



Electronic Circuits

Lecture 1.1: Installing Ltspice & First Glance

LTspice

- LTspice is a powerful, fast, and free SPICE simulator software, schematic capture and waveform viewer with enhancements and models for improving the simulation of analog circuits. Its graphical schematic capture interface allows you to probe schematics and produce simulation results, which can be explored further through the built-in waveform viewer.

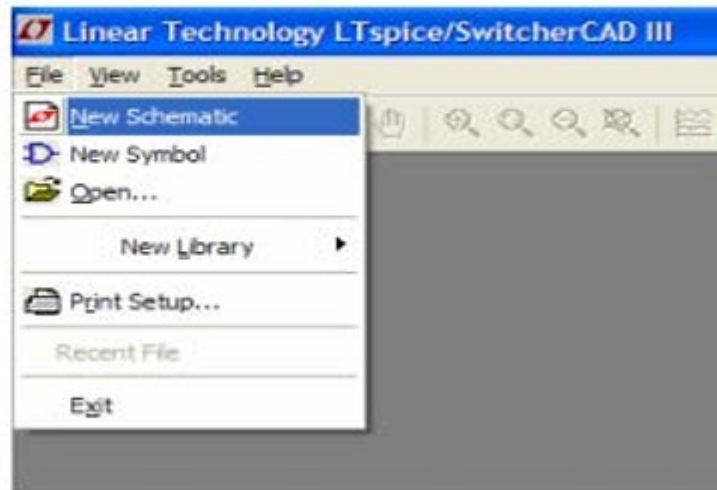
<https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator.html>

Download & Support

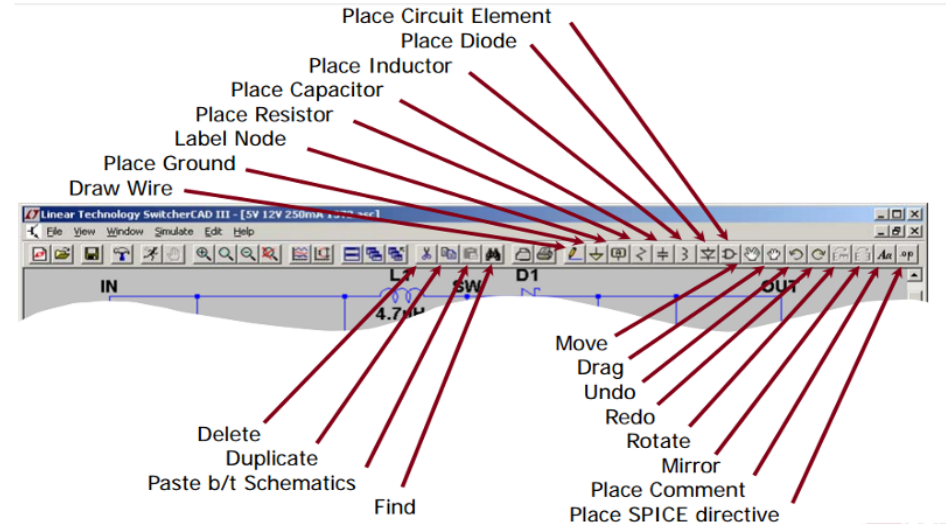
- [Download for Windows 10 64-bit and forward Version 24.0.9](#)
- [Download for MacOS 10.15 and forward Version 17.2.2](#)
- [Download for Windows XP \(End of Support\)](#)
- [Download for MacOS 10.9 \(End of Support\)](#)
- [Download LTspice XVII for Windows \(End of Support\)](#)
- Getting Started Tutorial:
<https://www.analog.com/en/education/education-library/videos/video-series/ltspice-getting-started-tutorial.html>
- Essentials Tutorial:
<https://www.analog.com/en/education/education-library/videos/video-series/ltspice-essentials-tutorial.html>

Use of LTspice: New design & Add a component

⇒ Select 'File' and 'New Schematic'.

