What's New with LTspice IV?

Gabino Alonso

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

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

NEW VIDEO: "IMPORTING AND EXPORTING WAV FILES AND PWL TEXT FILES" by Simon Bramble

This video shows how to import and export WAV audio files to and from LTspice®, and how to read a list of piece-wise linear values from a text file. www.linear.com/solutions/6087

SELECTED DEMO CIRCUITS

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

Linear Regulators

- **LT3042:** Low noise, high PSRR RF linear regulator (3.8V–20V to 3.3V @ 200mA) www.linear.com/solutions/5638
- **LT3088:** Wide safe operating area linear regulator (1.2V–36V to 1.5V @ 800mA) www.linear.com/solutions/5817

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.

Buck Regulators

- **LT8631:** High voltage buck converter (6.5V–100V to 5V @ 1A) www.linear.com/solutions/5945
- **LT8709:** Negative buck regulator with output current monitor & power good (–16V to –30Vin to –12V @ 8.5A) www.linear.com/solutions/5600
- **LTM4630A:** High efficiency dual 18A buck with output tracking (6V–15V to 3.3 V & 5.0V @ 18A) www.linear.com/solutions/5782

Boost Regulators

- **LT8330:** 48V boost converter (10V–36V to 48V @ 135mA) www.linear.com/solutions/5947
- **LT8570:** Boost converter (5V–10V to 12V @ 125mA) www.linear.com/solutions/5667
- **LT8709:** Negative boost regulator with output current monitor & power good (–4.5V to –9V input to –12V @ 4.5A) www.linear.com/solutions/5596
- **LTC3121:** 5V to 12V synchronous boost converter with output disconnect (1.8V–5.5V to 12V @ 400mA) www.linear.com/solutions/5982

Inverting Regulators

- **LT8330:** Inverting converter (4V–36V to –12V @ 270mA) www.linear.com/solutions/5947
- **LT8709:** Negative inverting regulator with output current monitor & power good (–4.5V to –42V input to 5V @ 4A) www.linear.com/solutions/5598

Buck-Boost Regulator

- **LTM8054:** Buck-boost regulator with accurate current limit & output current monitor (6V–35V to 12V @ 3A) www.linear.com/solutions/5964

Surge Stopper

- **LTC7860:** High voltage surge stopper with timer (3.5V–60V to 3.5V–17V @ 5A) www.linear.com/solutions/5748

Amplifier

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

SELECT 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 Regulator

- **LTM4677:** Dual 18A or single 36A µModule regulator with digital power system management www.linear.com/LTM4677

Boost Regulator

- **LTC3121:** 15V, 1.5A synchronous step-up DC/DC converter with output disconnect www.linear.com/LTC3121

Multitopology Regulators

- **LT8331:** Low I_Q boost/SEPIC/ flyback/ inverting converter with 0.5A, 140V switch www.linear.com/LT8331
- **LT8714:** Bipolar output synchronous controller with seamless four quadrant operation www.linear.com/LT8714

- **LTC3899:** 60V low I_Q, triple output, buck/buck/boost synchronous controller www.linear.com/LTC3899

Hot Swap Controllers

- **LTC4233:** 10A guaranteed SOA hot swap controller www.linear.com/LTC4233

- **LTC4282:** High current hot swap controller with I²C compatible monitoring www.linear.com/LTC4282

LED Driver

- **LT3744:** High current synchronous step-down LED driver www.linear.com/LT3744

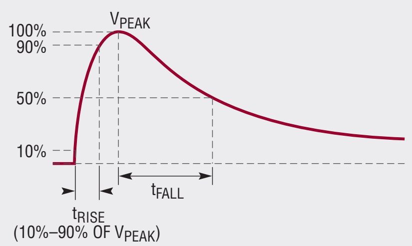
Amplifier

- **LTC6363:** Precision, low power rail-to-rail output differential op amp www.linear.com/LTC6363 ■

Power User Tip

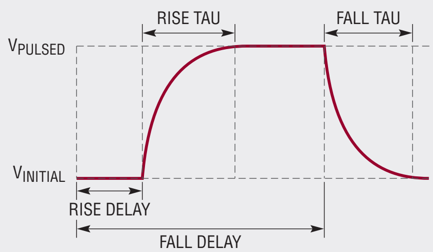
USING TIME-DEPENDENT EXPONENTIAL SOURCES TO MODEL TRANSIENTS

Occasionally there is a need to simulate a circuit's behavior with a specified voltage or current transient. These transients are usually modeled using a double exponential waveform characterized by a peak voltage, a rise time (usually 10%–90%), a fall time to 50% of the peak voltage and a series resistance.



