WIKIPEDIA

LTspice

LTspice is freeware[4][5] computer software implementing a SPICE electronic circuit simulator, produced by semiconductor manufacturer Linear Technology, now part of Analog Devices.[2] It is used in-house at Linear Technology for IC design, and the most widely distributed and used SPICE program in the industry.[3] LTspice isn't artificially crippled to limit its capabilities (no node limits, no component limits, no subcircuit limits).[3][6]

Contents

LTspice	
Original author(s)	Mike Engelhardt[1]
Developer(s)	Linear Technology, Analog Devices[2]
Initial release	October 1999[1]
Stable release	XVII / July 9, 2019[3]
Operating system	Windows 7, 8, 8.1, 10, macOS 10.7+
Platform	IA-32, x86-64
Size	Windows (41 MB), MacOS (101 MB)
Available in	English
Type	Electronic design automation
License	Freeware[4][5]
Website	LTspice webpage (https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html)

Overview

LTspice provides schematic capture to enter an electronic schematic for an electronic circuit, an enhanced SPICE type analog electronic circuit simulator, and a waveform viewer to show the results of the simulation.[2] Circuit simulation analysis based on transient, noise, AC, DC, DC transfer function, DC operating point can be performed and plotted as well as fourier analysis.[7] Heat dissipation of components can be calculated and efficiency reports can also be generated. It has enhancements and specialized models to speed the simulation of switched-mode power supplies (SMPS) in DC-to-DC converters.[2][8]

A user can create their own device models, import downloaded models from many electronic component manufacturers, or use a model from the device library.[9] LTspice ships with thousands of third-party models (capacitors, diodes, inductors, resistors, transistors, ferrite beads, opto-isolators, 555 timer, and more), as well as macro models for Analog

Devices and Linear Technology parts (ADCs, analog switches, comparators, DACs, filters, opamps, timers, voltage references, voltage supervisors, voltage regulators, 0.01% quad resistor networks, and more).[2][6] In the device library, Analog Devices part numbers start with "AD", and Linear Technology parts start with "LT".[10]

LTspice does not generate printed circuit board (PCB) layouts, but netlists can be exported to layout programs.[11] While LTspice does support simple logic gate simulation, it is not designed specifically for simulating logic circuits.

It is used by many users in fields including radio frequency electronics, power electronics, audio electronics, digital electronics, and other disciplines.

LTspice/SwitcherCAD III

In 1999, LTspice III was released, the first public release.[1] It's designed to run on Windows 95, 98, 98SE, ME, NT4.0, 2K, XP. This version is no longer available for download from Analog Devices. Initially, LTspice III was internally released to Linear Technology's Field Application Engineers (FAE) in October 1999, who then gave it away during customer visits via CDROM media.[1] In June 2001, it was released for public downloading from the Linear Technology website.[1][8] Originally, LTspice/SwitcherCAD ran only on Microsoft Windows platforms, but since 2003 it is able to run under the Wine Windows compatibility layer on Linux.[12]

LTspice IV

In 2008, LTspice IV was released.[1][6][13] It's designed to run on Windows 2K, XP, Vista, 7 with a processor that that contains a minimum instruction set similar to a Pentium 4 processor.[14] Though IV is still available for download, it is no longer maintained. LTspice was originally called SwitcherCAD, but that name was removed when IV was released.[1] A native Apple macOS 10.7+ application was introduced in 2013.[15]

LTspice XVII

In 2016, LTspice XVII was released, and currently it's the latest version.[3] It's designed to run on 32-bit or 64-bit editions of Windows 7, 8, 8.1, 10, and macOS 10.7+.[2]

Summary of major changes from LTspice IV to LTspice XVII are:

- Add 64-bit executables.[3]
- Add Unicode characters in schematics, netlists, plot.[3]
- Add device equations for IGBT, diode soft recovery, arbitrary state machine.[3]
- Add user symbol and library directory search paths. See option setting at LTspice -> Tools -> Control Panel -> Sym. & Lib. Search Paths.[16]
- Add schematic thumbnail and preview support on Microsoft Windows.[3]
- Add editors for most SPICE commands.[3]
- Add multi-monitor support.[3]

Updates

After Analog Devices purchased Linear Technology in 2016, models for Analog Devices parts were added to the LTspice device library over time.

Every month, minor updates are released by Analog Devices for software fixes, updates for existing device models, and new device models. A user can manually update their installed copy by doing a "Sync Release" from the pulldown menu. LTspice has a "Show Change Log" screen that lists every change to the software and models since LTspice XVII was released. LTspice doesn't show any subversion numbers as many software programs do, though the software "build date" could be treated as such. For example, on 64-bit Windows, the "about" screen shows "LTspice XVII(x64)" then a build date and time.

