

# What's New with LTspice IV?

Gabino Alonso

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

## NEW HOW-TO VIDEOS

### Using Transformers in LTspice® IV [video.linear.com/93](http://video.linear.com/93)

Transformers and coupled inductors are key components in many switching regulator designs, including flyback, forward and SEPIC converters. Although it is possible to make a dedicated subcircuit for a specific transformer, it is often better in LTspice IV to define a separate inductor for each transformer winding, and then couple them all together magnetically via a single mutual inductance ( $\kappa$ ) statement. This video shows how to define a transformer using inductors and specify the mutual inductance via a single  $\kappa$  statement in your LTspice IV simulations.

Two new LTspice IV how-to videos are now available



Transformers



3rd-Party Models

## What is LTspice IV?

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](http://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.

### Adding Third-Party Models to LTspice IV [video.linear.com/97](http://video.linear.com/97)

LTspice IV includes models for many discrete components, such as transistors and MOSFETs, but many component manufacturers make additional models that you can add to your LTspice IV circuit simulations. These third-party SPICE models are described in .MODEL and .SUBCKT statements. This video provides an overview of how to add a third-party .MODEL statement for an intrinsic SPICE device and how to add and create a symbol for a third party .SUBCKT statement.

## NEW LTspice DEVICE MACRO MODELS

To update your installation of LTspice IV with the latest models, choose Sync Release from the Tools menu. You can review the changelog.txt after Sync Release for the complete list of new models. Here is a list of some new models:

### µModule Regulators

- **LTM8048:** 3.1V to 32V input isolated µModule DC/DC converter with LDO post regulator [www.linear.com/8048](http://www.linear.com/8048)

- **LTM8047:** 3.1V to 32V isolated µModule DC/DC converter [www.linear.com/8047](http://www.linear.com/8047)

### Switching Regulators

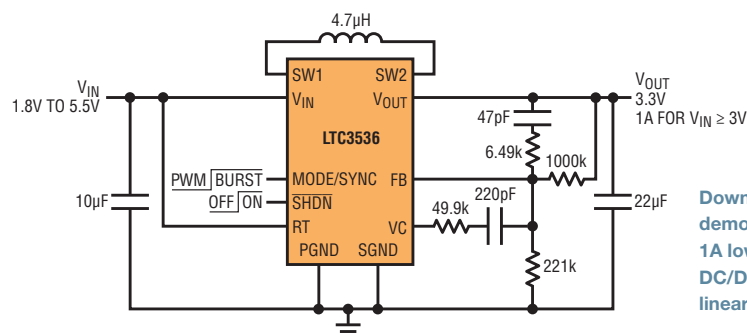
- **LTC3765:** Active clamp forward controller and gate driver [www.linear.com/3765](http://www.linear.com/3765)
- **LTC3766:** High efficiency, secondary-side synchronous forward controller [www.linear.com/3766](http://www.linear.com/3766)
- **LT®3759:** Wide input voltage range boost/SEPIC/inverting controller [www.linear.com/3759](http://www.linear.com/3759)
- **LTC3536:** 1A low noise, buck-boost DC/DC converter [www.linear.com/3536](http://www.linear.com/3536)
- **LT3507A:** Triple monolithic step-down regulator with LDO [www.linear.com/3507A](http://www.linear.com/3507A)

### Linear Regulators

- **LT3015:** 1.5A, low noise, negative linear regulator with precision current limit [www.linear.com/3015](http://www.linear.com/3015)

### Amplifiers & Comparators

- **LT6108:** High side current sense amplifier with reference and comparator [www.linear.com/6108](http://www.linear.com/6108)
- **LTC6360:** Very low noise single-ended SAR ADC driver with true zero output [www.linear.com/6360](http://www.linear.com/6360)



Download the LTspice IV demonstration circuit for this 1A low noise, buck-boost DC/DC converter at [www.linear.com/3536](http://www.linear.com/3536)

## HOW TO USE THE .STEP COMMAND TO PERFORM REPEATED ANALYSIS

There are two ways to examine a circuit by changing the value for a particular parameter: you can either manually enter each value and then resimulate the circuit to view the response, or use the .step command to sweep across a range of values in a single simulation run.

The .step command causes an analysis to be repeatedly performed while stepping through a model parameter, global parameter or independent source. Here is an example waveform response of an RC circuit, for which the capacitance is stepped through three values.

To implement this in LTspice IV, perform the following steps:

Define the component parameter with a variable by editing the component attribute (Ctrl-right-click on the component) and entering "{X}" for the

Value, where "X" is a user defined variable name. The addition of the curly braces around the variable is important as it tells LTspice IV that "X" is a parameter.

