

What's New with LTspice IV?

Gabino Alonso



Stepping Parameters
in LTspice IV

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

NEW HOW-TO VIDEOS

Stepping Parameters video.linear.com/103

There are two ways to examine a circuit by changing the value of a parameter. You can manually enter each value, then resimulate the circuit, or you can use the .STEP command to sweep across a range of values in a single simulation run for a side-by-side comparison. This video provides an overview of the basic steps for using the .STEP command to perform repeated analysis of a circuit.

FULLY DIFFERENTIAL OPERATIONAL AMPLIFIER DEMO CIRCUITS (from Tyler Hutchison)

Analyzing and interfacing with fully differential op amps in simulation may prove more complicated than with familiar, single-ended output op amps. These demo circuits provide examples of interfacing to fully differential amplifiers

including single-ended and differential impedance matching, noise analyses, mixer and high speed ADC signal chain models, and gain modification of fixed-gain amplifiers with external resistors.

- **LTC6400-20:** Single-ended impedance matching for fully differential amplifiers www.linear.com/LTC6400-20
- **LTC6400-20:** Differential impedance matching for fully differential amplifiers www.linear.com/LTC6400-20
- **LTC6400-20:** Noise simulation for fully differential amplifiers www.linear.com/LTC6400-20
- **LTC6401-8:** Unity gain for fully differential amplifiers with internal feedback resistors www.linear.com/LTC6401-8
- **LTC6404-1:** Impedance matching and noise measurements for fully differential amplifiers www.linear.com/LTC6404

- **LTC6405:** Fully differential ADC driver with simplified mixer and ADC models www.linear.com/LTC6405

NEW DEMO CIRCUITS

µModule Regulators

- **LTM®8048:** Isolated µModule DC/DC converter with LDO post regulator (4V–30V to 6V at 100mA & 5V at 100mA) www.linear.com/LTM8048
- **LTM8052:** 36V, 5A, 2-quadrant CVCC step-down µModule regulator (6V–36V to 2.5V at ±5A) www.linear.com/LTM8052

Switching Regulators

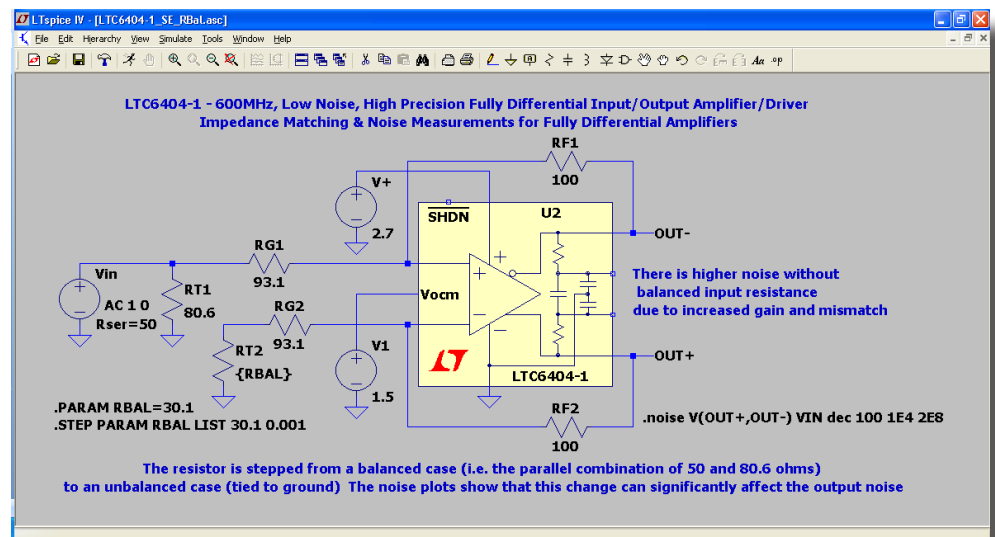
- **LT3758:** High efficiency SEPIC (8V–72V to 5V at 2A) www.linear.com/LT3758
- **LT3759:** Boost with low input voltage range (1.8V–4.5V to 5V at 2A) www.linear.com/LT3759
- **LT3759:** SEPIC with wide input voltage range (2.8V–36V to 12V at 1A) www.linear.com/LT3759

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.