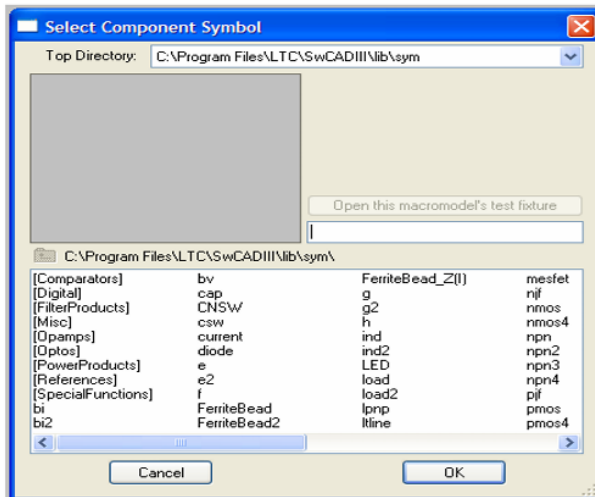


- To add a resistor, press 'R' or click the resistor button.
- Press 'L' or 'C' to insert an inductor or a capacitor respectively. Otherwise, click the respective buttons.



Use of LTspice: Add a component from the list & Change its properties

For any other components, press F2 or click the component button. A 'Select Component Symbol' window will open and then from there, browse to get the desired component.

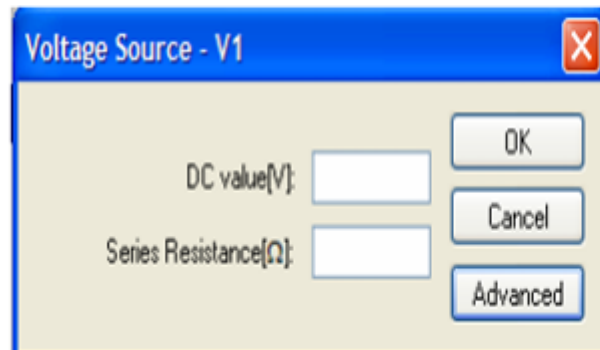


Right click on the component and enter the value in default unit (Ω , H, F etc.)

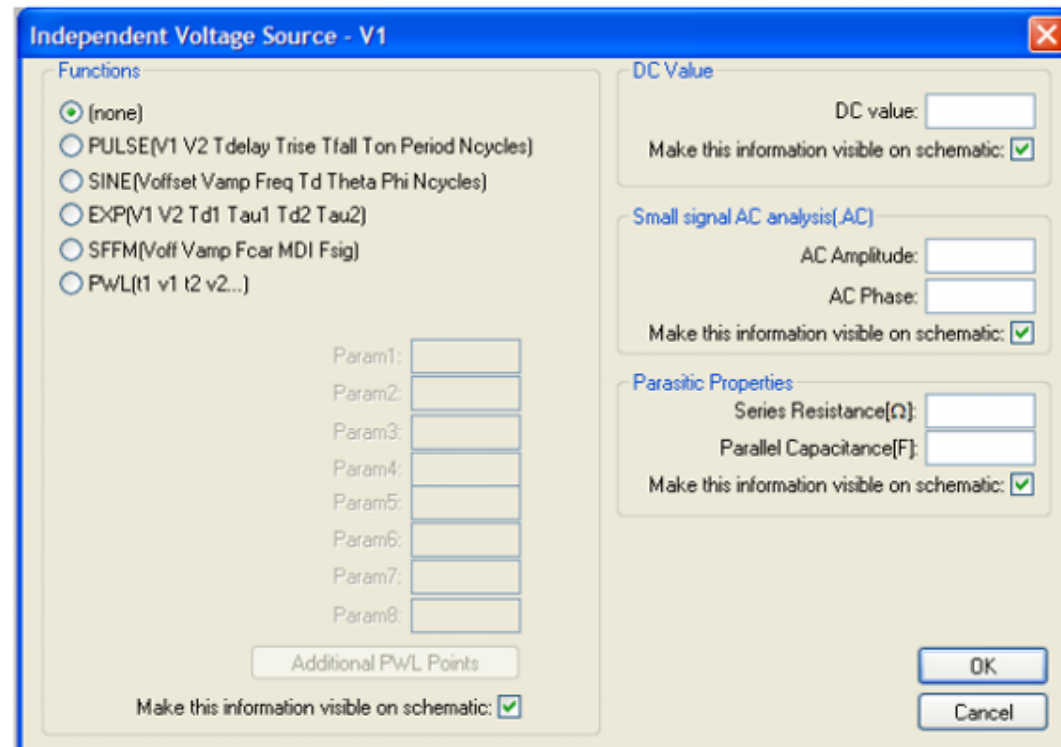


Use of LTspice: Add a DC & other sources

For a **DC Voltage source**,
enter the voltage and series resistance.



