



Cheatsheet: LTspice For Mac

General	2
General Information	2
Resources	2
Files	2
Main Menu	3
Schematic Capture Window	4
General Information	4
Toolbar	4
Contextual Menu	4
Object Actions	6
Probing	6
Adding Component Models And Subcircuits	6
Other Operations	7
Waveform Viewer Window	8
General Information	8
Toolbar	8
Contextual Menu	8
Object Actions	9
Other Operations	9
Revisions	10

General

General Information

- LTspice is a high performance SPICE simulator, schematic capture, and waveform viewer with enhancements and models that makes it easier to perform analog circuit simulations. It is known as the most widely used free SPICE simulator in the industry.
- LTspice was originally developed by Linear Technology, hence the LT in its name, which was later acquired by [Analog Devices](#) who continues to maintain it.
- DC operating point data will only be included in the SPICE log when the `.op` SPICE directive is used by itself.
- The SPICE engine solver can be changed from *Normal* (default) to *Alternate* for more accurate simulations, but they will take longer to run. This option can be changed within the SPICE Engine section of the SPICE tab in the Control Panel.
- This cheatsheet covers the most commonly used commands and operations as of version 17.0.21.0 (12/15/2020).

Resources

- [LTspice By Analog Devices](#)
- [LTspice Shortcuts On A Mac](#) (pdf)
- [LTspice Wiki](#)
- [LTspice On Wikipedia](#)
- [Getting Started With LTspice For Mac By Woolsey Workshop](#)

Files

- Locations
 - Application: /Applications/LTspice.app
 - Projects: ~/Documents/LTspice
 - Demo (Example) Circuits: ~/Documents/LTspice/examples
 - Copied from /Applications/LTspice.app/Contents/Resources/examples upon initial application invocation.
 - Libraries: ~/Library/Application\ Support/LTspice/lib
- Types
 - Schematic (.asc)
 - Symbol (.asy)
 - Bill Of Materials (.bom)
 - Efficiency Report (.eff)
 - Fast Fourier Transform Analysis (.fft)
 - SPICE Log (.log)
 - SPICE Netlist (.net, .cir, .sp)

- Plot Configuration (.plt)
- Raw Waveform Data (.raw)
- Subcircuit (.sub)
- While .net, .log, and .raw files are created during use, they are deleted when LTspice exits, leaving only the .asc schematic and .plt files.
- Files with extensions of .net, .cir, or .sp are recognized by LTspice as SPICE netlists and can be loaded and simulated separately without the need for an associated schematic.

Main Menu

- **Menu > LTspice > About LTspice** - Display the About window.
- **Menu > LTspice > Quit LTspice (⌘Q)** - Exit the application.
- **Menu > File > New > New Schematic (⌘N)** - Create a new schematic.
- **Menu > File > Open... (⌘O)** - Open an existing file.
- **Menu > File > Open Recent** - Open a recently opened file.
- **Menu > File > Close (⌘W)** - Close the active window.
- **Menu > File > Save (⌘S)** - Save the file in the active window.
- **Menu > File > Save As** - Save the file in the active window with a new name.
- **Menu > File > Print... (⌘P)** - Print the file in the active window.
- **Menu > Edit > Undo (⌘Z)** - Undo the last operation.
- **Menu > Edit > Redo (⇧⌘Z)** - Redo the last Undo operation.
- **Menu > Help > LTspice Help** - View the help page.

Schematic Capture Window

General Information

- The schematic capture window is where you visually build your circuit and add your SPICE directives for simulation.
- Schematic editing uses a "verb-noun" interface instead of "noun-verb". That means you first select the action (move, drag, copy, delete, etc.) and then the object (component, label, wire, etc.).
- All object placement operations (drafting, moving, deleting, etc.) continue to be performed, allowing for subsequent placements, until a right-click is performed or the **ESC** key is pressed to end the operation.
- Label orientations are based on how they are connected to wires or components and are not user adjustable.
- Reference designators (e.g. R2, Q1, D5, etc.) are assigned during placement.
- Devices may need to be rotated to show the preferred current direction. Although not required, this avoids negative currents being shown in the SPICE log for operating point simulations.

Toolbar

- **Run!** Button - Starts a circuit simulation.
- **Stop!** Button - Halts the current simulation.
- **Control Panel** Button - Opens the Control Panel where LTspice preferences can be edited.
- **Search** Field - Searches for items in the schematic.
- Available object actions are displayed in the middle of the toolbar when hovering over an object.

Contextual Menu

- View the contextual menu by right-clicking, or control-clicking, in an empty area of the schematic editor.
- **Menu > Run** - Starts a circuit simulation.
- **Menu > Halt** - Halts the current simulation.
- **Menu > View > Zoom to Fit (Space)** - Zoom to fit the schematic within the window.
- **Menu > View > Grid Dots** - Toggle the display of grid dots in the schematic editor.
- **Menu > View > Waveform Data** - View the raw data (.raw), i.e. simulation traces, in the waveform viewer.
- **Menu > View > SPICE Error Log (⌘L)** - View the SPICE log (.log) in a separate window.
- **Menu > View > SPICE Netlist** - View the SPICE netlist (.net) in a separate window.

