

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	'O' – 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	Command Line Switches	
	C – Capacitor	C – Circle		
	L – Inductor	L – Line	Flag	Short Description
	D – Diode	A – Arc	-ascii	Use ASCII .raw files. (Degrades performance!)
	G – GND		-b	Run in batch mode.
	S – Spice Directive		-big or -max	Start as a maximized window
	T – Text	T – Text	-encrypt	Encrypt a model library
	F2 – Component		-FastAccess	Convert a binary .raw file to Fast Access Format
	F4 – Label Net		-netlist	Convert a schematic to a netlist
	Ctrl+E – Mirror	Ctrl+E – Mirror	-nowine	Prevent use of WINE(Linux) workarounds
	Ctrl+R – Rotate	Ctrl+R – Rotate	-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		Constants	
		f	1e-15	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