CEC Continuing Education Center

DesignNews

Exploring Electronic Circuits with Breadboards, AI Circuit Analysis, and Simulators

DAY 4: Exploring Circuit Simulators – Part 1: Multisim Live and QSPICE

Sponsored by

DigiKey

informa markets

Webinar Logistics

- Turn on your system sound to hear the streaming presentation.
- If you have technical problems, click “Help” or submit a question asking for assistance.
- Participate in ‘Attendee Chat’ by maximizing the chat widget in your dock.

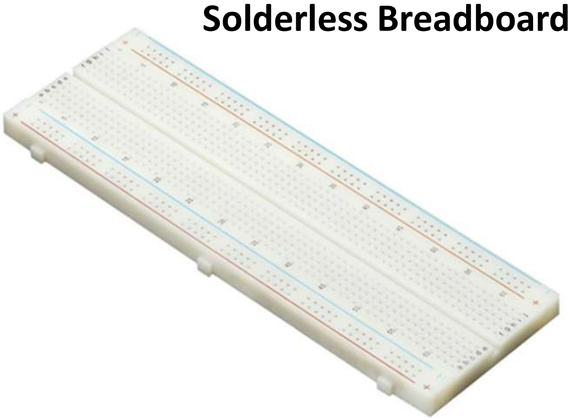
Dr. Don Wilcher

Visit 'Lecturer Profile' in your console for more details.

Course Kit and Materials

Adafruit Parts Pal Kit

Solderless Breadboard



Jumper Wires: Male to Male

Solderless Breadboard Power Supply

CEC Continuing Education Center

DigiKey

Research Perspective

“Breadboards are widely used in early-stage circuit prototyping since they enable users to rapidly try out different components and to change the connections between them” (Zhu et al., 2020).

Agenda:

- Why Circuit Simulators?
  a) Purpose
  b) History Perspective
  c) Examples
- Hands-On With Multisim Live
- Hands-On With QSPICE

Why Circuit Simulators?

- Circuit simulators are valuable tools for electronic circuit design and testing. Circuit simulators are useful for the reasons listed below.
  a) they are faster
  b) more cost-effective
  c) more flexible than physical prototyping.
- Cost-effective than physical manufacturing especially for integrated circuits (ICs).
- Time efficient – Allow quick creation and modification of circuit designs using pre-built models. Several designs can be run simultaneously without the physical setup of test and measurement equipment.

CEC Continuing Education Center

DigiKey

Why Circuit Simulators?. . .
Purpose: . . .

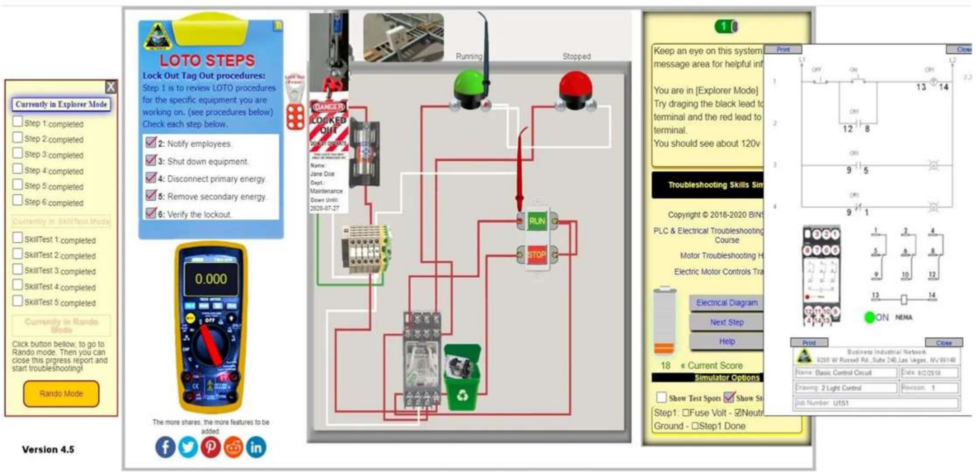
- Circuit simulators are software tools that predict electronic circuit behavior before it's built.
- They can test a circuit's performance and validate its performance.
- Circuit simulators are used in various industries such as:
  - a) manufacturing
  - b) automotive
  - c) aerospace
- Circuit simulators can help students and professionals become proficient at troubleshooting electrical and electronic circuits.
- Circuit simulators allow designers to make final modifications before the printed circuit board (PCB) is been manufactured.

