What's New with LTspice IV?

Gabino Alonso

New Video: "Stability of Op Amp Circuits"
www.linear.com/solutions/4449

NOW NATIVE ON MAC OS X

LTspice for Mac OS X 10.7+ platforms is now available at www.linear.com/LTspice. This new release has similar capabilities and features as its Windows counterpart. To access the menu you can right-click or use shortcuts. A guide to Mac OS X shortcuts is also available online.

BLOG UPDATE

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

New Video on the Blog, "Stability of Op Amp Circuits"—The latest video topic is available at www.linear.com/solutions/4449.

We all know that feedback circuits can oscillate. We may even know a few tricks to fix the problem, but wouldn't it be nice if our simulation tool could show us exactly what is happening, and why? This video illustrates how to use the .AC analysis to look at open loop gain and phase of operational amplifier feedback circuits in LTspice IV. It explains how to break the feedback loop in an op amp circuit while maintaining the correct operating point so that the open loop transfer function of the circuit can be obtained and the phase margin measured. This video also covers some common techniques on how to improve the phase margin of a design and improve your circuit intuition.

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 DEMO CIRCUITS

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

Linear Regulators

- **LT3007:** 3.3V, 20mA linear regulator with shutdown (3.8V–45V to 3.3V at 20mA) www.linear.com/LT3007
- **LT3090:** Negative linear regulator with current monitor (−5V to −1.25V at 600mA) www.linear.com/LT3090

µModule Regulators

- **LTM4620A/LTM4676:** High current, parallel µModule buck regulators with power system management (4.5V–16V to 1V at100A) www.linear.com/LTM4620A
- **LTM4676:** Dual 13A µModule buck regulator with digital interface for control & monitoring (5.75V–26.5V to 1V at 13A & 1.8V at 13A) www.linear.com/LTM4676
- **LTM4630:** High efficiency 8-phase 140A step-down regulator (4.5V–15V to 1V at 140A) www.linear.com/LTM4630

Flyback Controller

- **LT8302:** µPower no-opto isolated flyback converter (10V–30V to 5V at 2.2A) www.linear.com/LT8302

LED Drivers

- **LT3955:** 20W boost LED driver with internal PWM dimming (5V–60V to 67V LED string at 300mA) www.linear.com/LT3955

Boost Regulators

- **LTC3788-1 Demo Circuit:** High efficiency dual 12V/24V boost converter with R_{SENSE} current sensing (4.5V–24V to 24V at 5A & 12V at 10A) www.linear.com/LTC3788–1

Current Sense Amplifiers

- **LT1999-20:** High voltage bidirectional current sense −5V to 80V Input) www.linear.com/LT1999
- **LT6105:** Unidirectional current sense amplifier for a 1V supplies (0A to 10A) www.linear.com/LT6105

ADC Driver

- **LTC6360:** ±10V input signal to a 5V single-ended ADC driver www.linear.com/LTC6360

Oscillator

- **LTC6991:** Low frequency voltage controlled oscillator (250Hz to 1kHz) www.linear.com/LTC6991

twitter Follow @LTspice on Twitter for up-to-date information on models, demo circuits, events and user tips: *www.twitter.com/LTspice*

Like us on Facebook at *facebook.com/LTspice*

SELECTED MODELS

Buck Regulators

- **LT8610A/AB:** 42V, 2.5A synchronous step-down regulator with 2.5µA quiescent current www.linear.com/LT8610
- **LT8614:** 42V, 4A synchronous step-down Silent Switcher™ regulator with 2.5µA quiescent current www.linear.com/LT8614
- **LTC3607:** Dual 600mA 15V monolithic synchronous step-down DC/DC regulator www.linear.com/LTC3607

Charge Pump

- **LTC3255:** Wide V_{IN} range fault protected 50mA step-down charge pump www.linear.com/LTC3255

MOSFET Driver

- **LTC4440A-5:** High speed, high voltage, high side gate driver www.linear.com/LTC4440A-5

Ideal Diodes & Hot Swap Controllers

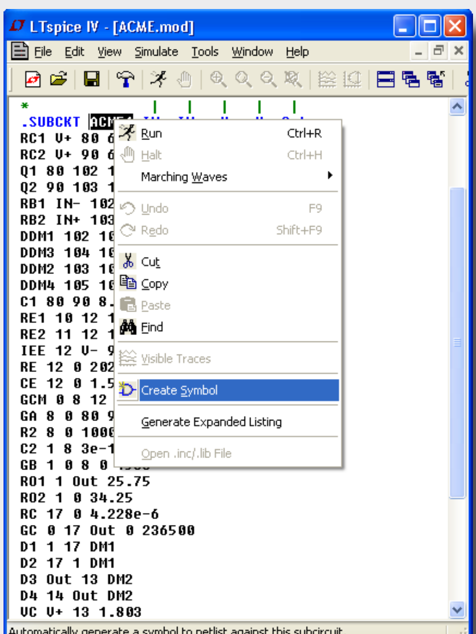
- **LTC4221:** Dual Hot Swap™ controller/power sequencer with dual speed, dual level fault protection www.linear.com/LTC4221
- **LTC4229:** Ideal diode and Hot Swap controller www.linear.com/LTC4229
- **LTC4280:** Hot Swap controller with I²C-compatible monitoring www.linear.com/LTC4280

Power User Tip

IMPORTING THIRD-PARTY MODELS

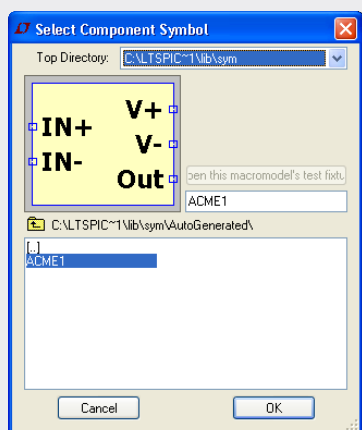
It is possible in LTspice to create a new symbol from scratch for a third-party model but who has the time. Follow theses easy steps to generate a new symbol for a third-party model defined in a subcircuit (.SUBCKT statement).

1. Open the netlist file that contains the subcircuit definitions in LTspice. (File > Open or drag file into LTspice)
2. Right-click the line containing the name of the subcircuit, and select Create Symbol:



3. Edit the symbol if needed and save.

To use the new symbol (and associated third party model) in a schematic, select the symbol from the AutoGenerated directory in the component library (F2) and place it in your schematic:



By using the automatic symbol generation you can focus on your simulations, not creating new symbols. For a more information on how to import third party models that use intrinsic SPICE device (.MODEL statement) see the video at www.linear.com/solutions/1083.

Happy simulations!