310:** 100V input forward converter controller www.linear.com/LT8310

Current Sense Amplifiers

- **LT6119:** Current sense amplifier, reference and comparators with POR www.linear.com/LT6119-1

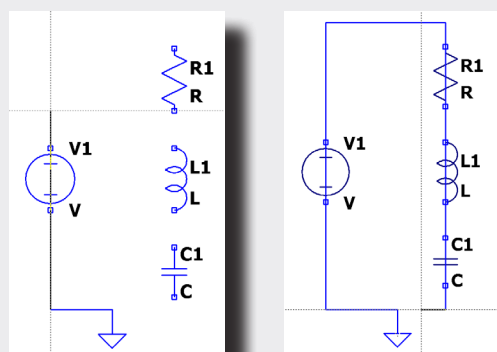
Hot Swap Controllers

- **LTC4231:** Micropower Hot Swap controller www.linear.com/LTC4231 ■

Power User Tip

CONNECTING THE DOTS

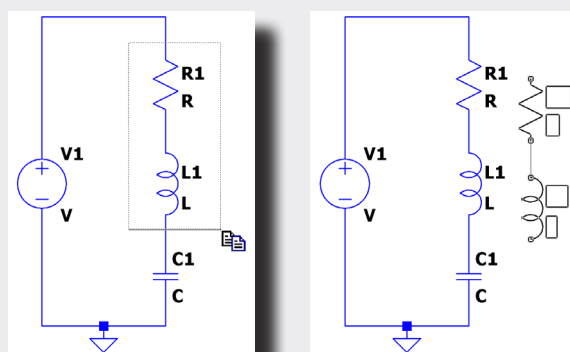
Sometimes the simplest things elude us. Typically, after placing components in an LTspice schematic you select draw wires (F3), left click to start a wire, left click again to change direction or join, repeat until your circuit is complete and then right click to cancel. But did you know you can draw wires through components like resistors and the wire will automatically be cut so that the components are in series with the wire? Drawing a wire straight through several components is an easy way to connect components in series.



COPY AND PASTE BETWEEN SCHEMATICS

Another feature not commonly understood is how to copy and paste between schematics using the duplicate command. To copy objects from one schematic to another, in the source schematic, invoke the duplicate command (F6 or Ctrl + C)—the crosshair pointer changes to the duplicate symbol, . Left-click to select the object you want to duplicate, or select a group of objects by dragging a box around them.

Once the object or section is copied, simply click in the target schematic window (or the tab) and click again to paste. In Windows, both schematics must be in the same invocation of LTspice.



Happy simulations!