Question 1

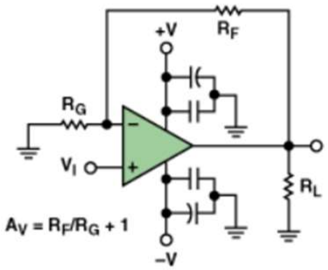
In reviewing slide 8, what industry is not listed as using circuit simulators?

- **a) manufacturing**
- **b)automotive**
- **c)aerospace**
- **d) pharmaceutical**

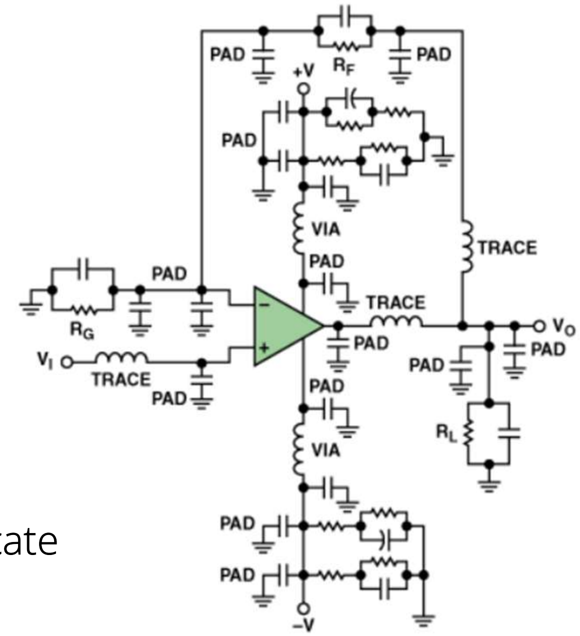
Why Circuit Simulators?. . .
Purpose: . . .



Train



Educate

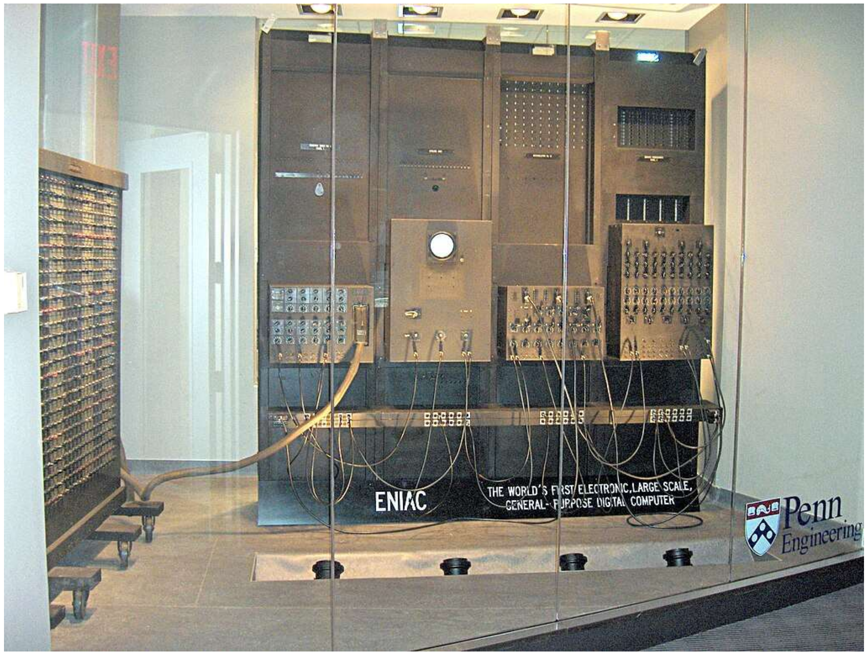


Why Circuit Simulators?...
Historical Perspective:

- The developments in computer-aided circuit analysis and circuit design started in the early 1950s within the IEEE Circuits and Systems (CAS) Society (Pederson, 1984).
- Initially, computer-aided circuit analysis of linear circuits was used to
  a) optimize design
  b) design focus or centering
  c) determine the effects of parasitics
- Early digital computers were used in the analysis of electrical circuits
- Electromechanical relay-based digital computers were programmed in the early 1950s to solve algebraic equilibrium-condition equations of a linear network stimulated by a sinusoidal waveform.

Why Circuit Simulators?...
Historical Perspective: ...

