What's New with LTspice IV?

Gabino Alonso

www.linear.com/blog/LTspice

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

LTspice BLOG

Check out the new LTspice blog (www.linear.com/blog/LTspice) for tech news, insider tips and interesting points of view regarding LTspice. Here are just a few of the topics:

- Simulating Power Planes
- Parametric Plots
- Importing & Exporting Data
- Noise Simulations
- Adding Third-Party Models

SELECTED DEMO CIRCUITS

Linear Regulators

- **LT3055:** 5V supply with 497mA precision current limit, 10mA I_{MIN} (5.4V–45V to 5V at 497mA) www.linear.com/LT3055
- **LT3081:** Extended safe operating area supply (2.7V–40V to 1.5V at 1.5A) www.linear.com/LT3081

Buck Regulators

- **LT3514:** 36V triple buck regulator (5.4V–36V to 5V at 1A, 3.3V at 2A and 1.8V at 1A) www.linear.com/LT3514
- **LT3995:** 3.3V step-down converter (4.3V–60V to 3.3V at 3A) www.linear.com/LT3995
- **LT8697:** 2MHz 5V step-down converter with cable drop compensation (6V–42V to 5V at 2.1A) www.linear.com/LT8697

LED Driver

- **LT3761:** 94% efficient boost LED driver for automotive headlamp with 25:1 PWM dimming (8V–60V to 60V LED string at 1A) www.linear.com/LT3761

Supercapcitor Charger

- **LTC3122:** Dual supercapacitor backup power supply (0.5V–5V to 5V at 50mA) www.linear.com/LTC3122

µModule Regulators

- **LTM®4637:** High efficiency 20A µModule buck regulator (4.5V–20V to 1.2V at 20A) www.linear.com/LTM4637
- **LTM8028:** Low output noise, 1.8V, 5A regulator (6V–36V to 1.8V at 5A) www.linear.com/LTM8028
- **LTM8045:** –5V inverting converter (2.8V–18V to –5V at 430mA) www.linear.com/LTM8045
- **LTM8050:** 5V step-down converter (7.5V–58V to 5V at 2A) www.linear.com/LTM8050

Linear Regulator

- **LT3030:** Dual, µPower, low noise linear regulator (2.2V–20V to 1.8V at 750mA and 1.5V at 250mA) www.linear.com/LT3030

TimerBlox® Silicon Timing Devices

- **LTC6995-1:** Active low power-on reset timer (1s POR) www.linear.com/LTC6995–1

Precision Amplifiers

- **LTC6090 and LT5400:** Wide common mode range 10x gain instrumentation amplifier www.linear.com/LTC6090

SELECTED MODELS

Buck Regulators

- **LT3514:** Triple step-down switching regulator with 100% duty cycle operation www.linear.com/LT3514
- **LT3995:** 60V, 3A, 2MHz step-down switching regulator with 2.7µA quiescent current www.linear.com/LT3995
- **LT8697:** USB 5V 2.5A output, 42V input synchronous buck with cable drop compensation www.linear.com/LT8697
- **LTC3374:** 8-channel parallelable 1A buck DC/DCs www.linear.com/LTC3374

LED Driver

- **LT3954:** 40V input LED converter with internal PWM generator www.linear.com/LT3954

Inverting Regulators

- **LTC3863:** 60V low IQ inverting DC/DC controller www.linear.com/LTC3863

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.

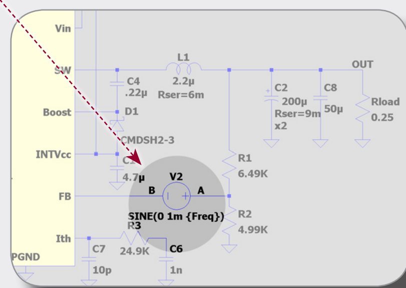
Power User Tip

GENERATING A BODE PLOT OF A SWITCH MODE POWER SUPPLY IN LTspice IV

Determining the open loop gain from a closed loop switch mode power supply (SMPS) is best solved using Middlebrook's method, which appears in the *International Journal of Electronics*, Volume 38, Number 4, 1975. This method injects test signals into the closed loop system to independently solve for the voltage and current gains so that the loop remains closed and operating points undisturbed. Using the voltage gain portion of the Middlebrook method is particularly useful in performing a frequency response analysis (FRA) of an SMPS in LTspice.

To perform a FRA of a switch mode power supply in LTspice:

- Insert a voltage source with a value of "SINE(0 1m {Freq})" in the SMPS feedback loop in series with the feedback pin and label the nodes of this voltage source "A" and "B" as shown. The choice of amplitude (1mV) impacts accuracy and the signal to noise ratio. Lower amplitudes lower the signal to noise and the larger the amplitude the less relevant the frequency response will be. A good starting point is 1mV to 20mV.