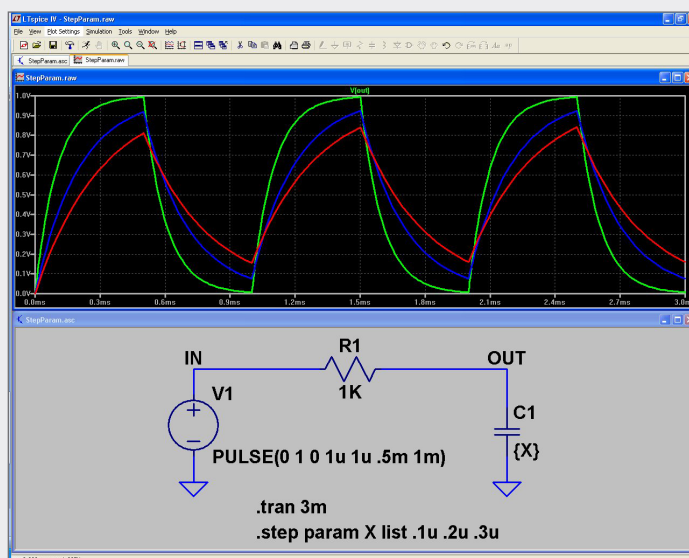
Add a .step command via a SPICE directive that specifies the steps for the parameter "X" by a linear, logarithmic or list of values.

Example A: ".step param X list .1u .2u .3u" steps the parameter X through each value listed.

Example B: ".step param X .1u .3u .1u" steps the parameter X from 0.1u to 0.3u in 0.1u increments.

For more information on how to use the .step command to improve your understanding of a schematic, review the Help Topics in LTspice IV.

Happy simulations!



## LTspice IV Power User Tip

## NEW LTspice IV DEMO CIRCUITS

The LTspice IV circuit collection is available at [www.linear.com/DemoCircuits](http://www.linear.com/DemoCircuits).

Here are some of the new demonstration circuits now available:

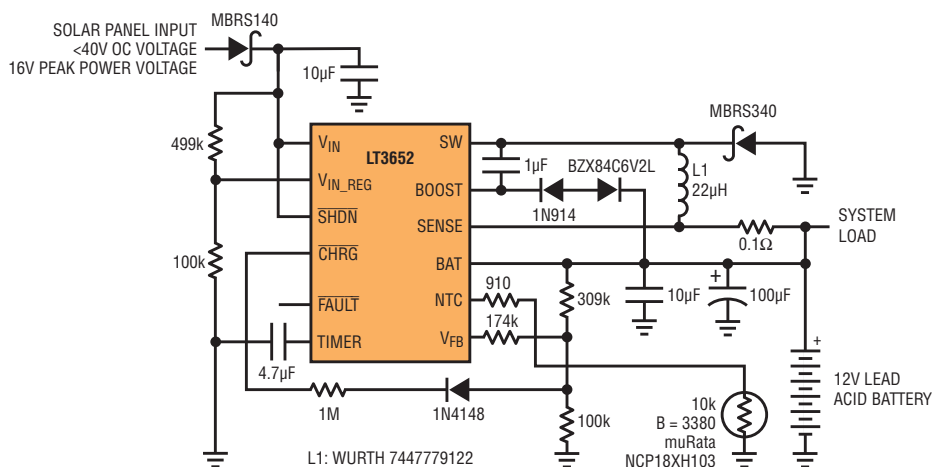
## Switching Regulators

- **LTC3618:** Dual monolithic synchronous step-down converter for DDR termination (2.25V–5.5V to  $V_{DDQ}$  at  $\pm 3A$ ,  $V_{TTR}$  at  $\pm 10mA$ ,  $V_{TT}$  at  $\pm 3A$ ) [www.linear.com/3618](http://www.linear.com/3618)
- **LTC3617:** Monolithic synchronous step-down regulator for DDR termination (2.25–5.5V to  $V_{TTR}$  at 10mA,  $V_{TT}$  at  $\pm 6A$ ) [www.linear.com/3617](http://www.linear.com/3617)

- **LTC3536:** 1A low noise, buck-boost DC/DC converter (1.8V–5.5V to 3.3V at 1.0A) [www.linear.com/3536](http://www.linear.com/3536)

## Battery Charger

- **LT3652:** 1A solar-panel-powered 3-stage 12V lead-acid fast/float charger (10V–16V to 12V at 1A) [www.linear.com/3652](http://www.linear.com/3652)

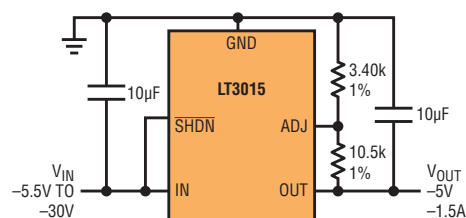


The LTspice IV demonstration circuit for this 1A solar-panel-powered 3-stage 12V lead-acid fast/float charger is available at [www.linear.com/3652](http://www.linear.com/3652)

## Linear Regulators

- **LT3032:** Dual 150mA Positive/Negative Low Noise LDO Linear Regulator (5V to 3.3V at 0.15A & –5V to –3.3V at 0.15A) [www.linear.com/3032](http://www.linear.com/3032)
- **LT3029:** Dual 500mA/500mA LDO, low noise,  $\mu$ power linear regulator (3V to 1.8V at 0.5A & 3V to 1.5V at 0.5A) [www.linear.com/3029](http://www.linear.com/3029)

- **LT3015:** 1.5A, low noise, negative linear regulator with precision current limit (–7V to –5.0V at –1.5A) [www.linear.com/3015](http://www.linear.com/3015) ■



The demonstration circuit for this 1.5A, low noise, negative linear regulator with precision current limit is available at [www.linear.com/3015](http://www.linear.com/3015)