

SCHEME AND WAVEFORM EDITING SHORTCUTS

Windows	Place Components*		Apple
[W]		wire	[F3]
[G]		ground	[G]
[Alt][G]		com	
[V]		voltage	[V]
[R]		resistor	[R]
[C]		capacitor	[C]
[L]		inductor	[L]
[D]		diode	[D]
[P]		component	[F2]
[N]		label net	[F4]
[T]		text/comment	[T]
[.]		spice directive right-click text field to open "Help me Edit" dialog	[S]
[B]		bus tap	[B]
[left-click]		toggle directive/comment	
*Press [Esc] or right-click to exit mode.			

Windows	Schematic Options		Apple
hold [Ctrl]		place angled wires	hold [⌘]
hold [Ctrl]		draw shapes off grid	hold [⌘]
[Ctrl][Alt][⌘][H]		show hidden text, e.g. parallel or series resistance	
[Ctrl][U]		show/hide unconn pin marks	
[Ctrl][A]		show/hide text anchor marks	
most options available in Settings			

Windows	Probe Schematic		Apple
click		Probe Wire plot voltage	click
		Probe Component plot current	
[Alt] click		Probe Wire plot current	[⌘] click
		Probe Component plot instantaneous power	
drag net-to-net		plot differential voltage	drag net-to-net
Probes available after simulation is run.			

Windows	Schemes, Waveforms, Symbols		Apple
[Ctrl][X] or [⌘] or [backspace]		delete	[F5]
[Ctrl][C]		copy/duplicate*	[F6]
[M]		move* select components to move	[F7]
[S]		stretch* select anchor points to move	[F8]
[Ctrl][R]		rotate	[⌘][R]
[Ctrl][E]		mirror	[⌘][E]
[Z]		Schematic zoom area (drag over area) zoom in (click on scheme) Waveform zoom area is default mode	Zoom in and out with scroll wheel or use pinch on track pad
[⌘][Z]		zoom out	
[Space]		zoom to fit. zoom extents	[Space]
[Ctrl][G]		toggle grid	
[Ctrl][Z]		undo	[F9] or [⌘][Z]
[Ctrl][⌘][Z]		redo	[⌘][F9] or [⌘][⌘][Z]
Choose mode first, then select component or waveform title. *Press [Esc] or right-click to exit mode.			

Edit Directives & Component Parameters			
right-click >			
	edit directive with help	edit limited parameters	
[Ctrl]	edit directive directly	edit all parameters	
Text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, "FAULT".			

Windows	Simulator		Apple
[A]		configure analysis	
[Alt][R]		run/pause	
[Alt][S]		stop	
[Ctrl][L]		view SPICE log	[⌘][L]
[O]		reset sim waveform T = 0	
Schematics can be edited even as a simulation runs. Edits affect subsequent simulations.			

Windows	Waveform Viewing		Apple
click or [C]		add cursor and see measure	click
[L]		label current cursor position	
[⌘][C] or [Esc]		clear all cursors	close measure dialog
[Alt] click		highlight corresponding net in schematic	[⌘] click
[Ctrl] click		integrate	[Ctrl] click
drag		move trace (to another pane)	drag
drag, hold [Ctrl]		copy trace (to another pane)	
[A]		add trace	
[P]		add pane above	
[B]		add pane below	
[U]		move active pane up	
[D]		move active pane down	
[D]		select steps	
		recenter	
Mouse actions are on waveform trace label.			

Waveform Pan & Cursor	
	No Cursors pan ~25%
	Cursor Present snap cursor to next time data point
	Cursor Present cycle cursors through traces at current time data point
	Cursor Present snap cursor to next data point No Cursors pan ~50%
[Ctrl] or [⌘] + [left] + [right]	Cursor Present bump cursor 10 data points
[Ctrl][⌘] + [left] + [right]	Cursor Present bump cursor 100 data points
[Ctrl]	pan with mouse
[Ctrl][⌘]	pan left and right with mouse
[Ctrl][Alt]	pan up and down with mouse
Click in waveform pane to apply keyboard functions to active frame.	

