RaRa Academy: Circuit Simulation With LTSpice

Karl Heinz Kremer – K5KHK

Some History

Initial release back in 1973

Punch Cards

Used for integrated circuit simulation

Some Other Tools

Elsie - http://tonnesoftware.com/elsie.html

MicroCAP - https://archive.org/details/mc12cd_202110

CircuitLab - https://www.circuitlab.com

KiCAD - https://www.kicad.org

LTSpice

Free to use – no implementation limitations in the free version

Graphical User Interface – no more punch cards

Created by Linear Technologies – hence the LT

LT was bought by Analog Devices

DC Operating Point Analysis	.op
DC Sweep Analysis	.dc
DC Transfer Function Analysis	.tf
AC Analysis	.ac
Transient Analysis	.tran
Noise Analysis	.noise
Parametric Analysis	.step
Temperature Analysis	.temp
Monte Carlo Analysis	.mc

Links

Download LTSpice:
https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

Keyboard shortcuts:
https://www.analog.com/media/en/news-marketing-collateral/solutions-bulletins-brochures/ltspice_shortcutflyer.pdf

Keyboard shortcuts for macOS version:
https://www.analog.com/media/en/news-marketing-collateral/solutions-bulletins-brochures/ltspiceshortcutsformacosx.pdf

SpiceMan:
https://spiceman.net

Groups.IO:
https://groups.io/g/LTspice