Generalized exponential waveform

LTspice features a double exponential function (EXP) that is ideal for modeling transients via a voltage source. However, it is not as simple as filling in the parameter with t_{RISE}, t_{FALL} and V_{PEAK}. Instead, the EXP function uses standard parameters: Vinital, Vpulsed, Rise & Fall Delay and Raise & Fall Tau time constants.

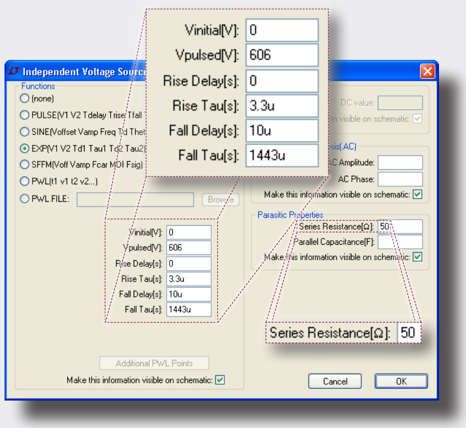


Exp voltage source parameters

For waveforms where t_{FALL}:t_{RISE} > 50:1 and t_{RISE} is defined from 10%–90%, you can use the following conversions for the EXP function parameters, and under the voltage source's parasitic properties, enter the appropriate series resistance or as a separate component:

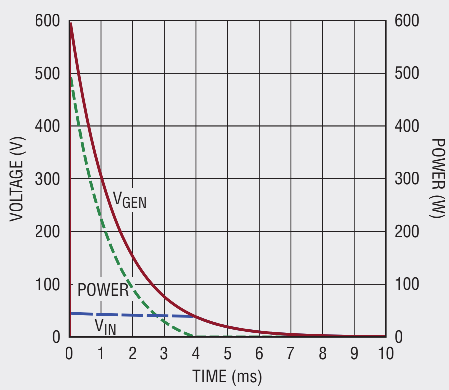
$V_{INITIAL}$ = V1
V_{PULSED} = V2 = V_{PEAK} • 1.01
Rise Delay = Td1 = (0 for no delay)
Rise Tau = Tau1 = t_{RISE}/2.2
Fall Delay = Td2 = t_{RISE}
Fall Tau = Tau2 = t_{FALL} • 1.443

Below is an example of a non-repetitive pulse waveform using EXP function with 10µs rise time, 1,000µs fall time, 600V peak and 50Ω series resistance.

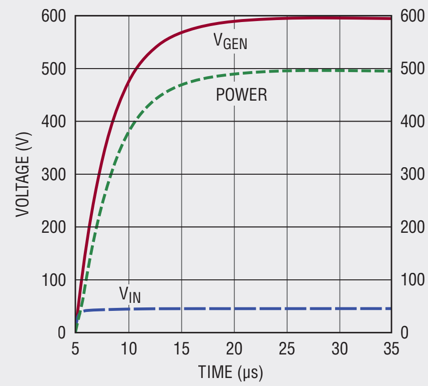


Sample EXP voltage source settings

The waveforms below show the results of the above EXP voltage source with an open circuit, V_{GEN}, and clamped with a TVS clamp, V_{IN}. Also shown is the instantaneous power dissipation (Alt + left-click) of the TVS.



Resulting waveform for an EXP voltage source



Detail of the EXP voltage source rise time

To simulate repeated bursts of transients as in Electrical Fast Transient, LTspice provides an extended syntax for the EXP function that is undocumented and not available in the standard component editor.

EXP(V1 V2 Td1 Tau1 Td2 Tau2 **Tpulse Npulse Tburst**)

Where Tpulse is the pulse period, Npulse is the number of pulses per burst and Tburst is the burst period. To add these to your exisitng EXP fuction, edit the EXP text string directly in your schematic by right-clicking it.

The following example shows an example of 75 transients at 200µs intervals which are repeated at 300ms intervals.

EXP(0 1.10 0 1.16n 1p 63.5n 200u 75 300m)

For waveforms where t_{FALL}:t_{RISE} < 50:1, implementing a rising and falling edge with a single EXP function is challenging. Instead, try using two voltage sources in series:

1. A piece wise linear (PWL) function for the rising edge where time1 = 0, value1 = 0, time2 = t_{RISE} (where t_{RISE} is 0%–100%), value2 = V_{PEAK}.
2. An EXP function for the falling edge where $V_{INITIAL}$ = 0, V_{PULSED} = –V_{PEAK}, Rise Delay = t_{RISE}, Rise Tau = (t_{FALL} – t_{RISE}) • 1.443 (falling edge of the waveform), Fall Delay = 1K (places the second exponential beyond the simulation time).

Happy simulations!