The Electronic Numerical Integrator and Computer (ENIAC) was used to perform early electrical circuit analysis.

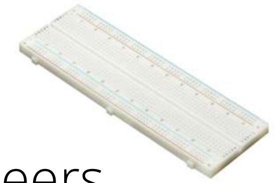


Question 2

What year were electromechanical relay-based digital computers programmed to solve algebraic equilibrium condition equations?

a) 1949
b) 1950s
c) 1938
d) 1960

Why Circuit Simulators?...
Examples:



The following list provides examples of circuit simulators used by engineers and students.

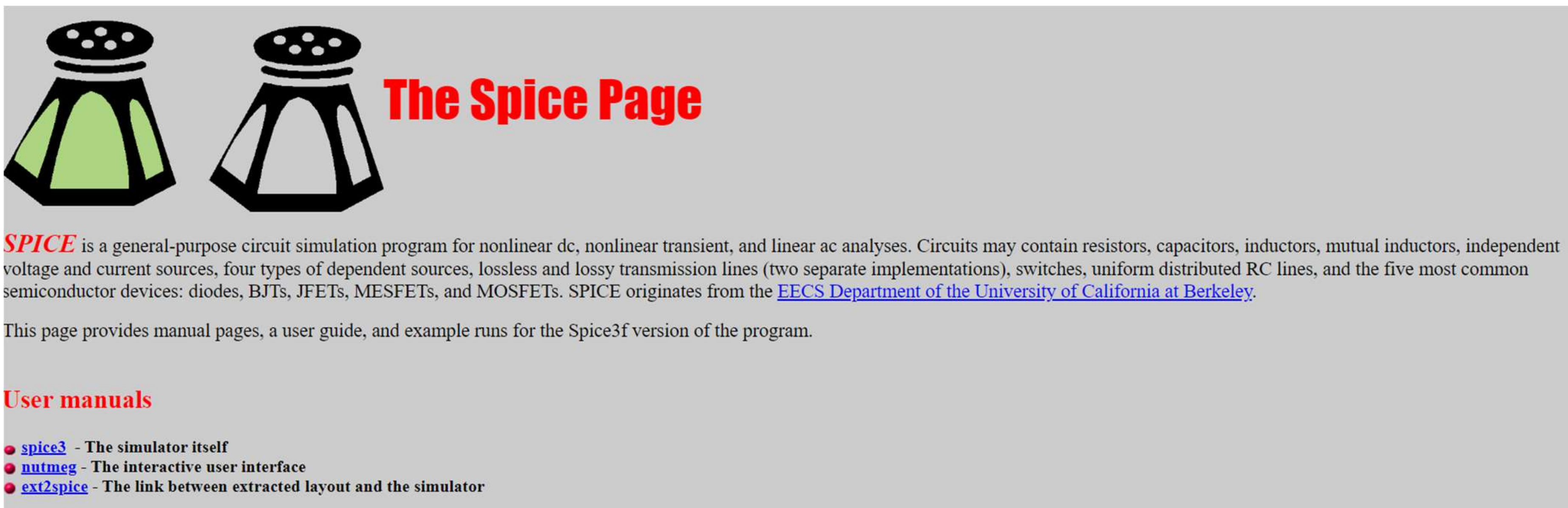
- PSpice is a simulator tool based on SPICE.
- LTspice is a SPICE simulator, schematic capture, and waveform viewer.
- Multisim is a versatile tool that can simulate analog, digital, and mixed-signal circuits. National Instruments is the developer of Multisim.
- Falstad Circuit Simulator is a simple powerful web-based simulation tool for electronic circuits.
- EveryCircuit is a user-friendly mobile app that can simulate real-time electronic circuits.
- Quite Universal Circuit Simulator (QUCS) (open source) that simulates analog, digital, and mixed-signal circuits.
- ngspice (open source) is a mixed-level/mixed-signal circuit simulator.

https://www.ema-eda.com/ema-resources/blog/electronic-circuit-simulation-software-for-students/

Why Circuit Simulators?. . .
Examples: . . .

- The Simulation Program with Integrated Circuit Emphasis (SPICE) was developed in the 1970s at Berkeley.
- SPICE was initially intended to model and simulate integrated circuits (ICs).
- SPICE was written in the Formulated Translator (Fortran) programming language.
- Donald O Pederson, a University of California- Berkeley Electrical Engineering Professor created the SPICE program.

Why Circuit Simulators?. . .
Examples: . . .