For **all other voltage sources**, click 'Advanced' and select the type of voltage source and enter the required parameters.



Use of Ltspice: Example source parameters

$v(t)=5 + 10\sin(2 * \pi * 60 + 45^\circ)\text{V}$ for 100ms as a transient

Independent Voltage Source - V1

Functions

- ☐ (none)
- ☐ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- ☒ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Voff Vamp Fcar MDI Fsig)
- ☐ PWL(t1 v1 t2 v2...)
- ☐ PWL FILE:

DC offset[V]:

Amplitude[V]:

Freq[Hz]:

T delay[s]:

Theta[1/s]:

Phi[deg]:

Ncycles:

Make this information visible on schematic: ☒

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis[AC]

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

Make this information visible on schematic: ☒

Use of LTspice: Some shortcuts

Delete a Component

- Click the scissors in the toolbar and press 'esc' to quit deleting.

Move an element

- To move a component, click the move button and left click on the component you want to move. Move the component to the new location and left click to place the component. Press 'esc' to quit moving components.

Add a wire

- Click the add wire button. Left click on the starting location, move to the ending location and left click again. Press 'esc' to quit adding wires.

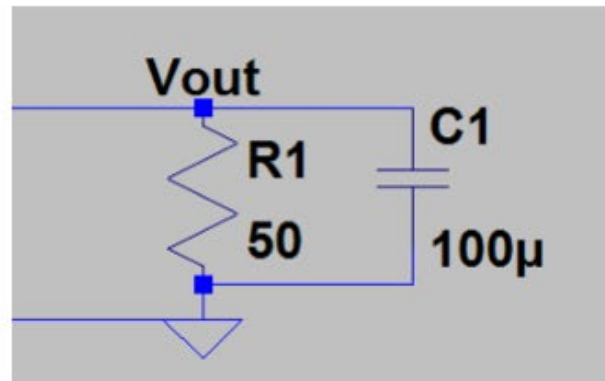
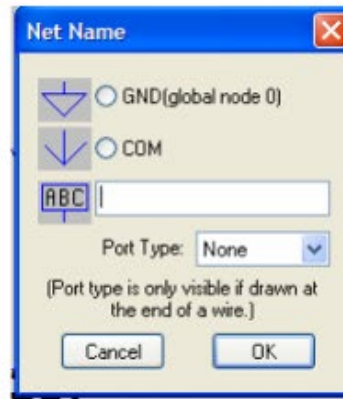
Add ground

- Press 'G' or click the ground button and click on the schematic to add a ground. Press 'esc' to quit adding grounds.

Use of LTspice: Labelling

Label a net

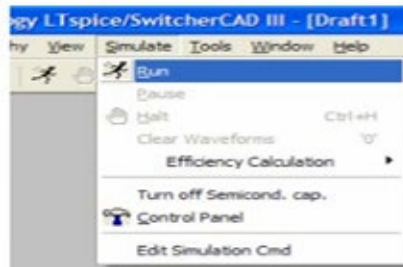
- Press 'F4' or click the add net button. Type the net name and click 'OK'. Click to add the net to the schematic.



