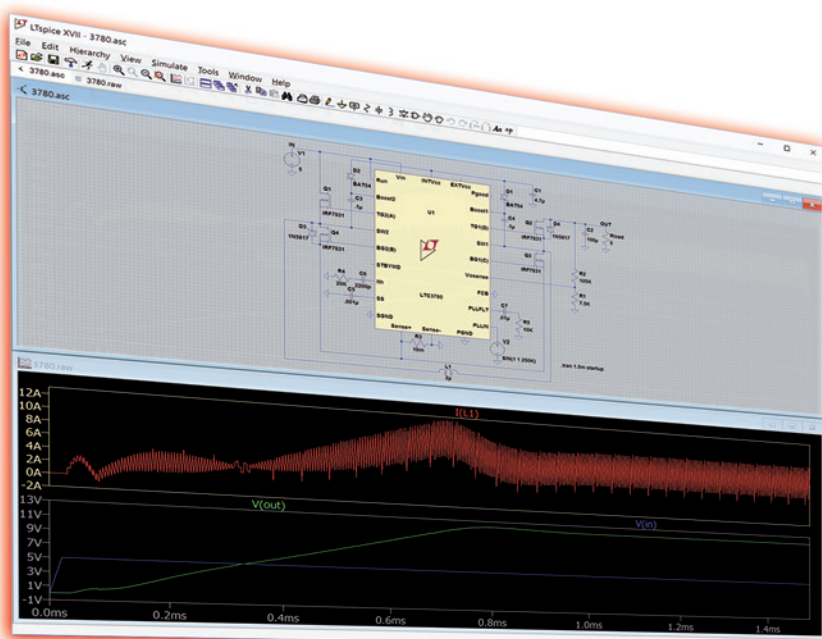


LTspice



- Freely Distributed
- Unlimited Nodes/Nets
- Fast Simulations

LTspice® is a high performance SPICE simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits. Our enhancements to SPICE have made simulating switching regulators extremely fast compared to normal SPICE simulators, allowing the user to view waveforms for most switching regulators in just a few minutes.



www.analog.com/LTspice

Included in the download of LTspice are macromodels for a majority of Analog Devices switching regulators and amplifiers, as well as a library of devices for general circuit simulation.



www.twitter.com/LTspice

Follow @ LTspice on Twitter for up-to-date information on models, demo circuits, events and user tips.

LTspice HotKeys			
	Schematic	Symbol	Netlist
Modes	ESC – Exit Mode	ESC – Exit Mode	
	F3 – Draw Wire		
	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate	
	F7 – Move	F7 – Move	
	F8 – Drag	F8 – Drag	
	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
View	Space – Zoom Fit		
	Ctrl+G – Toggle Grid		
	U – Mark Uncon. Pins	Ctrl+W – Attribute Window	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	
	Alt+Click – Power		
	Ctrl+Click – Attr. Edit		
	Ctrl+H – Halt Simulation		
	R – Resistor	R – Rectangle	
	C – Capacitor	C – Circle	
	L – Inductor	L – Line	
Place	D – Diode	A – Arc	
	G – GND		
	S – Spice Directive		
	T – Text	T – Text	
	F2 – Component		
	F4 – Label Net		
	Ctrl+E – Mirror	Ctrl+E – Mirror	
	Ctrl+R – Rotate	Ctrl+R – Rotate	

Simulator Directives – Dot Commands	
Command	Short Description
.AC	Perform a Small Signal AC Analysis
.BACKANNO	Annotate Subcircuit Pin Names on Port Currents
.DC	Perform a DC Source Sweep Analysis
.END	End of Netlist
.ENDS	End of Subcircuit Definition
.FOUR	Compute a Fourier Component
.FUNC	User Defined Functions
.FERRET	Download a File Given the URL
.GLOBAL	Declare Global Nodes
.IC	Set Initial Conditions
.INCLUDE	Include another File
.LIB	Include a Library
.LOADBIAS	Load a Previously Solved DC Solution
.MEASURE	Evaluate User-Defined Electrical Quantities
.MODEL	Define a SPICE Model
.NET	Compute Network Parameters in a .AC Analysis
.NODESET	Supply Hints for Initial DC Solution
.NOISE	Perform a Noise Analysis
.OP	Find the DC Operating Point
.OPTIONS	Set Simulator Options
.PARAM	User-Defined Parameters
.SAVE	Limit the Quantity of Saved Data
.SAVEBIAS	Save Operating Point to Disk
.STEP	Parameter Sweeps
.SUBCKT	Define a Subcircuit
.TEMP	Temperature Sweeps
.TF	Find the DC Small-Signal Transfer Function
.TRAN	Do a Nonlinear Transient Analysis
.WAVE	Write Selected Nodes to a .WAV file

Suffix	Suffix	Constants
	f	E 2.7182818284590452354
T 1e12	p 1e-12	Pi 3.14159265358979323846
G 1e9	n 1e-9	K 1.3806503e-23
Meg 1e6	u 1e-6	Q 1.602176462e-19
K 1e3	M 1e-3	TRUE 1
	Mil 25.4e-6	FALSE 0

©2018 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners. Ahead of What's Possible is a trademark of Analog Devices. LTspice-6/18(E)

analog.com



LTspice