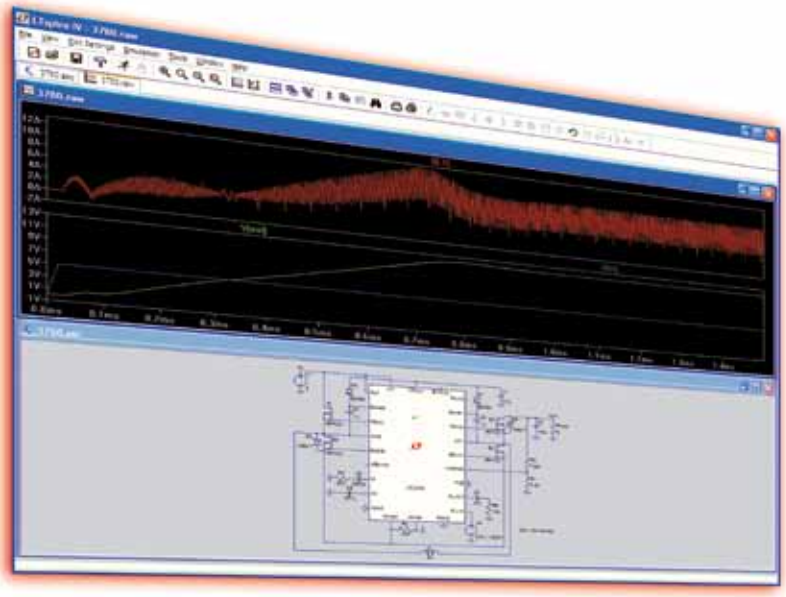
WÜRTH ELEKTRONIK

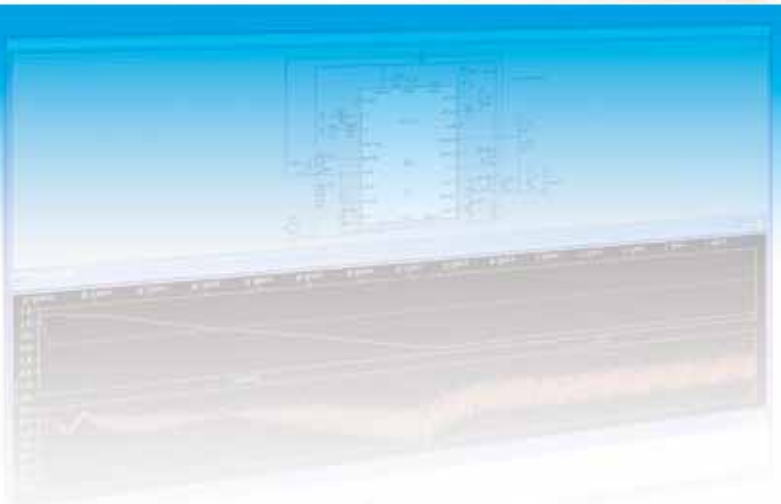
LINEAR TECHNOLOGY

LTspice

What you can do with the free LTspice simulation software:



- Do simulations of SMPS and other analog circuitry
- Use one of the 100s of demo circuits available on **www.linear.com/democircuits**
- Use a pre-drafted test fixture (JIG)
- Test models during development
- Get draft starting points
- Use the schematic editor to create your own design
- Add a macromodel & opening test fixture
- Get the latest datasheet
- Add sources, loads & additional circuit elements
- Run and edit simulations
- Run and probe a circuit using waveform viewer and test fixtures
- Plot plans, calculate frequencies and power dissipation
- Generate bill of materials and efficiency reports
- Benefit from a Built-in Help system and an independent **LTspice®** user´s group with many tutorials, libraries and examples



Linear Technology produces power management, data conversion, signal conditioning, RF and interface ICs, and µModule® subsystems across a large range of markets.

FREE LTspice® Download

www.linear.com/ltspice

Follow **LTspice** on Twitter:
www.twitter.com/ltspice

For e-mail news and updates sign up for **Linear Insider**:
www.linear.com/mylinear

Würth Elektronik component library for LTspice
Get all passive components of Würth Elektronik in one library.
Upload it in **LTspice** and start designing.

Check out more software tools on www.we-online.com/toolbox
The toolbox offers various design and select tools for passive and electromechanical components, such as Component Selector, Application Guide, Product Trainings, Parameter Search. etc.

www.linear.com **www.we-online.com**

WÜRTH ELEKTRONIK

LINEAR TECHNOLOGY

LTspice

LTspice Keyboard Shortcuts

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 the 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

LINEAR TECHNOLOGY

LTspice

www.we-online.com

www.linear.com