- Paste the following .measure statements on the schematic as a SPICE directive. These statements perform the Fourier transform of nodes A and B, compute the complex open loop gain of the SMPS, resulting magnitude in dB and phase in degrees.

```
.measure Aavg avg V(a)
.measure Bavg avg V(b)
.measure Are avg (V(a)-Aavg)*cos(360*time*Freq)
.measure Aim avg -(V(a)-Aavg)*sin(360*time*Freq)
.measure Bre avg (V(b)-Bavg)*cos(360*time*Freq)
.measure Bim avg -(V(b)-Bavg)*sin(360*time*Freq)
.measure GainMag param 20*log10(hypot(Are,Aim) / hypot(Bre,Bim))
.measure GainPhi param mod(atan2(Aim, Are) - atan2(Bim, Bre)+180,360)-180
```

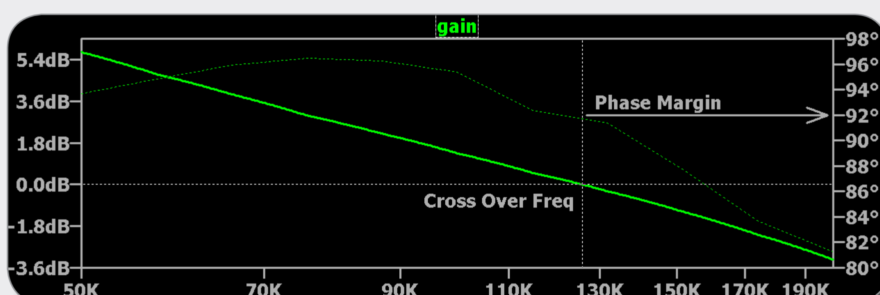
- Paste the following SPICE directive on the schematic. Parameter t0 is the length of time required for the system to settle to steady state and also sets when the simulator starts saving data. The difference between start and stop times in this case has been chosen as 25/freq so that the error from a non-integral number of switching cycles is small, since many cycles are included.

```
.param t0=.2m
.tran 0 {t0+25/freq} {t0}
```

- Insert a .step command to set the frequency range over which you want to perform the analysis. In this example, the simulation runs from 50kHz to 200kHz using five points per octave. Hint: Before stepping through the entire frequency range, test at a couple of frequencies (e.g., insert ".param Freq = 125K") and look at V(A) and V(B) to ensure you have sufficient amplitude in your voltage source, and if possible, tighten up the frequency range to minimize simulation time.

```
.step oct param freq 5K 500K 5
.save V(a) V(b)
.option plotwinsize=0 numdgt=15
```

- Run your simulation (see bottom left corner for status update).
- To view the Bode plot, open the SPICE Error Log (choose SPICE Error Log from the View menu) and right-click on the log to select "Plot .step'ed .meas data". Choose Visible Traces from the Plot Settings Menu. Select gain. From this plot you can then determine the crossover frequency and phase margin of your SMPS design.



Further examples and documentation can be found in the educational examples (..\LTspiceIV\examples\Educational\FRA\) and under the FAQ section of the Help Topics (press F1).

Happy simulations!

µModule Regulators

- **LTM4624:** 14V input, 4A step-down DC/DC µModule regulator www.linear.com/LTM4624
- **LTM4630:** Dual 18A or single 36A DC/DC µModule regulator www.linear.com/product/LTM4630
- **LTM4649:** 10A step-down DC/DC µModule regulator www.linear.com/LTM4649
- **LTM4676:** Dual 13A or single 26A µModule regulator with digital power system management www.linear.com/LTM4676
- **LTM8050:** 58V, 2A step-down µModule regulator www.linear.com/product/LTM8050

Linear Regulator

- **LT3007 Series:** 3µA I_Q, 20mA, 45V low dropout fault tolerant linear regulators www.linear.com/LT3007
- **LT3030 :** Dual 750mA/250mA low dropout, low noise, micropower linear regulator www.linear.com/LT3030
- **LT3081:** 1.5A single resistor rugged linear regulator with monitors www.linear.com/LT3081
- **LT3055:** 500mA, linear regulator with precision current limit and diagnostics www.linear.com/LT3055

Precision Amplifiers

- **LTC2057:** High voltage, low noise zero-drift operational amplifier www.linear.com/LTC2057

Ideal Diode

- **LT4320/-1:** Ideal diode bridge controller www.linear.com/LT4320