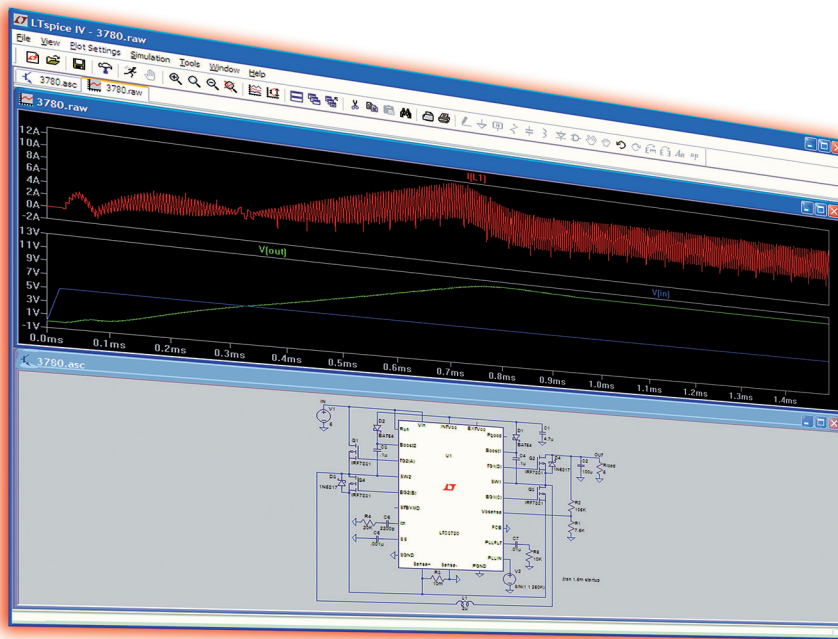


LTspice



- Free Analog Circuit Simulator
- Unlimited Nodes/Nets
- Fast Simulations

LTspice® is a high performance SPICE simulator, schematic capture and waveform viewer designed to speed the process of power supply design. LTspice adds enhancements and models to SPICE, significantly reducing simulation time compared to typical SPICE simulators, allowing one to view waveforms for most switching regulators in minutes compared to hours for other SPICE simulators.



www.twitter.com/ltspice

Follow @ LTspice on Twitter for up-to-date information on models, demo circuits, events and user tips.



www.linear.com/ltspice

Included in the download is a complete and fully functional SPICE program, help files, macro models for Linear Technology's power products, over 200 op amp models, as well as models for resistors, transistors and MOSFETs.



video.linear.com/ltspice

View Instructional Videos.

Sign up for the Linear Insider email newsletter to receive LTspice updates @ www.linear.com/mylinear



See Demo



LT, LT, LTC, LTM, LTspice, Linear Technology and the Linear logo are registered trademarks of Analog Devices, Inc. All other trademarks are the property of their respective owners.

LTspice HotKeys

| LTspice HotKeys | | | |
|--------------------------|---------------------------|-----------------------------|--------------------------|
| Schematic | Symbol | Waveform | Netlist |
| ESC - Exit Mode | ESC - Exit Mode | | |
| F3 - Draw Wire | | | |
| F5 - Delete | F5 - Delete | F5 - Delete | |
| F6 - Duplicate | F6 - Duplicate | | |
| F7 - Move | F7 - Move | | |
| F8 - Drag | F8 - Drag | | |
| F9 - Undo | F9 - Undo | F9 - Undo | F9 - Undo |
| Shift+F9 - Redo | Shift+F9 - Redo | Shift+F9 - Redo | Shift+F9 - Redo |
| Ctrl+Z - Zoom Area | Ctrl+Z - Zoom Area | Ctrl+Z - Zoom Area | |
| Ctrl+B - Zoom Back | Ctrl+B - Zoom Back | Ctrl+B - Zoom Back | |
| Space - Zoom Fit | | Ctrl+E - Zoom Extents | |
| Ctrl+G - Toggle Grid | | Ctrl+G - Toggle Grid | Ctrl+G - Goto Line # |
| U - Mark Unconn. Pins | Ctrl+W - Attribute Window | '0' - Clear | |
| A - Mark Text Anchors | Ctrl+A - Attribute Editor | Ctrl+A - Add Trace | |
| Alt+Click - Power | | Ctrl+Y - Vertical Autorange | Ctrl+R - Run Simulation |
| Ctrl+Click - Attr. Edit | | Ctrl+Click - Average | |
| Ctrl+H - Halt Simulation | | Ctrl+H - Halt Simulation | Ctrl+H - Halt Simulation |
| | | | |
| View | Symbol | Waveform | Netlist |
| R - Resistor | R - Rectangle | | |
| C - Capacitor | C - Circle | | |
| L - Inductor | L - Line | | |
| D - Diode | A - Arc | | |
| G - GND | | | |
| S - Spice Directive | | | |
| T - Text | T - Text | | |
| F2 - Component | | | |
| F4 - Label Net | | | |
| Ctrl+E - Mirror | Ctrl+E - Mirror | | |
| Ctrl+R - Rotate | Ctrl+R - Rotate | | |

