What's New with LTspice IV?

Gabino Alonso

—Follow @LTspice at *www.twitter.com/LTspice*

—Like us at *facebook.com/LTspice*

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 Article: "Loop Gain and its Effect on Analog Control Systems" by Simon Bramble

www.linear.com/solutions/5587

This article brings together the concepts of open loop gain, closed loop gain, gain and phase margin, and minimum gain stability, and shows how these parameters are interrelated in a feedback system. It examines loop gain in terms of a theoretical control system as well as practical electronic circuits, including linear regulators.

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.

SELECTED DEMO CIRCUITS

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

Linear Regulators

- **LT3063:** 1.8V Low noise regulator with output discharge (2.3V to 1.8V at 200mA) www.linear.com/solutions/5246

Buck Regulators

- **LT3667:** 40V step-down regulator with dual LDOs (6V–40V to 5V at 200mA, 2.5V/3.3V at 100mA) www.linear.com/solutions/5359
- **LT8609:** 5V, 2MHz, µPower step-down regulator (5.5V–40V to 5V at 2A) www.linear.com/LT8609
- **LT8640:** 5V 2MHz µPower ultralow EMI step-down converter (5.7V–42V to 5V at 5A) www.linear.com/solutions/5635
- **LTM4625:** 5A buck µModule regulator (4V–20V to 1.5V at 5A) www.linear.com/solutions/5613
- **LTM4630:** High efficiency single 36A step-down regulator (4.5V–15V to 1V at 36A) www.linear.com/solutions/5618

Buck-Boost Regulators

- **LT3790:** 240W high efficiency parallel buck-boost regulator (8V–6V to 12V at 10A) www.linear.com/solutions/5464
- **LTM8055:** High efficiency buck-boost regulator with accurate current limit & output current monitor (5V–36V to 12V at 6A) www.linear.com/solutions/5690
- **LTM8056:** High efficiency buck-boost regulator with accurate current limit & output current monitor (7V–58V to 24V at 3A) www.linear.com/solutions/5694

Isolated Converter

- **LTM8046:** 5V isolated flyback converter (3.2V–26V to 5V at 350mA) www.linear.com/solutions/5228

Constant Voltage, Constant Current Regulators

- **LT3081/LT8612/LTC3632:** 24V 3A constant voltage, constant current bench supply (10V–40V to 0-25V at 0A–3.1A) www.linear.com/solutions/5086
- **LT8705:** Bidirectional buck-boost supercapacitor backup supply (36V–80V to $15V_{CAP}$ at 1A) www.linear.com/solutions/1751

SELECT MODELS

To search the LTspice library for a particular device model, choose Component from the Edit menu or press F2.

Isolated Converter

- **LTM8057:** 3.1V–31V V_{IN} isolated µModule DC/DC converter www.linear.com/LTM8057

Supercap Charger

- **LTC3355:** 20V 1A buck DC/DC with integrated supercap charger and backup regulator www.linear.com/LTC3355

Linear Regulators

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

Buck Regulators

- **LT3697:** USB 5V, 2.5A output, 35V input buck with cable drop compensation www.linear.com/LT3697
- **LT8610AC:** 42V, 3.5A synchronous step-down regulator with 2.5µA quiescent current www.linear.com/LT8610AC
- **LT8613:** 42V, 6A synchronous step-down regulator with current sense and 3µA quiescent current www.linear.com/LT8613
- **LTM4623:** Ultrathin 20V V_{IN}, 3A step-down DC/DC µModule regulator www.linear.com/LTM4623

Boost/SEPIC/Inverting Regulator

- **LT8580:** Boost/SEPIC/inverting DC/DC converter with 1A, 65V switch, soft-start and synchronization www.linear.com/LT8580 ■

Power User Tip

SPEED UP YOUR SIMULATIONS

LTspice is designed from the ground up to produce fast circuit simulations, but there is margin in some simulations to increase the speed. Note, there may be trade-offs in accuracy using the methods described here. For further details on any of these approaches, please refer to the LTspice Help File (F1). To measure the effects of your changes, review the simulation time in the LTspice error log (Ctrl + L).

