What's New with LTspice IV?

Gabino Alonso

Blog by Engineers, for Engineers
www.linear.com/solutions/LTspice

twitter—Follow @LTspice at *www.twitter.com/LTspice*
—Like us at *facebook.com/LTspice*

NEW VIDEO: "BEHAVIORAL VOLTAGE SOURCES" by Simon Bramble

Nearly all circuit simulations require a voltage source. This video reveals some of the undiscovered talents of the not-so-humble LTspice® voltage source, specifically exploring the power of the behavioral voltage source. A behavioral voltage source outputs a voltage according to any number of circuit parameters, and it can be used to unleash the real mathematical power of LTspice.
www.linear.com/solutions/6106

SELECTED DEMO CIRCUITS

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

Buck Regulators

- **LT8602:** Automotive quad buck regulator (5.5V–42V to 5V at 1.5A, 3.3V at 2.5A, 1.8V at 1.8A, 1.25V at 1.8A) www.linear.com/solutions/5835
- **LTC3649:** High voltage monolithic synchronous buck regulator with cable drop compensation (4V–60V to 5V at 4A) www.linear.com/solutions/7117
- **LTM®4622:** Dual step-down regulator (3.6V–20V to 3.3V & 1.2V at 2.5A) www.linear.com/solutions/5847
- **LTM4675:** Paralleled µModule buck regulator with digital interface (10V–14V to 1V at 72A) www.linear.com/solutions/5833

Boost Regulators

- **LT3095:** Dual low noise, low ripple bias generator (3V–20V to 5V & 15V at 50mA) www.linear.com/solutions/6001
- **LT8331:** 120V boost converter (36V–72V to 120V at 60mA) www.linear.com/solutions/6022

SEPIC Regulator

- **LT8331:** 48V SEPIC converter (36V–72V to 48V at 165mA) www.linear.com/solutions/6019

Buck-Boost Regulator

- **LTM8055:** Paralleled synchronous buck-boost regulator with accurate current limit (7V–36V to 12V at 12A) www.linear.com/solutions/5617

4-Quadrant Converter

- **LT8714:** Synchronous four quadrant converter with power good indication (10V–14V to –5V to 5V at –5A to 5A) www.linear.com/solutions/6004

Operational Amplifier

- **LTC6268-10:** Oscilloscope differential probe www.linear.com/solutions/6058

SELECTED MODELS

To search the LTspice library for a particular device model, press F2. Since LTspice is often updated with new features and models, it is good practice to update to the current version by choosing Sync Release from the Tools menu.

Buck Regulators

- **LT8641:** 65V, 3.5A synchronous step-down Silent Switcher® with 2.5µA quiescent current www.linear.com/LT8641
- **LTM4650:** Dual 25A or single 50A DC/DC µModule regulator www.linear.com/LTM4650

Inverting Buck Regulator

- **LTC7149:** 60V, 4A synchronous step-down regulator for inverting outputs www.linear.com/LTC7149

Isolated Flyback Converter

- **LTM8068:** 2.8V to 40V input isolated µModule DC/DC converter with LDO post regulator www.linear.com/LTM8068

Charge Pumps

- **LTC3265:** Low noise dual supply with boost and inverting charge pumps www.linear.com/LTC3265

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.

Hot Swap Controller

- **LTC4281:** Hot swap controller with I²C compatible monitoring www.linear.com/LTC4281

Ideal Diode-OR Controller

- **LTC4371:** Dual negative voltage ideal diode-OR controller and monitor www.linear.com/LTC4371

Comparator

- **LTC6754:** High speed rail-to-rail input comparator with LVDS compatible outputs www.linear.com/LTC6754

PoE Powered Device

LT4276A: LTPoE++®/PoE+/PoE PD forward/flyback controller www.linear.com/LT4276

Op Amp

- **LT6375:** ±270V common mode voltage difference amplifier www.linear.com/LT6375

Voltage Reference

- **LT6657:** 1.5ppm/°C drift, low noise, buffered reference www.linear.com/LT6657 ■

Power User Tip

AC ANALYSIS USING THE STEP COMMAND

In LTspice, AC analysis involves computing the AC complex node voltages as a function of frequency using an independent voltage or current source as the driving signal. The small signal analysis results are plotted in the waveform viewer as magnitude and phase over frequency.