Use of LTspice: Simulating & Analyzing

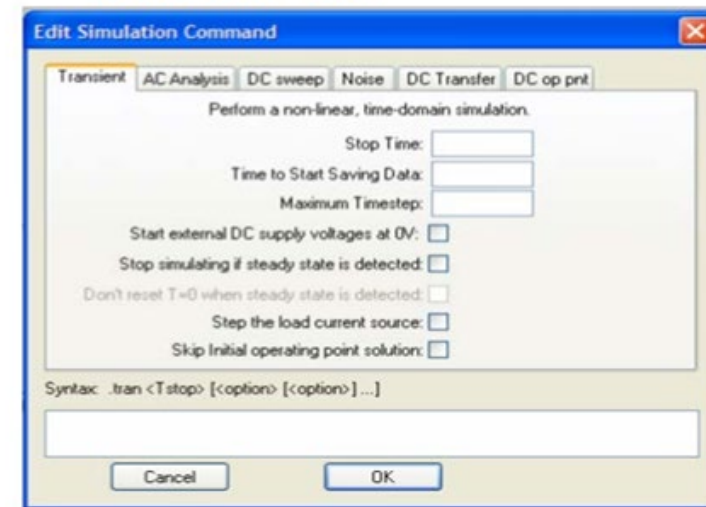
Simulate

- To run a circuit, select 'Simulate' from the file menu and 'Run' or click the run button.



Analysis

- Select the appropriate tab for simulation and enter the simulation options and click 'OK'.



Use of LTspice: Measurements

Measurement

- When making any measurement, a single click will add the measurement to the plot, while a double click will erase all existing measurements and plot the selected measurement by itself.
- To measure a voltage on a node with respect to ground, move the mouse over the node to be measured and click. The voltmeter probe cursor will appear when the mouse cursor is above an appropriate node.



- To measure voltage across a component, click on the positive node to be measured, drag the mouse to the negative node and release the mouse.



- To measure the current through a component, move the mouse cursor over the component and click. The clamp ammeter will appear when the mouse cursor is above an appropriate component.

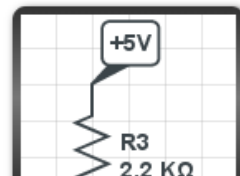


LTspice Alternative: Circuit Lab

■ Online circuit simulator & schematic editor – CircuitLab

Examples

- 555 Timer as Oscillator / PWM Generator
- 7805 & Wall-wart Experiencing Voltage Drop-out
- BJT audio amplifier
- BJT Cascoded Active-load Diff. Amp. with CMFB
- BJT current mirror
- Digital 4-bit counter and DAC
- Diode half-wave rectifier
- Diode full-wave (bridge) rectifier
- Diode turn-off time
- Laplace transform step response and Bode plot
- LED with resistor biasing
- JFET-based electret microphone amplifier
- Mechanical spring dashpot Laplace model
- MOSFET and resistor NAND / NOR gate
- MOSFET (CMOS) NAND / NOR gate
- MOSFET switching inductive load
- Op-Amp inverting / non-inverting amplifier
- RC vs. RL step response / frequency response
- RLC resonance
- Resistors in series and parallel
- Zener diode voltage reference
- *more examples in "Ultimate Electronics" book...*

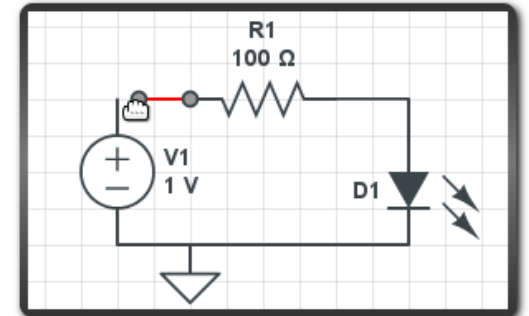


Common sense schematics let you name a node "+5V" and know that the simulator will do the right thing automatically, keeping your schematics compact and elegant.

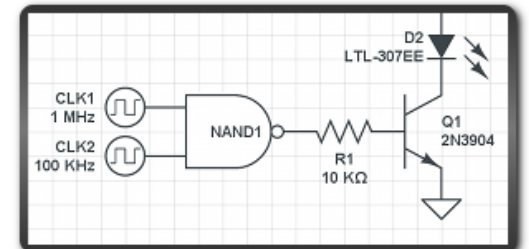
Easy-to-use Power Tools

Easy-wire mode lets you connect elements with fewer clicks and less frustration.

Cross-window copy/paste lets you easily explore and re-mix parts of public circuits from the CircuitLab community.

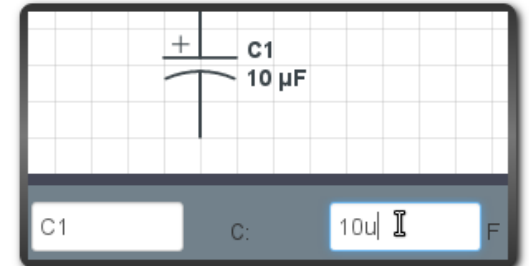


Mixed-mode circuit simulation lets you simulate analog and digital components side-by-side.