ANALOG DEVICES

LTspice[®] 24
 Fast • Free • Unlimited

SPICE QUICK REFERENCE

SPICE Analysis (requires exactly one*)



.ac	perform small signal AC analysis
.dc	perform DC source sweep analysis
.fra	perform a specialized transient simulation to analyze the frequency response of a feedback loop.
.noise	perform noise analysis
.op	find the DC operating point
.tf	find the DC small-signal transfer function
.tran	perform nonlinear transient analysis

* Simulation requires exactly one active spice analysis directive.
Tip: Open Configure Analysis to activate one directive and comment the others.

SPICE Directives

.backanno	annotate subcircuit pin names on port currents; automatically added by netlister
.end	end of netlist; required; added by netlister
.ends	end of subcircuit definition; use with .subckt
.four	compute fourier component
.func	user defined functions
.global	declare global nodes
.ic	set initial conditions
.include	include text from file
.lib	include library
.loadbias*	load a nodeset
.loadstate**	load a previously solved DC solution
.machine	arbitrary state machine
.measure	evaluate user-defined electrical quantities
.model	define a SPICE model
.net	compute network parameters in .AC analysis
.nodeset	supply hints for initial DC solution
.options	set simulator options
.param	user-defined parameters
.save	limit the quantity of saved data
.savebias*	save a nodeset to file
.savestate**	save comprehensive snapshot of state at time in a proprietary file format
.step	parameter sweeps
.subckt	define a subcircuit
.temp	temperature sweeps
.wave	write selected nodes to a .WAV file

* superseded by .savestate/.loadstate, **versions 24.1 and later

Spice Lines

Leading Character	Type of Line
*	comment
A	special function device
B	arbitrary behavioral source
C	capacitor
D	diode
E	voltage dependent voltage source
F	current dependent current source
G	voltage dependent current source
H	current dependent voltage source
I	independent current source
J	JFET transistor
K	mutual inductance
L	inductor
M	MOSFET transistor
O	lossy transmission line
Q	bipolar transistor
R	resistor
S	voltage controlled switch
T	lossless transmission line
U	uniform RC-line
V	independent voltage source
W	current controlled switch
X	subcircuit invocation
Z	MESFET or IGBT transistor
@	frequency response analyzer
&	frequency response analysis probe
.	simulation directive; for example: .options reltol=1e-4
+	continuation of the previous line



LTspice[®] 24
Fast • Free • Unlimited

NUMBERS

Constants

LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0

Used in waveform math

DRAWING

editor >						
t text						
↘ arrow						
/ line						
□ rectangle						
○ ellipse						
⌒ arc						

not all options available in all modes

Value Multipliers

LTspice	Means	Value
T or t	e12	tera 10^{12}
G or g	e9	giga 10^9
meg	e6	mega 10^6
K or k	e3	kilo 10^3
M or m	e-3	milli 10^{-3}
mil		mil 25.4×10^{-6}
U or u or μ	e-6	micro 10^{-6}
N or n	e-9	nano 10^{-9}
P or p	e-12	pico 10^{-12}
F or f	e-15	femto 10^{-15}

case insensitive
6K34 = 6.34K = 6.34k = 6.34e3
units not required, but allowed
kQ = kohm = K = k

COMMAND LINE FLAGS

-alt	set solver to Alternate
-ascii	use ASCII .raw files, degrading performance
-b <command>	batch mode of -run -netlist, or -sync, eg. ...-b -run
-big or -max	start LTspice as a maximized window
-ini <path>	use non-default .ini file
-l<path>	path to insert in the symbol and file search paths; no space after l (cap "i"); eg. -lC:\Users\...
-norm	set solver to Normal
-run	open the schematic and simulate
-encrypt	encrypt a model library
-FastAccess	convert a binary .raw file to Fast Access format
-FixUpSchematicFonts	convert the font size field of very old user-authored schematic/symbol text to the modern default
-FixUpSymbolFonts	convert the font size field of very old user-authored schematic/symbol text to the modern default
-netlist	batch conversion of a schematic to a netlist
-PCBnetlist	convert schematic to a PCB format netlist
-sync	update component libraries
-uninstall	uninstall LTspice

Syntax: LTspice.exe -l<path> <schematic.asc> -b -run -ini <path>
Path required for files not in same directory as LTspice.exe.
Can be stated as a full file path or defined using l<path>.