- **Menu > Edit > Move (F7)** - Move one or more objects in the schematic, without keeping connections.
 - Draw a bounding box with the hand cursor (fingers spread out) around the item(s) of interest.
 - Drag and drop the item(s) in their new location.
- **Menu > Edit > Drag (F8)** - Move one or more objects in the schematic, while keeping connections.
 - Draw a bounding box with the hand cursor (fingers tucked in) around the item(s) of interest.
 - Drag and drop the item(s) in their new location.
- **Menu > Edit > Delete (F5)** - Delete one or more objects.
 - Either click on, or draw a bounding box around, one or more objects with the scissors cursor.
- **Menu > Edit > Duplicate (F6)** - Duplicate one or more objects.
 - Either individually click on, or draw a bounding box around, one or more objects to duplicate with the green + cursor.
 - Click in the schematic editor where you want to place the duplicated object(s).
- **Menu > Edit > Undo (F9)** - Undo the last operation.
- **Menu > Edit > Redo (⇧F9)** - Redo the last Undo operation.
- **Menu > Draft > Wires (F3)** - Place one or more wires in the schematic.
 - Click on the beginning, intermediate, and end points of the wire(s).
 - Right-click to cancel placement of current wire.
- **Menu > Draft > Component (F2)** - Select and place one or more components in the schematic.
 - Enter a component name in the search field and select it from the list.
- **Menu > Draft > Net Name (F4)** - Place one or more net labels in the schematic.
 - Choose or enter a net name.
 - Choose the port type.
 - Net labels replace the arbitrary node names generated by the netlister.
- **Menu > Draft > SPICE Directive (S)** - Place SPICE directive(s) text in the schematic.
 - Right-click in the text field and select **Help me edit > Analysis Cmd** from the contextual menu to use the helper for building complete SPICE directives.
 - Type Control-Enter to start a new line.
 - The SPICE directives will be included in the netlist.
- **Menu > Draft > Comment Text (T)** - Place comment text in the schematic.
 - The comment text will not be included in the netlist.
- **Menu > Draft > .op Data Label** - Display one or more node voltages.
 - Click on a circuit net (wire) to attach a label to that node.
 - The values will be displayed once a circuit simulation is run.

Object Actions

- Initiate object actions by right-clicking on an object within the schematic editor.
- Available actions are displayed in the middle of the toolbar when hovering over an object.
- **Component Symbol** - Edit the component's parameters, including its model and value.
 - The cursor will change to a left-pointing hand when hovering over a component's symbol.
- **Component Name** - Edit the component's reference designator and display options.
 - The cursor will change to a vertical bar when hovering over a component's name.
- **Component Value** - Edit the component's value and display options.
 - The cursor will change to a vertical bar when hovering over a component's value.
- **Net Label** - Edit the net label's parameters.
 - The cursor will change to a vertical bar when hovering over a label.
- **Net (Wire)** - Apply a special operation to the net, e.g. adding a label or highlighting the net.
 - The cursor will not change.

Probing

- When the waveform viewer window is open, the probe tool can be used within the schematic editor to probe simple circuit parameters.
- Hover the cursor over a two-terminal component and the cursor will change to an ammeter with a red arrow showing the direction of positive current flow.
- Click on a wire (probe cursor) to plot the voltage at that node.
- Click on a component (ammeter cursor) to plot the current through the two-terminal component.
- Click on one of the pins (ammeter cursor) of a component with more than two terminals to plot the current through that particular pin of the component.
- Option-click on a wire (ammeter cursor) to plot the current through the wire.
- Option-click on a component (thermometer cursor) to plot the instantaneous power dissipation of the component.
- Control-click on a component (thermometer cursor) to plot the average power dissipation of the component.
- Click and hold on a wire (red probe cursor), drag the cursor to a different wire (gray probe cursor), and release the click to plot the voltage difference between the nodes.
- Text will be displayed in the middle of the schematic capture window toolbar to let you know what you are about to plot.
- Clicking the same voltage or current twice will delete all other traces and plot only the double-clicked one.