SPICE-like component models give you accurate results for nonlinear circuit effects.

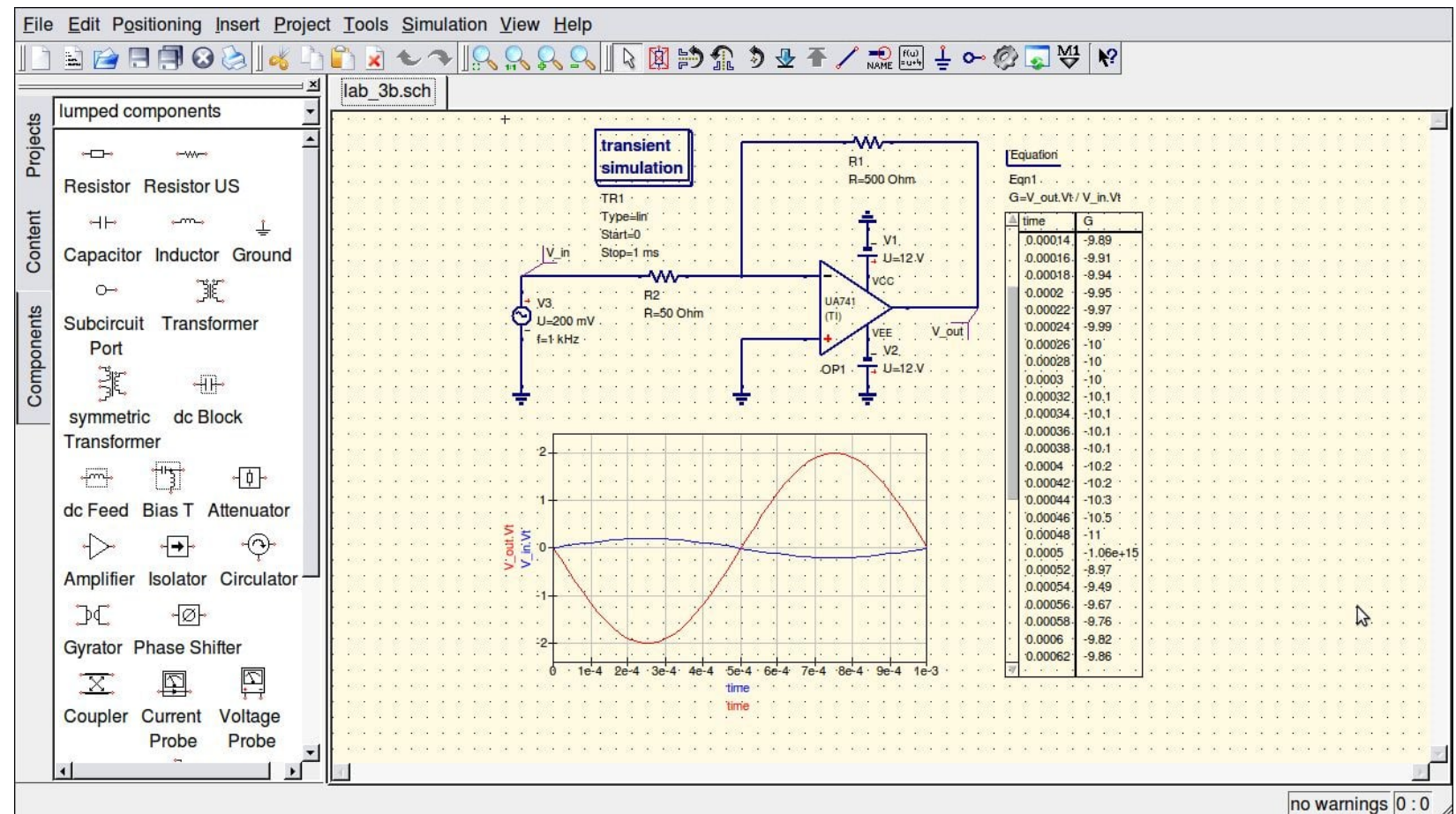
Human-friendly formats let you enter and display values concisely, just like you would on a paper schematic.



