What’s New with LTspice IV?

Gabino Alonso

World Tour
www.linear.com/LTspiceEvents

New Video: “AC Analysis”
www.linear.com/solutions/4581

LTspice IV WORLD CIRCUIT SEMINAR TAKES WORLD TOUR

Mike Engelhardt, the creator of LTspice, is embarking on a world tour to teach the ins and outs of LTspice in a series of free half-day seminars. Each seminar will cover 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 as a general-purpose SPICE simulator for AC analysis, noise analysis and circuit simulations. The presentation includes perspectives on the inner workings of LTspice IV and its capabilities.

For more information on these upcoming seminars, please visit www.linear.com/LTspiceEvents.

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.

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.

New Video on the Blog: “AC Analysis” — the latest video topic is available at www.linear.com/solutions/4581.

Sometimes the frequency response of a circuit is more important than looking at the individual voltages or currents at a specific part of the schematic. LTspice can help you achieve this with its AC analysis function. This video shows how to perform a basic AC analysis in LTspice as well as pointing out some new capabilities.

SELECTED DEMO CIRCUITS

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

Step-Down Regulators

- **LT8610AB:** 5V 2MHz µPower step-down converter with light load efficiency (5.5V–42V to 5V at 3.5A) www.linear.com/LT8610A
- **LTM®4624:** 4A step-down µModule® regulator (4V–14V to 1.5V at 4A) www.linear.com/LTM4624
- **LTM4644:** Quad 4A step-down µModule regulator (4V–14V to 3.3V, 2.5V, 1.5V and 1.2V at 4A) www.linear.com/LTM4644
- **LTM4649:** 10A step-down µModule regulator (4.5V–16V to 1.5V at 10A) www.linear.com/LTM4649

Isolated Controller

- **LTC3765 & LTC3766:** 120W isolated forward converter with synchronous rectification (9V–36V to 12V at 10A) www.linear.com/LTC3765

Boost Regulators

- **LT3905:** Adjustable APD bias supply (2.7V–12V to 54V at 1mA) www.linear.com/LT3905
- **LTC3862-1:** High power, high voltage, 4-phase boost converter (6V–36V to 50V at 10A) www.linear.com/LTC3862-1

Inverting Regulators

- **LTC3805-5 & LT1797:** Positive-to-negative Cuk converter (8V–16V to –12V at 3A) www.linear.com/LTC3805-5
- **LTC3863:** Low I_Q inverting DC/DC converter (4.5V–16V to –12V at 1A) www.linear.com/LTC3863

Constant Current, Constant Voltage Regulators

- **LT3795:** Short-circuit robust boost LED driver with spread spectrum frequency modulation (8V–60V to 87V LED string at 400mA) www.linear.com/LT3795
- **LTC4000-1 & LT3845A:** Battery charger for three $LiFePO_4$ cells with a solar panel input (20V–60V to 10.8V float at 10A max) www.linear.com/LTC4000-1

Overvoltage and Overcurrent Protection

- **LTC4366-2:** Surge protected automotive 12V supply (9V–100V to 18V clamp at 4A) www.linear.com/LTC4366

For up-to-date information on models, demo circuits, events and user tips:
twitter—Follow @LTspice on Twitter *www.twitter.com/LTspice*
f—Like us on Facebook at *facebook.com/LTspice*

Operational Amplifiers

- **LT6105:** Current sense monitor for +15V and −15V supplies (0A to 2A) and www.linear.com/LT6105
- **LTC6090 & LTC2054:** µV preamplifier for a digital voltmeter www.linear.com/LTC6090

SELECT MODELS

Linear Regulators

- **LT3086:** 40V, 2.1A low dropout adjustable linear regulator with monitoring and cable drop compensation www.linear.com/LT3086

Buck Regulators

- **LTC3875:** Dual, 2-phase, synchronous controller with low value DCR sensing and temperature compensation www.linear.com/LTC3875
- **LTM4633:** Triple 10A step-down DC/DC µModule regulator www.linear.com/LTM4633

Constant Current/Constant Voltage Regulators

- **LT3797:** Triple output LED driver controller www.linear.com/LT3797
- **LTC4020:** 55V buck-boost multi-chemistry battery charger www.linear.com/LTC4020

Wireless Power Transfer

- **LTC4120:** Wireless power receiver and 400mA buck battery charger www.linear.com/LTC4120

SIMULATING TRANSFORMERS

Here is the simple approach to simulate a transformer in LTspice:

1. Draft an inductor for each transformer winding
2. Couple them using a single mutual inductance (K) statement via a SPICE directive:

   ```
   K1 L1 L2 L3 1
   ```

   The last entry in the K statement is the coupling coefficient, which can vary between 0 and 1, where 1 represents no leakage inductance. For practical circuits, it is recommended you start with a coupling coefficient of 1.

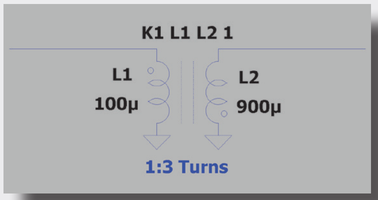
   Only a single K statement is needed per transformer; LTspice applies a single coupling coefficient to all inductors within a transformer. The following is an equivalent to the statement above:

   ```
   K1 L1 L2 1
   K2 L2 L3 1
   K3 L1 L3 1
   ```

3. Adjust the inductor positions to match the transformer polarity by using move (F7), rotate (Ctrl + R) and mirror (Ctrl + E) commands. Adding the K statement displays the phasing dot of the included inductors.
4. LTspice simulates the transformer using individual component values, in this case, the inductance of the individual inductors, not the turns ratio of the transformer. The inductance ratio corresponds to the turns ratio as follows:

$$\frac{L_{PRIMARY}}{L_{SECONDARY}} = \left(\frac{N_{PRIMARY}}{N_{SECONDARY}}\right)^2$$

For example, for a 1:3 turns ratio, enter inductance values to produce a one to nine ratio:



For more information on how to simulate a transformer see the video at www.linear.com/solutions/1079

Happy simulations!

Power User Tip

Energy Harvesting

- **LTC3330:** µPower buck-boost DC/DC with energy harvesting battery life extender www.linear.com/LTC3330

Overvoltage & Overcurrent Protection

- **LTC4365-1:** Overvoltage, undervoltage and reverse supply protection controller www.linear.com/LTC4365

Transceivers

- **LTC2862, LTC2863, LTC2864 & LTC2865:** ±60V fault protected 3V to 5.5V RS485/RS422 transceivers www.linear.com/LTC2865

Operational Amplifiers

- **LT6238:** Rail-to-rail output 215MHz, 1.1nV/$\sqrt{Hz}$ op-amp/SAR-ADC driver www.linear.com/LT6238 ■