AC analysis in LTspice has a number of settings: the x-axis scaling (linear, octave or decade), number of simulation points and frequency range. For example, if you want to see how your circuit performs from 100Hz to 1MHz with 1,000 points per decade you would edit your simulation command to the following:

```
.ac dec 1K 100 1Meg
```

Repeated AC Analysis with Parameter Sweeps

AC analysis usually involves using fixed parameters to calculate the small signal AC response of a circuit, but you may want to refine your design by viewing the response under varying parameters. This can be accomplished by stepping the parameter of interest using a .step command. For example, you could sweep a capacitance logarithmically through the range of 10pF to .5nF with 30 points per octave using the .step directive (press S to insert a spice directive in the schematic editor):

```
.step oct param C 10p .5n 30
```

The schematic for this case and its resulting waveform are shown below.

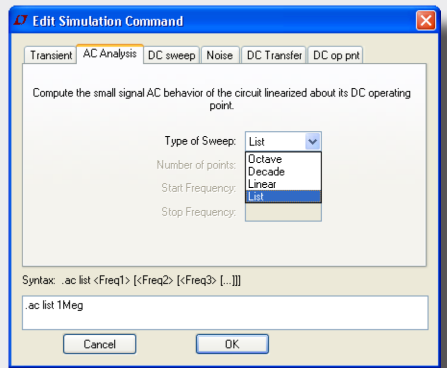
Note that using a .step command with AC analysis can drastically increase simulation time, so carefully choose the values, ranges, increments, and frequency range for each parameter sweep.

Single Frequency Analysis with a Swept Parameter

LTspice offers an elegant solution for holding frequency constant and performing small signal analysis over a varying parameter. It is as simple as using the 'list' AC Analysis option in the simulation command, and specifying the frequency at which you want to perform the analysis, in this case, 1MHz:

```
.step oct param C 10p .5n 30
.ac list 1Meg
```

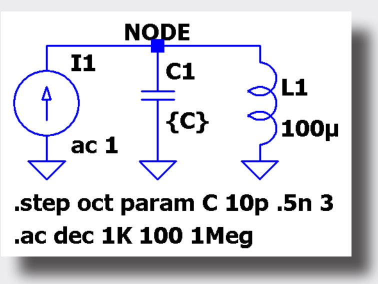
The beauty of single frequency analysis with a .step command is that the resulting plot shows magnitude and phase as a function of parameter sweep, not frequency. Below is the result of a simulation using a single frequency analysis where the x-axis is the capacitance sweep as defined in the .step function.

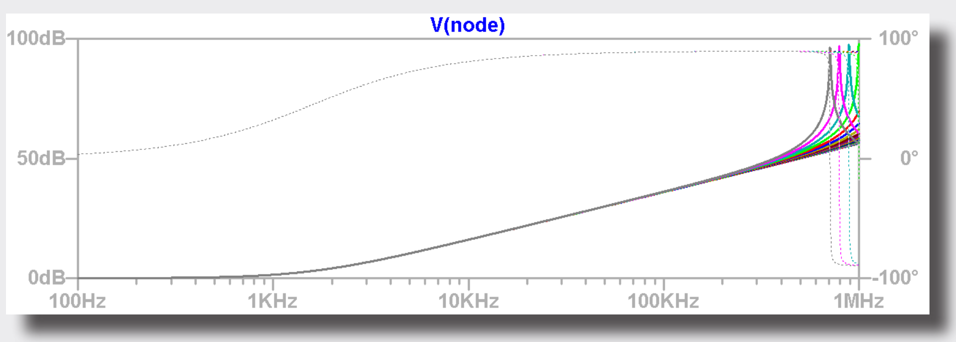


AC analysis commands can be edited using the Edit Simulation Command dialog.

Happy simulations!

Repeated AC analysis with stepped capacitance





Single frequency analysis with swept capacitance