

SCHEME AND WAVEFORM EDITING SHORTCUTS

Place Components*		
[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.

Schematic Options		
hold [Ctrl]	place angled wires	hold [Alt]
hold [Ctrl]	draw shapes off grid	hold [Alt]
[Ctrl][Alt][H]	rsr←-1 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		

most options available in Settings

Probe Schematic		
click	Probe Wire plot voltage Probe Component plot current	click
[Alt] click	Probe Wire plot current Probe Component plot instantaneous power	[Alt] click
drag net-to-net	Probe Wire plot differential voltage	drag net-to-net

Probes available after simulation is run.

Schemes, Waveforms, Symbols		
[Ctrl][X] or [Delete] 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	[Alt][R]
[Ctrl][E]	mirror	[Alt][E]
[Z]	Schematic zoom area (drag over area) Waveform zoom area is default mode	Zoom in and out with scroll wheel or use pinch on track pad
[Space]	zoom out	
[Ctrl][G]	toggle grid	
[Ctrl][Z]	undo	[F9] or [Alt][F9]
[Ctrl][Shift][Z]	redo	[Alt][F9] or [Alt][Shift][Z]

Choose mode first, then select component or waveform title.

*Press [Esc] or right-click to exit mode.

Edit Directives & Component Parameters		
right-click >	.t	[C]
	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".		
Simulator		
[A]	configure analysis	
[Alt][R]	run/pause	
[Alt][S]	stop	
[Ctrl][L]	view SPICE log	[Alt][L]
[O]	reset sim waveform T = 0	

Schematics can be edited even as a simulation runs.
Edits affect subsequent simulations.

Waveform Viewing		
click or [C]	add cursor and see measure	click
[L]	label current cursor position	
[Alt][C] or [Esc]	clear all cursors	close measure dialog
[Alt] click	highlight corresponding net in schematic	[Alt] 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	
[Q]	recenter	

Mouse actions are on waveform trace label.

Waveform Pan & Cursor		
[Up]	No Cursors pan ~25%	
[Left][Right]	Cursor Present snap cursor to next time data point	
[Up][Down]	Cursor Present cycle cursors through traces at current time data point	
[Home] + [Left][Right]	Cursor Present snap cursor to next data point No Cursors pan ~50%	
[Ctrl] or [Home] + [Left][Right]	Cursor Present bump cursor 10 data points	
[Ctrl][Home] + [Left][Right]	Cursor Present bump cursor 100 data points	
[Ctrl]	pan with mouse	
[Ctrl][Home]	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
or
A

.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

* superceded 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						
arrow						
line						
rectangle						
ellipse						
arc						

not all options available in all modes

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 "l"); 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>.