What's New with LTspice IV?

Gabino Alonso

twitter—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.

New Article: "Parallel MOSFETs in Hot Swap Circuits" by Dan Eddleman
www.linear.com/solutions/5677

While it is often desirable, and sometimes absolutely critical, to use multiple parallel MOSFETs in Hot Swap™ circuits, careful analysis of safe operating area (SOA) is essential. Each additional parallel MOSFET added to a circuit improves the voltage drop, power loss, and accompanying temperature rise of the application. But, the parallel MOSFETs do not necessarily improve the transient power capability of the circuit. Unless every MOSFET is driven by an independent control loop, temporary high power events such as initial turn-on into a load or current limiting into a short-circuit fault have a tendency to concentrate the power into a single MOSFET. That being said, it is safe to connect MOSFETs in parallel to reduce the overall resistance, using a single control loop as long as each MOSFET's SOA is capable of withstanding the entire transient event.

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

- **LT3086:** Adjustable voltage controlled current source www.linear.com/solutions/4475

Buck Regulators

- **LT8610AC:** 5V, 3.5A, 2MHz step-down converter (5.5V–42V to 5V at 3.5A) www.linear.com/solutions/5721
- **LTC3892:** High efficiency dual 3.3V/36V output step-down converter (7.5V–60V to 3.3V at 5.0A & 36V at 2A) www.linear.com/solutions/5668
- **LTM®4623:** Ultrathin 3A buck µModule® regulator (4V–20V to 1.5V at 3A) www.linear.com/solutions/5520

Boost Regulators

- **LT8580:** 1.5MHz, 5V to 12V boost converter (3.5V–6V to 12V at 200mA) www.linear.com/solutions/5236

Buck-Boost Regulators

- **LTC3111:** 15V, 800kHz wide input voltage buck-boost regulator (2.5V–15V to 5V at 1.5A) www.linear.com/solutions/4714
- **LTC3114-1:** Wide V_{IN} range regulator with bootstrapped LDO (2.7V–40V to 5V at 1A) www.linear.com/solutions/5084

SEPIC Converters

- **LT8495:** 450kHz, 5V output SEPIC converter (3V–60V to 5V at 1A) www.linear.com/solutions/5727

Multitopology Converters

- **LT8471:** Dual output buck & inverting converter (6V–32V to +5V at 1.4A & –5V at 800mA) www.linear.com/solutions/4676

Isolated Converters

- **LT3798/LT8309:** Energy Star compliant isolated converter (85V–150VAC to 5V at 2.2A) www.linear.com/solutions/5623

Surge Stoppers

- **LTC3810:** High efficiency switching surge stopper (36V–75V to 57Vclamp at 5A) www.linear.com/solutions/5639

Hot Swap Design

- **LTC4218:** 12V/100A Hot Swap design using parallel MOSFETs www.linear.com/solutions/5685

Filter Building Blocks

- **LT1568:** Multiple examples of bandpass, lowpass and highpass filters, and a sine wave converter www.linear.com/solutions/5740

SELECT MODELS

To search the LTspice library for a particular device model, choose Component from the Edit menu or 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. The changelog.txt file (see root installation directory) list provides a revision history of changes made to the program.

Buck Regulators

- **LTC3882:** Dual output PolyPhase® step-down DC/DC voltage mode controller with digital power system management www.linear.com/LTC3882

LED Drivers

- **LT3952:** 60V LED driver with 4A switch current www.linear.com/LT3952

Supercapcitor Chargers

- **LTC3128:** 3A monolithic buck-boost supercapacitor charger and balancer with accurate input current limit www.linear.com/LTC3128

Hot Swap Controllers

- **LTC4232-1:** 5A integrated Hot Swap controller (PCIe compliant) www.linear.com/LTC4232-1
- **LTC4234:** 20A guaranteed SOA Hot Swap controller www.linear.com/LTC4234

Op Amps

- **LTC6268-10/LTC6269-10:** Single/dual 500MHz ultralow bias current FET input op amp www.linear.com/LTC6268 ■

Power User Tip

SIMPLE IDEALIZED DIODE

LTspice semiconductor diode models are essential for simulations, especially when you want to see results that include breakdown behavior and recombination current. However, as complete as the semiconductor diode model is in LTspice, there are times when you need a simple "idealized diode" model to quickly simulate, for example, an active load, a current source or a current limiting diode. To assist, LTspice provides a representation of an idealized diode model.

