# SCHEMATIC AND WAVEFORM EDITING SHORTCUTS

## Place Components*

| Windows | | Mac |
|---|---|---|
| W | wire | W |
| G | ground | G |
| Alt G | com | ⌥ G |
| V | *voltage* | V |
| R | resistor | R |
| C | capacitor | C |
| L | inductor | L |
| D | diode | D |
| P | component | P |
| N | label net | N |
| T | text/comment | T |
| . | spice directive<br>*right-click text field to open "Help me Edit" dialog* | . |
| B | bus tap | B |
| ⇧ left-click | toggle directive/comment | ⇧ left-click |

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

## Schematic Options

| Windows | | Mac |
|---|---|---|
| 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 | ⌘ ⌥ ⇧ H |
| Ctrl U | show/hide unconn pin marks | ⌘ U |
| Ctrl A | show/hide text anchor marks | ⌘ A |

*most options available in Settings*

## Probe Schematic*

| Windows | | Mac |
|---|---|---|
| click | **Probe Wire** plot voltage<br>**Probe Component** plot current | click |
| Alt click | **Probe Wire** plot current<br>**Probe Component** plot instantaneous power | ⌥ click |
| drag net-to-net | plot differential voltage | drag net-to-net |

**Probes available once simulation is running.*

## General Editing

| Windows | | Mac |
|---|---|---|
| Ctrl X or ⌦ or backspace | delete | ⌘ X or ⌦ or backspace |
| Ctrl C | copy/duplicate* | ⌘ C |
| M | move*<br>*select components to move* | M |
| S | stretch*<br>*select anchor points to move* | S |
| Ctrl R | rotate | ⌘ R |
| Ctrl E | mirror | ⌘ E |
| Z | **Schematic**<br>zoom area (drag over area)<br>zoom in (click on scheme)<br>**Waveform**<br>zoom area is default mode | Z |
| ⇧ Z | zoom out | ⇧ Z |
| Space | zoom to fit. zoom extents | Space |
| Ctrl G | toggle grid | ⌘ G |
| Ctrl Z | undo | ⌘ Z |
| Ctrl ⇧ Z | redo | ⌘ ⇧ Z |

*Choose mode first, then select component or waveform title.*
**Press Esc or right-click to exit mode.*

## Edit Directives & Component Parameters

| right-click > | **text** | **component body** |
|---|---|---|
| | 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

| Windows | | Mac |
|---|---|---|
| A | configure analysis | A |
| Alt R | run/pause | ⌥ R |
| Alt S | stop | ⌥ S |
| Ctrl L | view SPICE log | ⌘ L |
| 0 | reset sim waveform T = 0 | 0 |

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

## Waveform Viewing

| Windows | | Mac |
|---|---|---|
| click or C | add cursor and see measure | click |
| L | label current cursor position | L |
| ⇧ C or Esc | clear all cursors | ⇧ C or Esc |
| Alt click | highlight corresponding net in schematic | ⌘ click |
| Ctrl click | integrate | ⌘ click |
| drag | move trace (to another pane) | drag |
| drag, hold Ctrl | copy trace (to another pane) | drag, hold ⌘ |
| A | add trace | A |
| P | add pane above | P |
| B | add pane below | B |
| U | move active pane up | U |
| D | move active pane down | D |
| ⇧ S | select steps | ⇧ S |
| | recenter | |

*Mouse actions are on label of waveform trace*

## Waveform Pan & Cursor

| Windows | | Mac |
|---|---|---|
| ↑ ← ↓ → | **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<br>**No Cursors** pan ~50% | ⇧ + ← → |
| Ctrl or ⇧ + ← → | **Cursor Present** bump cursor 10 data points | ⌘ or ⇧ + ← → |
| 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 | ⌘ Alt |

*Click in waveform pane to apply keyboard functions to active frame.*

# LTspice® 26



ANALOG DEVICES

# 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 |

or

** 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 Netlist Lines/Components

| *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 |
| NA | jumper, schematic only, not netlisted |

# LTspice® 26

# 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>.*

# NUMBERS

## 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 | $25.4\times10^{-6}$ |
| 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*

## 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 >* | | | | |
|---|---|---|---|---|
| | text | T | T | T |
| | line | | I | L |
| | rectangle | | R | R |
| | ellipse | | | C |
| | arc | | | A |

