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	net	label net	F4
T	t	text/comment	T
.	.t	spice directive *right-click text field to open "Help me Edit" dialog*	S
B		bus tap	B
⇧ left-click	t .t	toggle directive/comment	

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

Schematic Options

hold Ctrl		place angled wires	hold ⇧
hold Ctrl	○	draw shapes off grid	hold ⇧
Ctrl Alt ⇧ H	rser=1	show hidden text, e.g. parallel or series resistance	
Ctrl U	□IN	show/hide unconn pin marks	
Ctrl A	○V1	show/hide text anchor marks	

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	⌘ click
drag net-to-net	plot differential voltage	drag net-to-net

Probes available after simulation is run.

Schemes, Waveforms, Symbols

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 >	.t	
	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, "$\overline{\text{FAULT}}$".

Simulator

A	configure analysis	
Alt R	run/pause	
Alt S	stop	
Ctrl L	view SPICE log	⌘ L
0	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	
⇧ 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 ⇧ + ← →	**Cursor Present** bump cursor 10 data points
Ctrl ⇧ + ← →	**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 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 anlalysis 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
*	t	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	t	subcircuit invocation
Z		MESFET or IGBT transistor
@		frequency response analyzer
&		frequency response analysis probe
.	t	simulation directive; for example: .options reltol=1e-4
+	↲	continuation of the previous line

ANALOG DEVICES | 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

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
kΩ = kohm = K = k

DRAWING

	editor >			
t	text	T	T	T
↘	arrow			
╲	line		I	L
□	rectangle		R	R
○	ellipse			C
◠	arc			A

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
-I<path>	path to insert in the symbol and file search paths; no space after I (cap "i"); eg. -IC:\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 -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 -I<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 I<path>.

SIMULATION COMMANDS

Key

Description	Key	Replace Key with...	Examples
required literal	foo	foo	Replace V(<node>) with V(out) or V(n001)
<required value>	<foo>	a value	Replace <freq> with 1000 for freq = 1kHz Replace <Tstop> with 2 for stop time = 2s
[optional literal]	[foo]	foo, or leave out	Replace [startup] with startup
[<optional value>]	[<foo>]	a value, or leave out	Replace [I(<source>)] with I(R1)
<list, of, values>	<foo, bar>	foo or bar	Replace <oct, dec, lin> with oct
[<optional, mutually exclusive, list, of, values>]	[<foo, bar>]	foo or bar, or leave out to for default value, bar	
<filename>	<filename>	filename or path to save/retrieve a file	if no path is specified, the file is saved in the same directory as the netlist or schematic
[...][more]	[...]	optionally more versions of preceding item(s)	Replace V(<node>)[...] with V(n001) V(n002) V(in) V(out)
[parameter=<value>]	[foo=<bar>]	foo=value	Replace [Tstart=<val>] with Tstart=1ms
I(<source>)			<source> can be discrete component, Vsource name or pin. I(R1), I(V1), Ib(Q1)

parentheses () are always literal

SPICE Analysis (requires exactly one*)

.ac or A	.ac	perform small signal AC analysis	.ac <oct, dec, lin> <Npoints> <startfreq> <endfreq> .ac <list> <freq> [...]
	.dc	perform DC source sweep analysis for up to three V or I sources; overrides named source settings with DC sweep of source	.dc <sourcename> <oct, dec, lin> <startvalue> <stopvalue> <incr> [more sources] .dc <sourcename> list <value> [...][more sources]
	.fra	perform a transient simulation to analyze the frequency response of a feedback loop land produce a Bode plot	.fra [Tstart=<time>][dTmax=<time>][Tstep=<time>] + [Tstop=<time>][uic][startup]
	.noise	perform noise analysis	.noise V(<node>[,<refnode>]) <src> + <oct, dec, lin> <Npoints > <startfreq> <endfreq> .noise V(<node>[,<refnode>]) <src> list <freq> [...]
	.op	find the DC operating point	.op
	.tf	find the DC small-signal transfer function	.tf V(<node> [, <refnode>]) <source> .tf I(<Vsource>) <source>
	.tran	perform nonlinear transient analysis	.tran <Tstep> <Tstop> + [Tstart [dTmax]] + [uic]** [steady][nodiscard][startup][step] .tran <Tstop> + [uic]** [steady][nodiscard][startup][step]

** Simulation requires exactly one active spice anlalysis directive.*
Tip: Opening Configure Analysis comments all but one anlalysis command.
*** Use of this modifier is highly discouraged. In particular, it is not a viable workaround for DC operating point convergence problems.*

SPICE Directives

.backanno	annotate subcircuit pin names on port currents; automatically added by netlister	.backanno
.end	end of netlist; required; added by netlister	.end
.ends	end of subcircuit definition	.ends
.four	compute fourier component	.four <frequency> [Nharmonics][Nperiods] + <datatrace> [...]
.func	user defined functions	.func <name> ([arguments]) {<expression>}
.global	declare global nodes	.global <node> [...]
.ic	set initial conditions	.ic [V(<node>)=<voltage>][...] + [I(<inductor>)=<current>][...]
.include	include text from file	.include <filename.ext>
.lib	include library	.lib '<filename>' <entryname> .lib "<filename>" <entryname>
.loadbias*	load a nodeset from a file	.loadbias <filename>
.loadstate**	load a previously solved DC solution	.loadstate <statefilename> [reset]
.machine	arbitrary state machine	.mach[ine][<tripdt>] .state <name> <value> .rule <old state> <new state> <condition> .output (<posnode> [, <negnode>]) <expression> .endmach[ine]
.measure	evaluate user-defined electrical quantities at a point on the abscissa or over a range	.meas[sure][ac, dc, op, tran, tf, noise] <name> + [<find, deriv, param> <expr> [when <expr>, at=<expr>]] + [td=<val1>][<rise, fall, cross>=[<count1>, last]]
.model	define a SPICE model	.model <name> <type> [(<parameter list>)]
.net	compute network parameters in .AC analysis	.net [V(out[,ref]), I(Rout)] < Vin, lin> +[Rin=<val>][Rout=<val>]
.nodeset	supply hints for initial DC solution	.nodeset V(<node>)=<voltage> [...]
.options	set simulator options	.options
.param	user-defined parameters	.param
.save	limit the quantity of saved data	.save V(<node>)[...][V(n2)[I(L1)[I(S2)]]]
.savebias*	save a nodeset to file	.savebias <filename> [internal] + [temp=<temp>][time=<time> [repeat]] + [step=<step#>] + [DC1=<value>][DC2=<value>][DC3=<value>]
.savestate**	save comprehensive snapshot of state	.savestate <filename> [time=<time>]
.step	parameter sweeps	.step [<oct, dec, lin>] <item> <startval> <endval> <incr> .step <item> list <value> [...]
.subckt	define a subcircuit	.subckt <name> [<node>][...]
.temp	temperature sweep; same as .step temp list	.temp <temp> [...]
.wave	write selected nodes to a .WAV file	.wave <filename.wav> <Nbits> <SampleRate> + V(<node>)[...][I(<source>)][...]

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

ANALOG DEVICES | LTspice® 24
Fast • Free • Unlimited