To use of this idealized model in LTspice, insert a .model statement for a diode (D) with a unique name and define one or more of the following parameters: Ron, Roff, Vfwd, Vrev or Rrev.

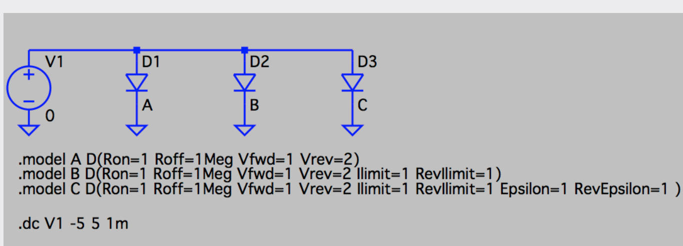
```
.model MyIdealDiode D(Ron=1 Roff=1Meg Vfwd=1 Vrev=2)
```

The idealized diode model in LTspice has three linear regions of conduction: on, off and reverse breakdown. The forward conduction and reverse breakdown can further be specified with current limit parameters Ilimit and revIlimit.

```
.model MyIdealDiode D(Ron=1 Roff=1Meg Vfwd=1
  Vrev=2 Ilimit=1 RevIlimit=1)
```

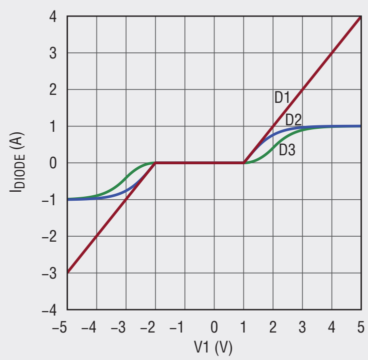
Furthermore, to smooth the switch between the off and conducting states the parameters epsilon and revepsilon can also be defined.

```
.model MyIdealDiode D(Ron=1 Roff=1Meg Vfwd=1 Vrev=2
  Ilimit=1 RevIlimit=1 Epsilon=1 RevEpsilon=1)
```

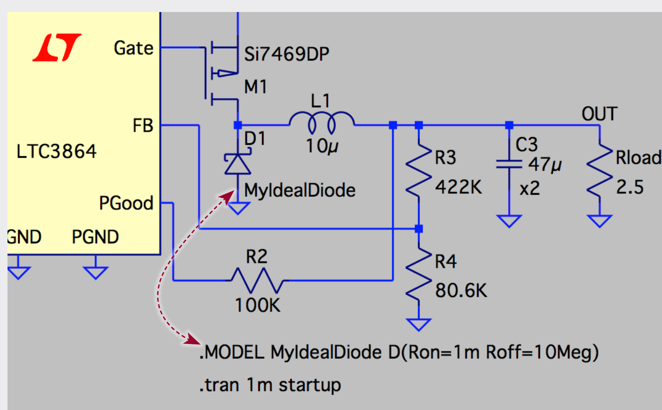


A quadratic function is also used between the off and on state such that the idealized diode IV curve is continuous in value and slope, so that the transition occurs over a voltage specified by the value of epsilon and revepsilon.

Once you have inserted your .model statement in your schematic you can edit the diode symbol's Value in the component attributes (Ctrl + Right Click) to match the name you specified in your statement. For more information on LTspice diode models, please refer to the help topics (F1).



Just for fun, in the circuit example below an idealized diode model is used to simulate a MOSFET's $R_{DS(ON)}$ in an otherwise nonsynchronous step-down controller. By using an idealized diode model instead of the traditional Schottky diode, the conduction losses of synchronous rectification can be easily compared.



Happy simulations!