The Spice Page

SPICE is a general-purpose circuit simulation program for nonlinear dc, nonlinear transient, and linear ac analyses. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, lossless and lossy transmission lines (two separate implementations), switches, uniform distributed RC lines, and the five most common semiconductor devices: diodes, BJTs, JFETs, MESFETs, and MOSFETs. SPICE originates from the EECS Department of the University of California at Berkeley.

This page provides manual pages, a user guide, and example runs for the Spice3f version of the program.

User manuals

- **spice3 - The simulator itself**
- **nutmeg - The interactive user interface**
- **ext2spice - The link between extracted layout and the simulator**

https://web.archive.org/web/20231208045915/http://bwrcs.eecs.berkeley.edu/Classes/IcBook/SPICE/

Why Circuit Simulators?. . .
Examples: . . .

SPICE Program

Example Circuits

ircuit 1: Differential Pair

e following deck determines the dc operating point of a simple differential pair. In addition, the ac small-signal response is computed over the frequency range 1Hz to 100MEGHz.

```
SIMPLE DIFFERENTIAL PAIR
VCC 7 0 12
VEE 8 0 -12
VIN 1 0 AC 1
RS1 1 2 1K
RS2 6 0 1K
Q1 3 2 4 MOD1
Q2 5 6 4 MOD1
RC1 7 3 10K
RC2 7 5 10K
RE 4 8 10K
.MODEL MOD1 NPN BF=50 VAF=50 IS=1.E-12 RB=100 CJC=.5PF TF=.6NS
.TF V(5) VIN
.AC DEC 10 1 100MEG
.END
```

Hands-On With Multisim Live

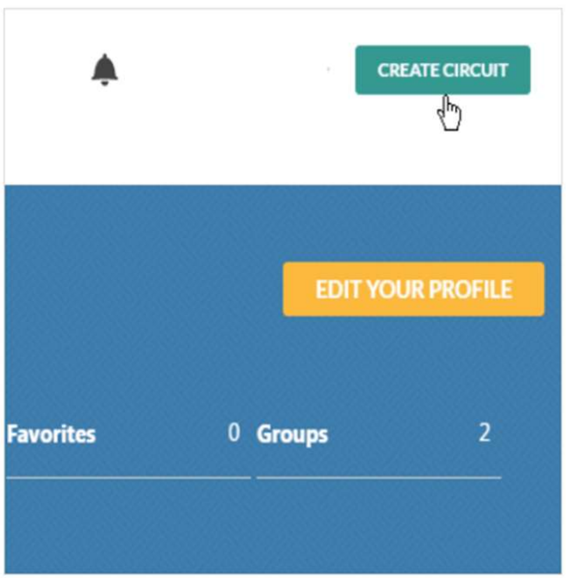
- Multisim Live is an online circuit simulation package developed by National Instruments (NI).
- Digilent, a company owned by NI, operates the Multisim Live website.
- A free trial of the online circuit simulation package allows 4 circuits with 5 components to be verified with the simulator.

Digilent is now part of Emerson.

https://www.multisim.com/

Hands-On With Multisim Live . . .

This mini lab activity will Get You Started on exploring the Multisim Live online circuit simulator platform. The circuit that will be explored is an RC Timer.