Number conventions

In LTspice, numeric values can be expressed in four different ways: integer (i.e. 1000), real (i.e. 1000.0), scientific e-notation (i.e. 1e3, 1.0e3), scale factor notation (i.e. 1K, 1K0).[17]

If the first character after a number isn't the letter "e" for scientific e-notation or a scale factor suffix (left column of table), then trailing characters are ignored.[17] For example, 5 is treated the same as 5V / 5Volt / 5Volts / 5 Hz / 5Hertz.

Scale factors

Integer and real numbers supports a scale factor (multiplier) suffix.[17] These are based mostly on metric conventions.

The suffix (left column) can be upper / lower / mixed case, known as case insensitive.[17] For example, 1MEG / 1meg / 1Meg represents 1000000; 1k / 1K represents 1000.

Any appended text after the suffix (left column) is ignored.[17] For example, 2MegHz / 2MegaOhm represents 2000000; 3mV / 3mOhm represents 0.003; 4uF / 4uHenry represents 0.000004.

In LTspice, any suffix (left column) can replace the decimal point of a real number, a common format for printed schematics.[17][16] For example, 4K7 represents 4700, 1u8 represents 0.0000018.

SPICE Suffix[17]	Metric Name	English Name	Power of 10	Numeric Value	Notes and Common Mistakes
T	tera	Trillion	10^{12}	1000000000000	
G	giga	Billion	10^{9}	1000000000	
MEG	mega	Million	10^{6}	1000000	Wrong use of m / meg / mil are common mistakes in all SPICE programs
K	kilo	Thousand	10^{3}	1000	
m	milli	Thousandth	10^{-3}	0.001	"1m" & "1M" doesn't mean "1 megaohm, instead "1MEG" is correct[17]
u or μ	micro	Millionth	10^{-6}	0.000001	Older SPICE software doesn't support the μ (Mu) character[16]
n	nano	Billionth	10^{-9}	0.000000001	
p	pico	Trillionth	10^{-12}	0.000000000001	
f	femto	Quadrillionth	10^{-15}	0.000000000000001	"1f" & "1F" doesn't mean "1 farad, instead "1" is correct[17]
mil	thou		25.4 x 10^{-6}	0.0000254	mil is a thousandth of an inch (0.001") which is 25.4 μm[17]

Compatibility

Although LTspice was originally based upon Berkeley SPICE 3f5 source code,[1] it no longer does, thus some of its features may create non-portable files. It should be noted that competitor SPICE programs have non-portable features too.

LTspice features that may not be supported by some SPICE programs:

- Unicode characters in schematics, netlists, plot.[3] This is a new feature in LTspice XVII, thus older versions of LTspice don't support Unicode.[3]
- μ (Mu) character as an alternate symbol for micro (10^{-6}) scale factor, which is Unicode decimal 181 (hex 0xB5). See option setting at LTspice -> Tools -> Control Panel -> Netlist Options -> Convert 'μ' to 'u'.[16]
- Suffix replacing decimal point. See option setting at LTspice -> Tools -> Control Panel -> SPICE -> Accept 3K4 as 3.4K.[16]

LTspice doesn't support the following features:

- LTspice doesn't support scale factors of "atto" 10^{-18}, "peta" 10^{15}, or "exa" 10^{18}.
- HSPICE supports 'a' for "atto" 10^{-18}, which must be replaced with 'e-18' to be compatible with LTspice and other SPICE software.
- HSPICE supports 'X' as a synonym for "meg" 10^6, which must be replaced with 'e6' or 'MEG' to be compatible with LTspice and other SPICE software.

Node name conventions

In LTspice, a node/net (connection point) on the schematic can be labeled by using the Label Net tool button or F4 key. The "Label Net" wizard has three choices for a label, two predefined graphical symbols (GND, COM), or a user-defined node/net name.[18]

The two graphical symbols represent:

- GND - The ground symbol assigns a node with a special global net name of "0".[18]
- COM - The COM symbol assigns a mode with a net name of "COM", which doesn't have any special significance.[18]

Historically, SPICE and older version of LTspice software only supported printable ASCII characters for node/net names, then LTspice XVII added support for Unicode characters.[3]

A user-defined name supports two optional features that can be prepended to the text name:

- _ - An underscore causes an overbar to be placed above the entire name, which commonly means an active low signal. For example, "_RESET" is shown on the schematic as "RESET".[19]
- $G_ - This means a node is global, no matter where the name occurs in the circuit hierarchy. For example, "$G_ENABLE" / "$G_ERROR". It should be noted the ground symbol is treated in a similar way, but it doesn't have "$G_" prepended to it.[18]