Command Line Switches

| Flag | Short Description |
|--------------|-----------------------------------------------------|
| -ascii | Use ASCII .raw files. (Degrades performance!) |
| -b | Run in batch mode. |
| -big or -max | Start as a maximized window. |
| -encrypt | Encrypt a model library. |
| -FastAccess | Convert a binary .raw file to Fast Access Format. |
| -netlist | Convert a schematic to a netlist. |
| -nowine | Prevent use of WINE(Linux) workarounds. |
| -PCBnetlist | Convert a schematic to a PCB netlist. |
| -registry | Store user preferences in the registry. |
| -Run | Start simulating the schematic on open. |
| -SOI | Allow MOSFET's to have up to 7 nodes in subcircuit. |
| -uninstall | Executes one step of the uninstallation process. |
| -wine | Force use of WINE(Linux) workarounds. |

LTspice



See Demo

Simulator Directives - Dot Commands

| Command | Short Description |
|-----------|----------------------------------------------------|
| .AC | Perform a Small Signal AC Analysis |
| .BACKANNO | Annotate the Subcircuit Pin Names on Port currents |
| .DC | Perform a DC Source Sweep Analysis |
| .END | End of Netlist |
| .ENDS | End of Subcircuit Definition |
| .FOUR | Compute a Fourier Component |
| .FUNC | User Defined Functions |
| .FERRET | Download a File Given the URL |
| .GLOBAL | Declare Global Nodes |
| .IC | Set Initial Conditions |
| .INCLUDE | Include another File |
| .LIB | Include a Library |
| .LOADBIAS | Load a Previously Solved DC Solution |
| .MEASURE | Evaluate User-Defined Electrical Quantities |
| .MODEL | Define a SPICE Model |
| .NET | Compute Network Parameters in a .AC Analysis |
| .NODESET | Supply Hints for Initial DC Solution |
| .NOISE | Perform a Noise Analysis |
| .OP | Find the DC Operating Point |
| .OPTIONS | Set Simulator Options |
| .PARAM | User-Defined Parameters |
| .SAVE | Limit the Quantity of Saved Data |
| .SAVEBIAS | Save Operating Point to Disk |
| .STEP | Parameter Sweeps |
| .SUBCKT | Define a Subcircuit |
| .TEMP | Temperature Sweeps |
| .TF | Find the DC Small-Signal Transfer Function |
| .TRAN | Do a Nonlinear Transient Analysis |
| .WAVE | Write Selected Nodes to a .WAV file |

Suffix

| Suffix | Suffix |
|--------|---------|
| | f |
| T | 1e12 |
| G | 1e9 |
| Meg | 1e6 |
| K | 1e3 |
| | Mil |
| | 25.4e-6 |

Constants

| | |
|-------|------------------------|
| E | 2.7182818284590452354 |
| Pi | 3.14159265358979323846 |
| K | 1.3806503e-23 |
| Q | 1.602176462e-19 |
| TRUE | 1 |
| FALSE | 0 |