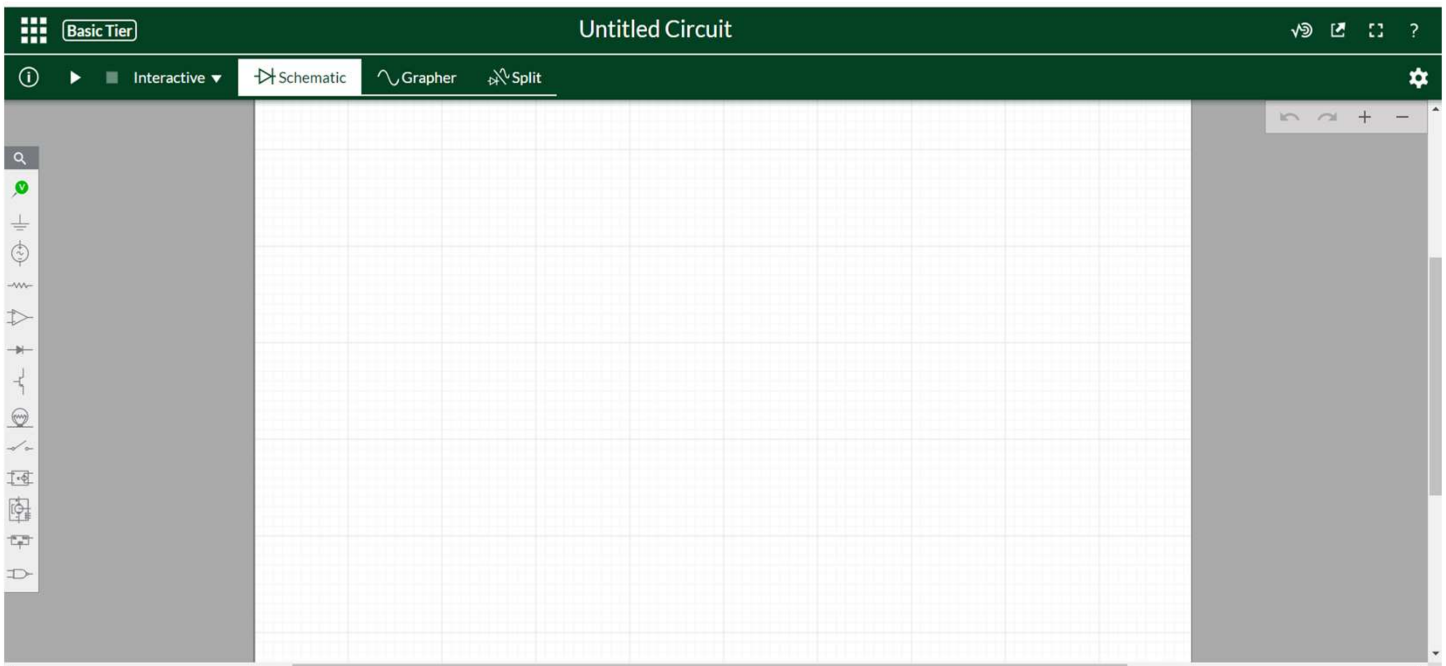


Create your first Multisim Live circuit

- Create an account and sign in
- Start a new design with the Create Circuit button

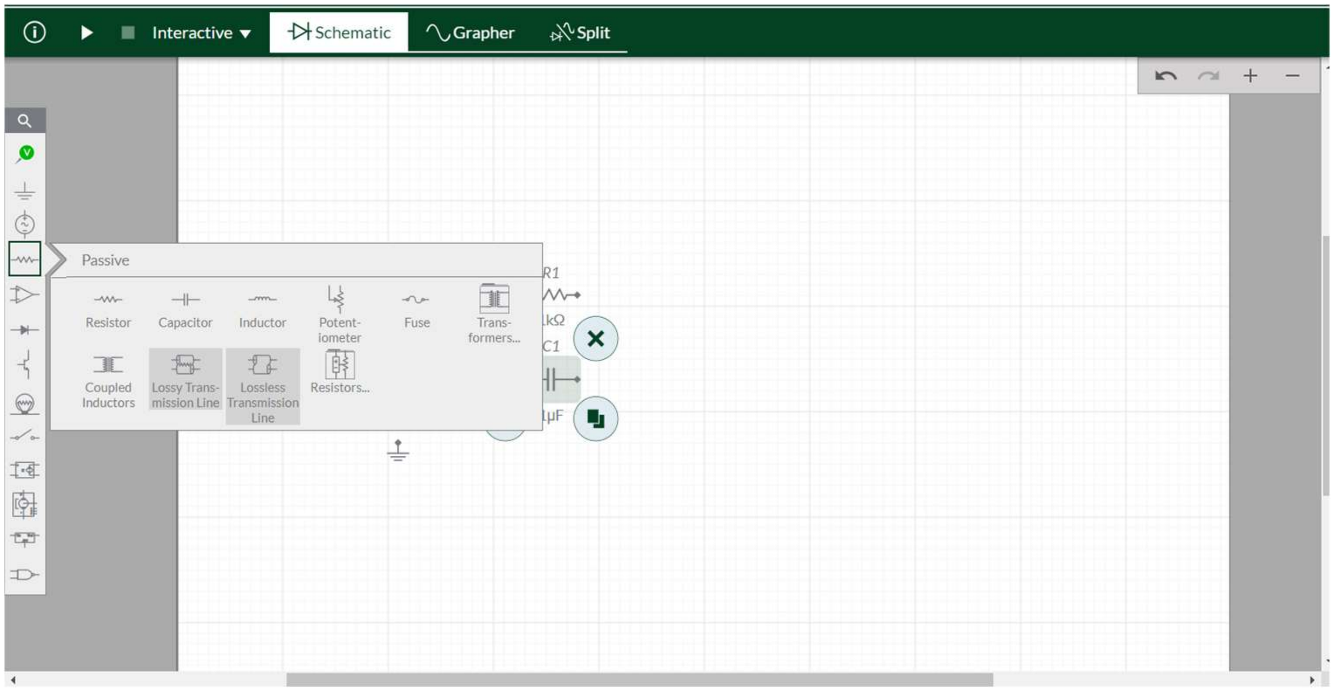
Hands-On With Multisim Live . . .