*draw arrow available for waveform*


LTspice-2/26(A) analog.com

# 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<br>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) |
| <required, mutually exclusive, list, of, values> | <foo, bar> | foo or bar | Replace <oct, dec, lin> with oct or leave unspecified for lin |
| [<optional, mutually exclusive, list, of, values>] | [<foo, bar>] | foo or bar, or leave out for default value, bar | Replace [<oct, dec, lin>] with dec |
| '<filename>'<br>"<filename>" | <filename> | filename or path to save/retrieve a file | Replace filename with "my path\my file". If no path is specified, the file is saved in the same directory as the netlist or schematic. Enclose the file name or path in single or double quotes. |
| [...][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><br>.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]<br>.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>]<br>+ [Tstop=<time>] [uic] [startup] |
| | .noise | perform noise analysis | .noise V(<node>[,<refnode>]) <src><br>+ <oct, dec, lin> <Npoints > <startfreq> <endfreq><br>.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><br>.tf I(<Vsource>) <source> |
| | .tran | perform nonlinear transient analysis | .tran <Tstep> <Tstop><br>+ [Tstart [dTmax]]<br>+ [uic]** [steady] [nodiscard] [startup] [step]<br>+ [loadstate=[<filename>]]<br>+ [savestate=[<filename>]] [savestatetime=[<time>]]<br>.tran <Tstop><br>+ [uic]** [steady] [nodiscard] [startup] [step]<br>+ [loadstate=[<filename>]]<br>+ [savestate=[<filename>]] [savestatetime=[<time>]] |

** Simulation requires exactly one active spice anlalysis directive.*
*Tip: Opening Configure Analysis comments out 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 (used with .subckt) |
| .four | compute fourier component | .four <freq> [Nharmonics] [Nperiods]<br>+ <datatrace> [...] |
| .func | user defined functions | .func <name> ([arguments]) {<expression>} |
| .global | declare global nodes | .global <node> [...] |
| .ic | set initial conditions | .ic [V(<node>)=<voltage>] [...]<br>+ [I(<inductor>)=<current>] [...] |
| .include | include text from file | .include <filename.ext> |
| .lib | include library | .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>]<br>.state <name> <value><br>.rule <old state> <new state> <condition><br>.output (<posnode> [, <negnode>]) <expression><br>.endmach[ine] |
| .measure | evaluate user-defined electrical quantities at a point on the abscissa or over a range | .meas[ure] [ac, dc, op, tran, tf, noise] <name><br>+ [<find, deriv, param> <expr> [when <expr>, at=<expr>]]<br>+ [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><br>+[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]<br>+ [temp=<temp>] [time=<time> [repeat]]<br>+ [step=<step#>]<br>+ [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><br>.step <item> list <value> [...] |
| .subckt | begin a subcircuit definition | .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><br>+ V(<node>) [...] [I(<source>)] [...] |