When a node/net name is placed on a schematic, it will have one of five different visual representations. Two are automatically determined, while three others are chosen by the "Port Type" field in the "Label Net" wizard.[18]

- None - Bare text. This is the default.[20]
- Global - "Rectangle" around the text. This is automatically shown for a global net name that starts with "$G_".[20]
- Input - "Rectangle with triangle end" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.[20]
- Output - "Rectangle with triangle on other end" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.[20]
- Bidirectional - "Rectangle with triangle on two ends" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.[20]

File format

Many of the LTspice files are stored as an ASCII text file, which can be viewed or edited with any ASCII text editor programs. One of the side benefits of an ASCII file format is that a schematic can be listed in a printed document / book / magazine / datasheet / research paper / homework assignment, which allows the reader to recreate LTspice files without electronic file distribution.

LTspice filename extensions:[21]

- `asc` - schematic. It consists of a netlist based on SPICE text-based commands.[21]
- `asy` - electronic symbol shown in a schematic.[21]
- `cir` - external netlist input.[21]
- `fft` - FFT binary output.[21]
- `lib` - model library subcircuits.[22]
- `plt` - waveform viewer plot settings.[21]
- `raw` - binary output, optional ASCII output.[21]
- `sub` - subcircuit.[22]
- `lib` / `sub` / `mod` / `model` - device model. While any file extension is allowed, users tend to gravitate towards common ones.[21]

Example

The following example can be viewed by copying each into two different text files. For each, copy the text in the gray box from this article, paste into an ASCII text editor, saving as a text file. Both files must have the same "base name" and sit in the same directory. To see it, opening the "asc" file with LTspice then click the "Run" button inside LTspice software.

- LTspice_RC.asc
- LTspice_RC.plt

Schematic file

LTspice schematics are stored as an ASCII text file with a filename extension of "**asc**".[21]

The following example shows the contents from a small LTspice schematic file for a simple RC circuit with four schematic symbols: V1 is 10 volt DC voltage source, R1 is 1K ohm resistor, C1 is 1 uF capacitor, ground. The bottom three TEXT lines are: 1) a transient simulation directive with a stop time parameter of 10 ms (`.tran 10mS`), 2) a SPICE directive to set the initial condition of RC "out" net to zero volts (`.ic v(OUT)=0V`), and 3) a text comment (title).

```
Version 4
SHEET 1 880 680
WIRE 224 96 128 96
WIRE 128 160 128 96
WIRE 224 192 224 176
WIRE 288 192 224 192
WIRE 224 208 224 192
WIRE 128 288 128 240
WIRE 224 288 224 272
WIRE 224 288 128 288
WIRE 224 304 224 288
FLAG 224 304 0
FLAG 288 192 OUT
IOPIN 288 192 Out
SYMBOL res 208 80 R0
SYMATTR InstName R1
SYMATTR Value 1K
```

DOCKET ALARM

Explore Litigation Insights

Docket Alarm provides insights to develop a more informed litigation strategy and the peace of mind of knowing you're on top of things.

Real-Time Litigation Alerts

Keep your litigation team up-to-date with **real-time alerts** and advanced team management tools built for the enterprise, all while greatly reducing PACER spend.

Our comprehensive service means we can handle Federal, State, and Administrative courts across the country.

Advanced Docket Research

With over 230 million records, Docket Alarm's cloud-native docket research platform finds what other services can't. Coverage includes Federal, State, plus PTAB, TTAB, ITC and NLRB decisions, all in one place.

Identify arguments that have been successful in the past with full text, pinpoint searching. Link to case law cited within any court document via Fastcase.

Analytics At Your Fingertips

Learn what happened the last time a particular judge, opposing counsel or company faced cases similar to yours.

Advanced out-of-the-box PTAB and TTAB analytics are always at your fingertips.

API

Docket Alarm offers a powerful API (application programming interface) to developers that want to integrate case filings into their apps.

LAW FIRMS
Build custom dashboards for your attorneys and clients with live data direct from the court.

Automate many repetitive legal tasks like conflict checks, document management, and marketing.

FINANCIAL INSTITUTIONS
Litigation and bankruptcy checks for companies and debtors.

E-DISCOVERY AND LEGAL VENDORS
Sync your system to PACER to automate legal marketing.

WHAT WILL YOU BUILD? | sales@docketalarm.com | 1-866-77-FASTCASE

fastcase®
Smarter legal research.