Unit-aware expression evaluation lets you plot arbitrary signals of interest, such as differential signals or power dissipation.

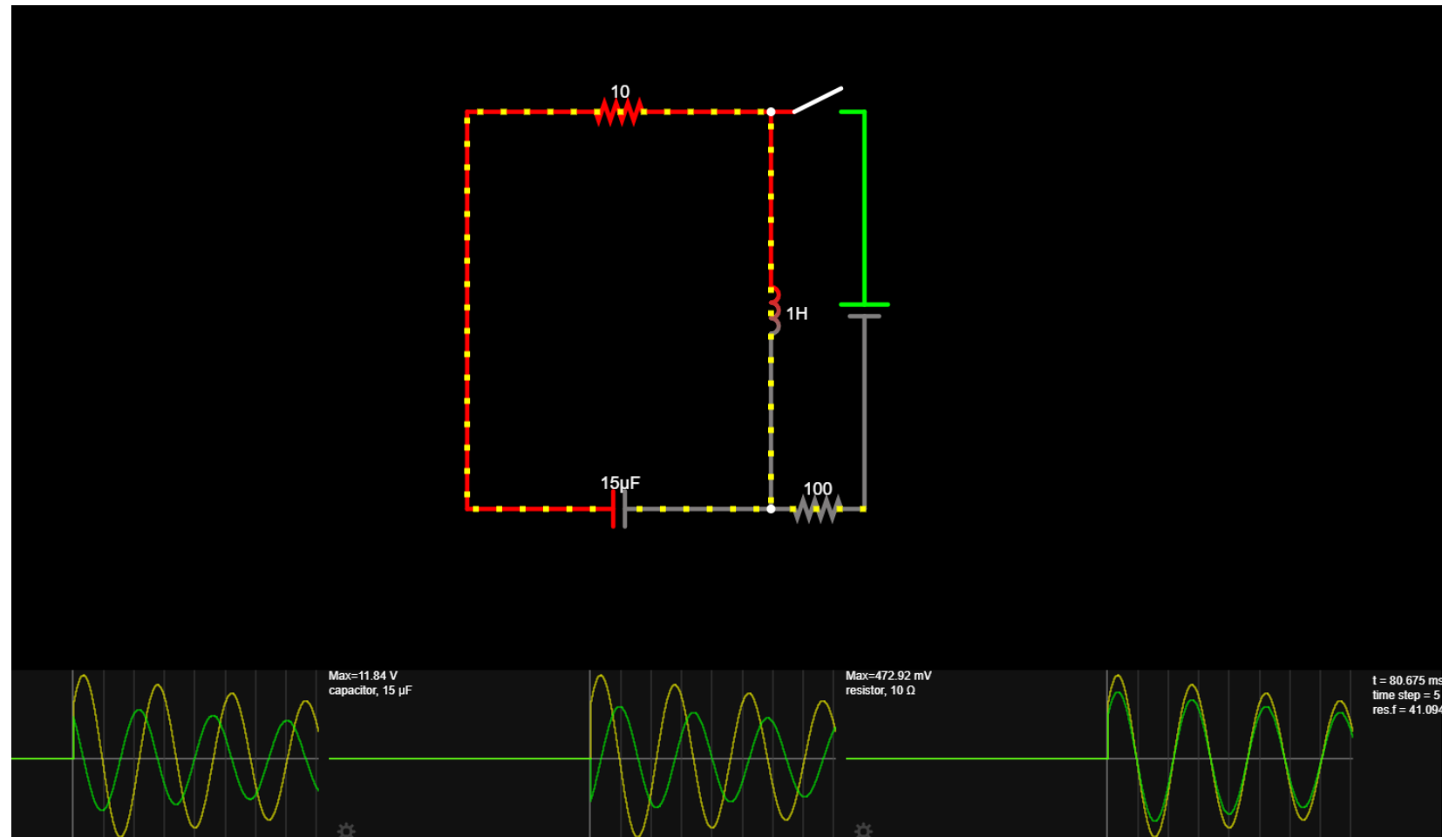
LTspice Alternative: QUCS

- <https://qucs.sourceforge.net/>



