What's New with LTspice IV?

Gabino Alonso

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

LTspice® IV is a high performance SPICE simulator, schematic capture and waveform viewer specifically designed to speed up 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.

What is LTspice IV?

COOKING WITH LTspice IV SEMINAR TAKES WORLD TOUR

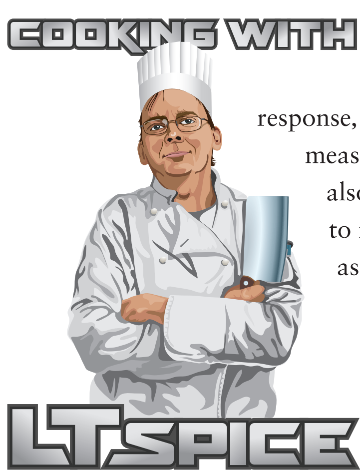
Mike Engelhardt, the author and creator of LTspice IV, is embarking on a world tour to teach you the ins and outs of LTspice IV in a series of free half-day seminars. At each seminar, Mr. Engelhardt will show you how to quickly simulate switch mode power supplies, compute efficiencies and observe power supply start-up behavior and transient response. You will also learn how to use LTspice IV as a general-purpose SPICE simulator for AC analysis, DC sweeps, noise analysis and circuit simulations. The presentation includes a description of the algorithms used in LTspice IV to give you a unique and powerful perspective on the inner workings of LTspice IV.

For more information on these upcoming seminars and other events please visit www.linear.com/LTspiceEvents.

NEW HOW-TO VIDEOS

One of the fastest ways to get started with LTspice IV and learn a few user tips, is to watch the instructional videos available at www.linear.com/LTspiceVideos. Two new videos are now available:

- The first new instructional video covers the *LTspice IV Schematic Editor* (video.linear.com/84). This video shows how to use the LTspice IV schematic capture program in the layout of a simple circuit so you can quickly draft and make edits to your design.
- The second video covers the *LTspice IV Waveform Viewer* (video.linear.com/88). This video shows you how to quickly probe the circuit for current and voltage response, and how to view and measure the waveforms. It also includes techniques to navigate the waveforms as you analyze results.



NEW DEVICE MODELS

To update your installation of LTspice IV with the latest models, choose Sync Release from the Tools menu in LTspice IV. Here is a list of some new models:

LT3029: Dual 500mA/500mA low dropout, low noise, µpower linear regulator www.linear.com/3029

LT6109-1/LT6109-2: High side current sense amplifier with reference and comparators www.linear.com/6109

LT3970-3.3/LT3970-5: 40V, 350mA step-down regulator with 2.5µA quiescent current and integrated diodes www.linear.com/3970

LTC3618: Dual 4MHz, ±3A synchronous buck converter for DDR termination www.linear.com/3618

LT6107: High temperature, high side current sense amp in SOT-23 www.linear.com/6107

LTC4225-1/LTC4225-2: Dual ideal diode and Hot Swap controller www.linear.com/4225

LTC3867: Synchronous step-down DC/DC controller with differential remote sense and nonlinear control www.linear.com/3867

LTC3388-1/LTC3388-3: 20V high efficiency nanopower step-down regulator www.linear.com/3388

LTC3634: Dual 15V, 3A monolithic step-down regulator for DDR power www.linear.com/3634

Get the Schedule

Two new LTspice IV how-to videos are now available

LTspice IV Schematic Editor

LTspice IV Waveform Viewer

COMPUTING THE AVERAGE OR RMS VALUE OF A TRACE IN LTSPICE IV

The LTspice IV waveform viewer can integrate a trace to produce the average or RMS value over a given region.

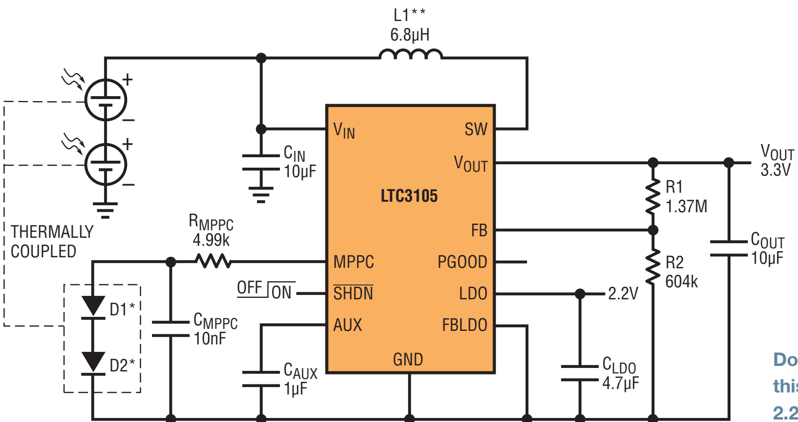
To integrate a trace in the waveform viewer:

1. Zoom in to the region of interest.
2. Hold down the control key and click the label of the trace you want to integrate.