Download this demo circuit showing impedance matching and noise measurements for fully differential amplifiers www.linear.com/LTC6404



- **LTC4000/LTC3891:** High voltage high current step-down, PowerPath controller & Li-ion battery charger (24V–60V to 16.8V at 6A) www.linear.com/LTC4000

Amplifiers, References, Comparators and TimerBlox® ICs

- **LT6108/LTC6994:** Energy-tripped circuit breaker with automatic delayed retry (5V–80V input, 500mA threshold) www.linear.com/LT6108

NEW DEVICE MACRO MODELS

Switching Regulators

- **LT8582:** Dual 3A boost/inverting/SEPIC DC/DC converter with fault protection www.linear.com/LT8582
- **LTC3103:** 1.8μA quiescent current, 15V, 300mA synchronous step-down DC/DC converter www.linear.com/LTC3103
- **LTC3115V-1:** 40V, 2A synchronous buck-boost DC/DC converter www.linear.com/LTC3115-1
- **LTC3613:** 24V, 15A monolithic step-down regulator with differential output sensing www.linear.com/LTC3613
- **LTC3839:** Fast, accurate, 2-phase, single-output step-down DC/DC controller with differential output sensing www.linear.com/LTC3839
- **LTC3866:** Current mode synchronous controller for sub-mΩ DCR sensing www.linear.com/LTC3866
- **LTC3876:** Dual DC/DC controller for DDR power with differential VDDQ sensing and ±50mA VTT reference <http://www.linear.com/LTC3876>

Hot Swap™ Controllers and Surge Stoppers

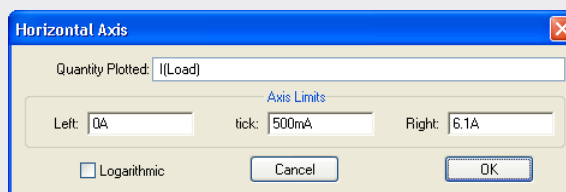
- **LTC4232:** 5A integrated Hot Swap controller www.linear.com/LTC4232
- **LTC4366:** High voltage surge stopper www.linear.com/LTC4366 ■

PARAMETRIC PLOTS

Plotting results in LTspice IV is as easy as clicking on a node to show voltage, or a component to show current—the trace is then displayed in the waveform viewer. In transient analysis, the horizontal axis defaults to showing time, but you can always change the horizontal axis to show other quantities (such as current) to validate model parameters.

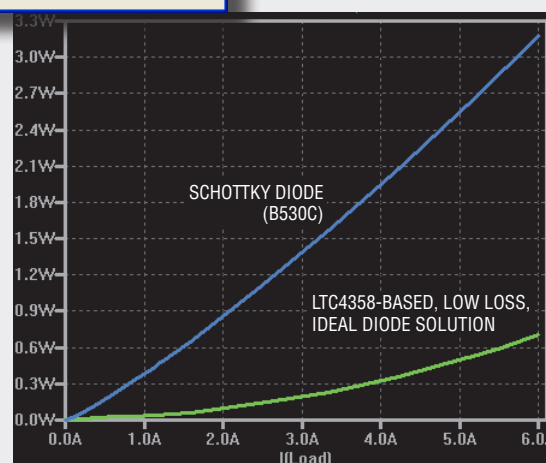
To change the default settings of the x-axis:

1. Click on a node/component to plot its voltage/current in the waveform viewer.
2. Move the cursor to the horizontal axis of the waveform viewer (the cursor will turn into a ruler) and left-click.
3. In the Horizontal Axis dialog, enter an expression for the “Quantity Plotted.”
4. Click OK.



Happy simulations!

This parametric plot compares the instantaneous power dissipation (Alt + click on component) of an LTC4358 5A ideal diode to a Schottky diode. In this example, the quantity plotted on the horizontal axis has been changed from the default of “time” to “load current.” The resulting plot confirms the advantage of the LTC4358 as a low loss replacement to Schottky diodes in high current diode applications. More information is available at www.linear.com/LTC4358.



Power User Tip

Download the LTspice IV demonstration circuit for this energy-tripped circuit breaker with automatic delayed retry at www.linear.com/LT6108