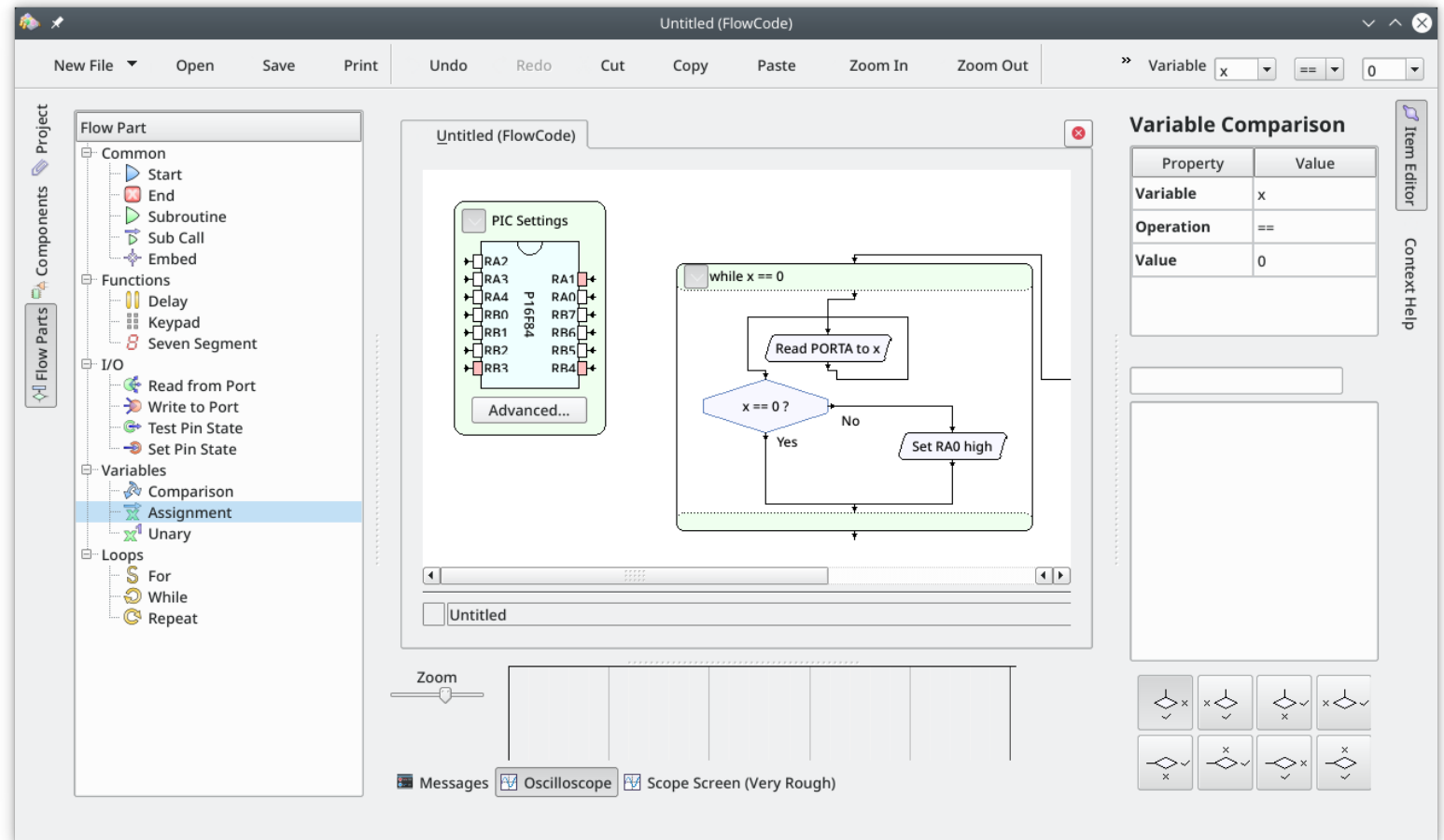
LTspice Alternative: CircuitJS

- <https://www.falstad.com/circuit/circuitjs.html>



LTspice Alternative: KTechlab

- <https://apps.kde.org/tr/ktechlab/>





Thanks for
listening 😊

YALÇIN İŞLER

Assoc. Prof.