Based on the physical units of the data trace, LTspice IV displays a meaningful average for that type of data. For example, if the units are a voltage or current, LTspice IV displays the average and the RMS values. Otherwise, LTspice IV displays the average and integral of the data displayed in the waveform viewer. If you're plotting noise densities from a .noise simulation, LTspice IV shows total RMS noise.

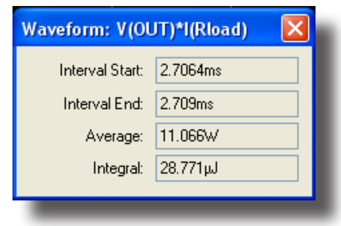
Happy simulations!

LTspice IV Power-User Tip



* MRA4003T3
** PANASONIC ELL-VEG6R8N

It's easy to calculate the RMS or average value of a waveform trace in LTspice IV. For more information, see the LTspice IV Power-User Tip above.



Download the LTspice IV demonstration circuit for this 2-cell photovoltaic to dual output, 3.3V and 2.2V, converter at www.linear.com/3105

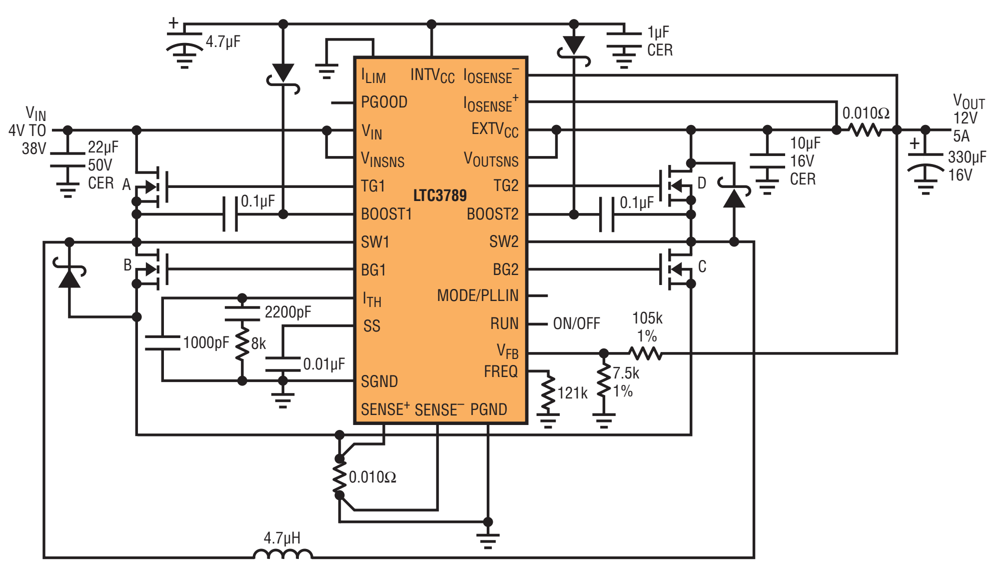
LTC3617: ±6A monolithic synchronous step-down regulator for DDR termination www.linear.com/3617

LTM®4613: EN55022B-compliant, 36V input, 15V, 8A output, DC/DC µModule regulator www.linear.com/4613

NEW LTspice IV DEMO CIRCUITS

The LTspice IV circuit collection is available at www.linear.com/DemoCircuits. Here are some of the new demonstration circuits now available:

- 36V to 12V, 8A integrated step-down DC/DC converter using the LTM4613. www.linear.com/4613
- 5V to 12V, 900mA step-up DC/DC converter using the LT3581. www.linear.com/3581
- Automotive ±30V supply protection circuit with 3.5V undervoltage and 18V overvoltage using the LTC4365. www.linear.com/4365
- 4V–15V to 1.8V, 2.5A monolithic synchronous step-down DC/DC converter using the LTC3603. www.linear.com/3603
- A 4.5V–10V to 2.5V, 2.5A monolithic synchronous step-down DC/DC using the LTC3602. www.linear.com/3602
- A dual synchronous step-up converter that takes a 5V–24V input to 24V, 3A–5A and 12V, 8A–10A using the LTC3788. www.linear.com/3788 ■



The LTspice IV demonstration circuit for this 12V, 5A automotive high efficiency buck-boost DC/DC solution with programmable output current limit is available at www.linear.com/3789