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


LTspice-2/26(A) analog.com

LTspice® 26

# .OPTIONS FLAGS/MODIFIERS AND PARAMETERS

| Keyword | Type | Default | Description |
|---|---|---|---|
| abstol | = | 1pA | Absolute current error tolerance |
| allow_ambiguous_models | flag | | Each model used in simulation can have exactly one definition. This option tells LTspice to ignore the error, choose a model with no indication of which, and continue simulating. |
| baudrate | = | (none) | Used for eye diagrams. Tells the waveform viewer how to wrap the abscissa time to overlay the bit transitions. |
| chgtol | = | 10fC | Absolute charge tolerance |
| cshunt | = | 0 | Optional capacitance added from every node to ground |
| cshuntintern | = | cshunt | Optional capacitance added from every device internal node to ground |
| debugtran | flag | | Add device and node convergence difficulty scores to the log file. Slows down the simulation. Results are relative, but generally, values below 1 are negligible, while values above ~50 are problematic. |
| defad | = | 0 | Default MOS drain diffusion area |
| defas | = | 0 | Default MOS source diffusion area |
| defl | = | 100µm | Default MOS channel length |
| defw | = | 100µm | Default MOS channel width |
| delay | = | 0 | Used for eye diagrams. Shifts the bit transitions in the diagram. |
| fastaccess | flag | | Convert to fastaccess file format at end of simulation |
| flagloads | flag | | Flags external current sources as loads |
| gfarad | = | 1e-12 | Conductivity leakage per farad of every capacitor element |
| gfloat | = | 1e-12 | Conductivity added to from every floating node to ground if topology checking is enabled |
| gmin | = | 1e-12 | Conductivity added to every PN junction to aid convergence |
| gminsteps | = | 25 | Set to zero to prevent gminstepping for the initial DC solution |
| gshunt | = | 0 | Optional conductance added from every node to ground |
| itl1 | = | 100 | DC iteration count limit |
| itl2 | = | 50 | DC transfer curve iteration count limi |
| itl4 | = | 10 | Transient analysis time point iteration count limit |
| itl6 | = | 25 | Set to zero to prevent source stepping for the initial DC solution |
| srcsteps | = | 25 | Alternative name for itl6 |
| list | flag | | Put expanded netlist into log file. This has priority over the corresponding choice in the Settings window. |
| logparams | flag | | Puts all parameters into the log file. Useful as a diagnostic. |
| logopinfo | flag | | Puts operating point information of semiconductor devices into log file |
| maxclocks | = | ∞ | maximum number of clock cycles to save |
| maxstep | = | ∞ | Maximum step size for transient analysis |
| measplxfmt | = | bode | Complex number format of .meas statement results. One of "polar", "cartesian", or "bode". |
| measdgt | = | 6 | Number of significant figures used for .measure statement output |
| method | = | trap | Numerical integration method, either "trap", "modtrap", or "gear". This has priority over the corresponding choice in the Settings window. |
| minclocks | = | 10 | Minimum number of clock cycles to save |
| mindeltagmin | = | 1e-4 | Limit for termination of adaptive gmin stepping |

| Keyword | Type | Default | Description |
|---|---|---|---|
| no_caret_warning | flag | | Suppresses all caret operator warnings. |
| nomarch | flag | | Do not plot marching waveforms |
| noopiter | flag | | Go directly to gmin stepping |
| numdgt | = | 6 | Historically "numdgt" was used to set the number of significant figures used for output data. In LTspice, if "numdgt" is set to be > 6, double precision is used for dependent variable data. |
| pivrel | = | .001 | Relative ratio between the largest column entry and an acceptable pivot value |
| pivtol | = | 1e-13 | Absolute minimum value for a matrix entry to be accepted as a pivot |
| plotreltol | = | 0.0025 | Relative error tolerance for waveform compression |
| plotvntol | = | 10µV | Absolute voltage error tolerance for waveform compression |
| plotabstol | = | 1nA | Absolute current error tolerance for waveform compression |
| plotwinsize | = | 300 | Number of data points in each compression window. With compression, each compression window is reduced to fitted polynomials. Smaller numbers mean less compression. Set to zero to disable compression. |
| ptrantau | = | 0.1 | Characteristic source start-up time for a damped pseudo transient analysis to find the operating point. Set to zero to disable pseudo transient. |
| ptranmax | = | 0 | If set non-zero, that time of the damped pseudo transient analysis is used as the operating point whether the circuit has settled or not |
| reject_number_tails | flag | | Report any numeric literal with additional text following the number as syntax error |
| reltol | = | .001 | Relative error tolerance |
| solver | = | (none) | Solver to use. Values are either "norm" or "alt". This setting overrides solver choice in Settings, and/or "-alt" and "-norm" command line flags. |
| srcstepmethod | = | 0 | Which source stepping algorithm to start with |
| sstol | = | .001 | Relative error for steady-state detection |
| ststclocks | = | 10 | Number of clock cycles to process in continuous switching mode after steady state detection |
| ststdelay | = | 0 | Time to wait before trying to detect steady state |
| temp | = | 27°C | Default temperature for circuit element instances that don't specify temperature |
| tnom | = | 27°C | Default temperature at which device parameters were measured for models that don't specify this temperature |
| topologycheck | = | 1 | Set to zero to skip check for floating nodes, loops of voltage sources, and non-physical transformer winding topology |
| trtol | = | 2.0 | Transient error tolerance. This parameter is an estimate of the factor by which the actual truncation error is overestimated |
| trytocompact | = | 1 | When non-zero, the simulator tries to condense LTRA transmission lines' history of input voltages and currents |
| vntol | = | 1µV | Absolute voltage error tolerance |

*Flags have no value; either the flag is present or it is not. Syntax is .options [ flag ][ ... ][ <keyword> = <value> ][ ... ]*


LTspice-2/26(A) analog.com

LTspice® 26