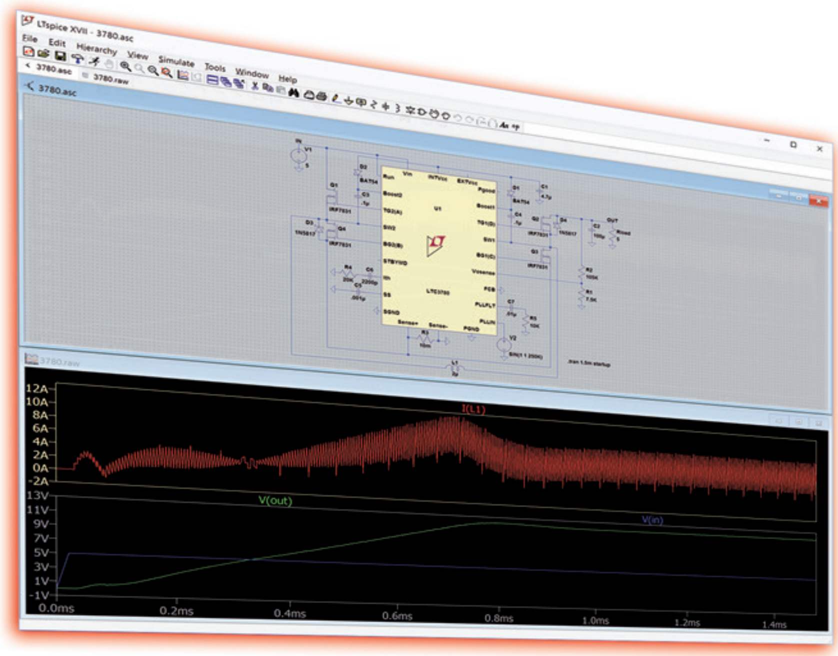
AHEAD OF WHAT'S POSSIBLE™

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.

FREE
LTspice
Download

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.

Visit *analog.com*

LTspice HotKeys

	Schematic	Symbol	Waveform	Netlist
Modes	ESC – Exit Mode	ESC – Exit Mode		
	F3 – Draw Wire			
	F5 – Delete	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate		
	F7 – Move	F7 – Move		
	F8 – Drag	F8 – Drag		
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
View	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
	Space – Zoom Fit		Ctrl+E – Zoom Extents	
	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #
	U – Mark Unncon. Pins	Ctrl+W – Attribute Window	‘0’ – Clear	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A – Add Trace	
	Atl+Click – Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation
	Ctrl+Click – Attr. Edit		Ctrl+Click - Average	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation
Place	**R – Resistor**	**R – Rectangle**		
	C – Capacitor	**C – Circle**		
	L – Inductor	**L – Line**		
	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**		

Command Line Switches

Flag	Short Description
-ascii	Use ASCII .raw files. (Degrades performance!)
-b	Run in batch mode.
-big or -max	Start as a maximized window
-encrypt	Encrypt a model library
-FastAccess	Convert a binary .raw file to Fast Access Format
-netlist	Convert a schematic to a netlist
-nowine	Prevent use of WINE(Linux) workarounds
-PCBnetlist	Convert a schematic to a PCB netlist
-registry	Store user preferences in the registry
-Run	Start simulating the schematic on open
-SOI	Allow MOSFET’s to have up to 7 nodes in subcircuit
-uninstall	Executes one step of the uninstallation process
-wine	Force use of WINE(Linux) workarounds

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	
		f	1e-15
T	1e12	p	1e-12
G	1e9	n	1e-9
Meg	1e6	u	1e-6
K	1e3	M	1e-3
		Mil	25.4e-6

Constants	
E	2.7182818284590452354
Pi	3.14159265358979323846
K	1.3806503e-23
Q	1.602176462e-19
TRUE	1
FALSE	0

LTspice-6/18(E)

analog.com

ANALOG DEVICES
AHEAD OF WHAT’S POSSIBLE™

LTspice