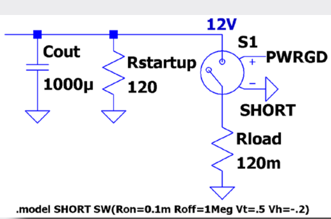
Reduce Power Supply Start-Up Time

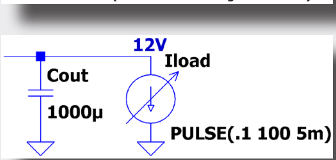
Reduce the time required for switch mode power supply (SMPS) simulation by shortening the voltage ramp of the output by changing the value of the soft-start capacitor. Before doing so, make sure you have a good understanding of the power supply's start-up performance. Then, reduce the soft-start capacitor value—using 0.001µF instead of the of the 0.1µF default—to quickly ramp to the desired output voltage.

Note that the soft-start capacitor should not be decreased to the point where the rising output allows the V_C/Ith pin to ramp well beyond its nominal control point and slew further down to stop overshoot.

Delay the Application of the Load to a Power Supply

Another effective technique to speed up simulation of an SMPS is to delay application of the load via a voltage controlled switch (SW). By using a switch that turns on the main load when the output voltage is near regulation (or at a known time), all the SMPS output energy goes into charging up the large output capacitors prior to the load being applied. A simpler approach can be achieved using a current load configured with a pulse function.





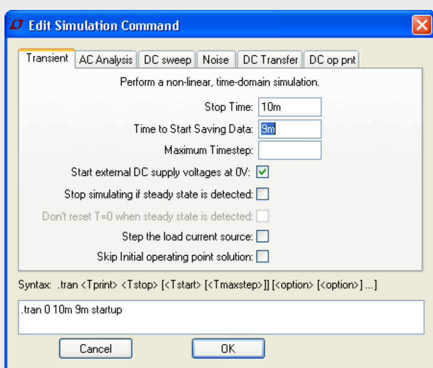
Set the Initial Conditions

Similarly, it might be effective to use the .ic spice directive to set initial conditions for selected nodes. For example, specify the initial voltage on the output so that it is close to regulation when the simulation starts. Likewise, you can specify the voltage at the compensation node to eliminate the initial droop at start-up.

```
.ic V(out)=11 V(vc)=1
```

Reduce the Amount of Transient Analysis Data

Normally, LTspice transient analysis starts at time = 0. You can edit the .trans simulation command's "Time to start saving data" to delay saving until a later time of interest, thus decreasing your overall simulation time. Of course this assumes you do not need the initial data points, which are not saved.

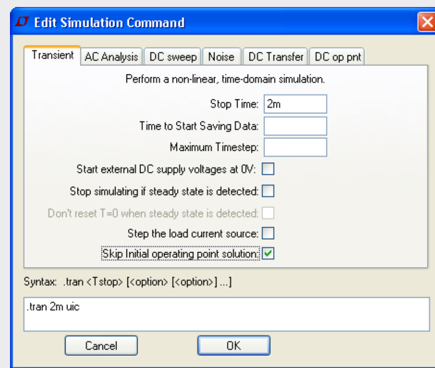


Alternately, if you are only interested in a few node voltages and device currents, you can restrict the quantity of saved data by using the .save directive to save only those specific node voltages and device currents. In the directive, add the "dialogbox" option to display all available nodes and currents so you can choose to save additional data of interest.

```
.save V(out) I(L1) V(in) dialogbox
```

Skip the Initial Operating Point Solution

Occasionally you will notice that a simulation stays in "Damped Pseudo-Transient Analysis" for a long period of time (see the lower left corner of the window for simulation status information). This usually occurs when a DC solution is sought in order to find the operating point of the circuit. If it is acceptable in your simulation, you can select Esc to skip finding the initial operating point and continue with the simulation. Likewise you can also "Skip initial operating point solution" by editing the simulation command.



If you prefer to save a difficult to solve DC operating point, you can use the .savebias command to save the preferred solution to a file in the initial simulation, and then in subsequent simulations, use the .loadbias command to quickly find the DC solution before proceeding with the rest of the simulation.

Use .savebias directive in initial simulation:

```
.savebias filename.txt internal time=10m
```

Used .loadbias in subsequent simulations:

```
.loadbias filename.txt
```

Convert to Fast Access Format When Viewing Waveforms

To maintain fast simulation speed, LTspice uses a compressed binary file format that allows additional simulation data to be quickly appended on the fly. However, once the simulation has completed, this format is non-optimal for waveform viewing. To speed up waveform plotting after the simulation is complete, convert the file to an alternate, "Fast Access," format. Click in the waveform window and choose Files > Convert to Fast Access. This can also be implemented using the .option fastaccess directive:

```
.option fastaccess
```

It is important to note that in some simulations this conversion may take longer than the actual simulation.

Happy simulations!