Adding Component Models And Subcircuits

- Place an intrinsic component whose symbol matches the type of device you plan to use. For instance, choose *npn* for a generic NPN transistor.
- To utilize a built-in model supplied by LTspice:

- Edit the component (right-click on symbol).
- Click the **Pick New ...** button from the dialog window.
- Select the existing model from the list that you want to use.
- Click the **OK** button.
- To utilize a user supplied model or subcircuit:
 - If supplying a subcircuit specifically:
 - Edit the component's attributes (control-right-click on symbol).
 - Change the value of the *Prefix* attribute, e.g. *QN*, to *X*, denoting that we will use a subcircuit to describe the device.
 - Click the **OK** button.
 - Change the component's value, e.g. *NPN*, to the device name used for the `.MODEL/.SUBCKT` SPICE directive.
 - Include the model or subcircuit, by either
 - including the `.MODEL/.SUBCKT` directive directly in the schematic, or
 - adding an `.INCLUDE` directive in the schematic that points to the file containing the `.MODEL/.SUBCKT` directive.
 - Make sure the pin order for the `.MODEL/.SUBCKT` directive matches the pin order of the component symbol.

Other Operations

- Move around the schematic window by clicking and dragging the schematic to a new location.
- **ESC** Key - Exit action mode, end an operation.
- **Control-E** - Mirror a component during a placement, move, or drag operation.
- **Control-R** - Rotate a component during a placement, move, or drag operation.
- **G** - Place a ground symbol.
- **Zoom In** - Two-finger pinching or two-finger scrolling up.
- **Zoom Out** - Two-finger spreading or two-finger scrolling down.

Waveform Viewer Window

General Information

- The waveform viewer provides an interface for viewing traces (plots) of simulated circuit parameters and automatically appears when a circuit simulation is run.
- Expressions of circuit parameters can be plotted by manually adding traces (clicking the **Add Trace(s)** button from the toolbar or selecting **Add Traces** from the contextual menu).
 - Available circuit parameters will be provided that can be clicked to add them to your expressions for plotting.
 - Various operations and functions are available for use within expressions. They are listed in the *Waveform Viewer > Waveform Arithmetic* section of the LTspice Help page.
- All trace manipulation operations (deleting, duplicating, etc.) continue to be performed, allowing for subsequent manipulations, until a right-click is performed or the **ESC** key is pressed to end the operation.

Toolbar

- **Zoom to Fit** Button - Zoom to fit all the trace(s) within the window.
- **Autorange** Button - Applies an autorange to the axes of an already zoomed in area.
- **Add Trace(s)** Button - Add trace(s) to plot in the viewer.
- **Control Panel** Button - Opens the Control Panel where LTspice preferences can be edited.
- Locations, x and y, will be shown on the right side of the toolbar when hovering the cursor over plot(s).
- Available object actions are displayed in the middle of the toolbar when hovering over an axis or a trace label.

Contextual Menu

- View the contextual menu by right-clicking, or control-clicking, in a plot pane of the waveform viewer.
- **Menu > Zoom to Fit** - Zoom to fit all the trace(s) within the window.
- **Menu > View > Grid** - Toggle the display of a grid in the waveform viewer.
- **Menu > Add Traces (A)** - Add trace(s) to plot in the viewer.
- **Menu > Add Plot Plane** - Add an additional plotting pane above the existing pane(s).
- **Menu > Delete Plot Plane** - Delete the plotting pane.
- **Menu > Edit > Delete (F5)** - Delete one or more traces.
 - Click on one or more trace labels with the scissors cursor.
- **Menu > Edit > Duplicate (F6)** - Duplicate one or more traces.
 - Click on a trace to duplicate with the green + cursor.
 - Click where you want to place the duplicated trace.

- **Menu > Edit > Undo (F9)** - Undo the last operation.
- **Menu > Edit > Redo (⇧F9)** - Redo the last Undo operation.
- **Menu > Reload Plot Settings (space)** - Reload the plot configuration file.

Object Actions

- Initiate object actions by right-clicking on an object within the waveform viewer.
- Available actions are displayed in the middle of the toolbar when hovering over an object.
- **Axis** - Edit the axis' display options.
 - The cursor will change to a horizontal (x-axis) or vertical (y-axis) ruler when hovering over an axis.
- **Trace Label** - Edit the trace's expression and display options.
 - The cursor will change to a vertical bar when hovering over a trace's label.

Other Operations

- Draw a bounding box with the cursor to zoom into an area of interest. The size of the box being drawn is shown within the toolbar. This operation is also useful for measuring the difference between two points on a plot, just press the **ESC** key before ending the drag operation so as to not actually perform the zoom operation.
- Click a trace label to attach a cursor. Click and drag the origin of the attached cursor (hand cursor) to follow the trace lines. A second attached cursor can be attached to another trace to display the difference between the two attached cursors.
- Traces can be moved between plot panes by clicking and dragging a trace label from one pane to another. If you press the Control key when releasing the click, the trace will be copied instead of being moved.
- Save the plot configuration view (.plt) for the current trace(s) by selecting **File > Save** or **File Save As** from the main menu. The plot configuration with the same base name as the schematic will be automatically loaded upon running the next simulation.

Revisions

Version	Date	Modifications
1.0	01/06/2021	Initial version.