Upon creating an account, click the Create Circuit button to enter the online circuit simulation environment.



Hands-On With Multisim Live . . .

Select and place the electrical components from the Parts Menu onto the virtual bench.

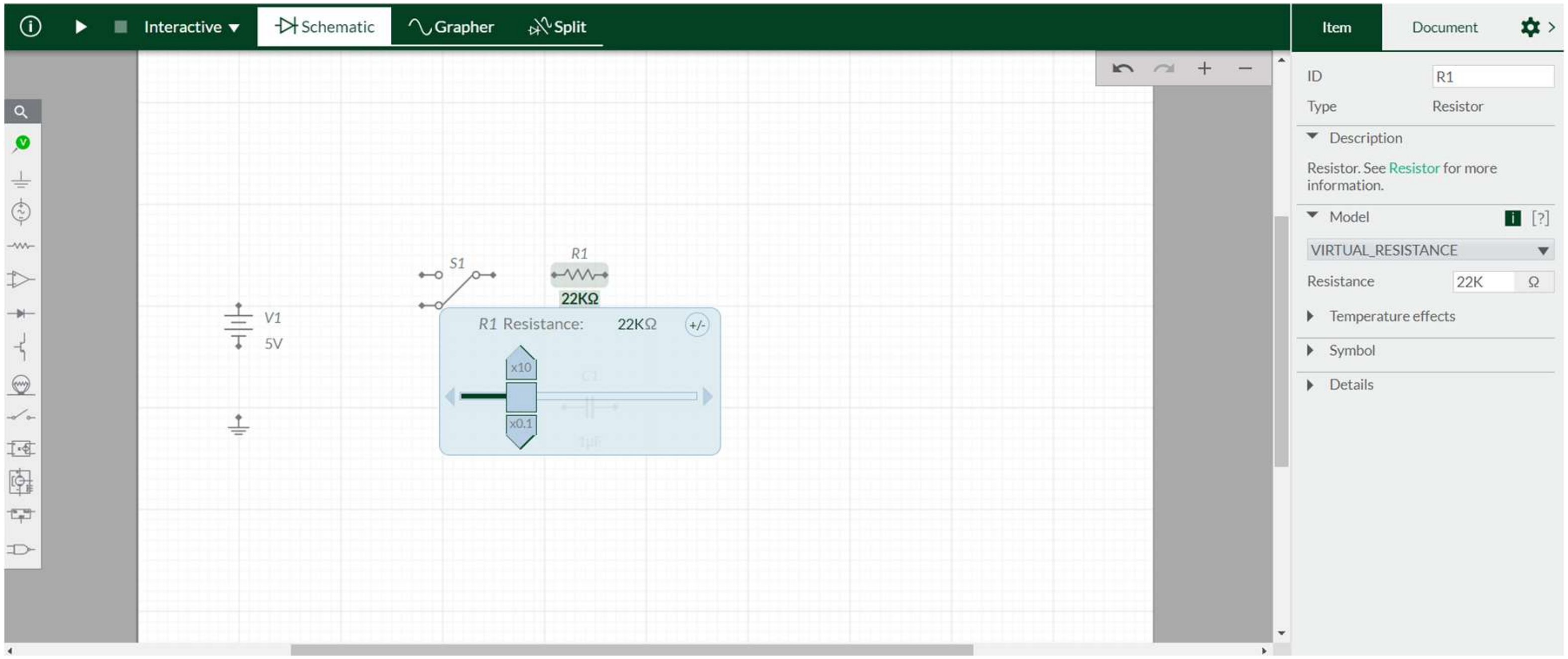


CEC Continuing Education Center

DigiKey

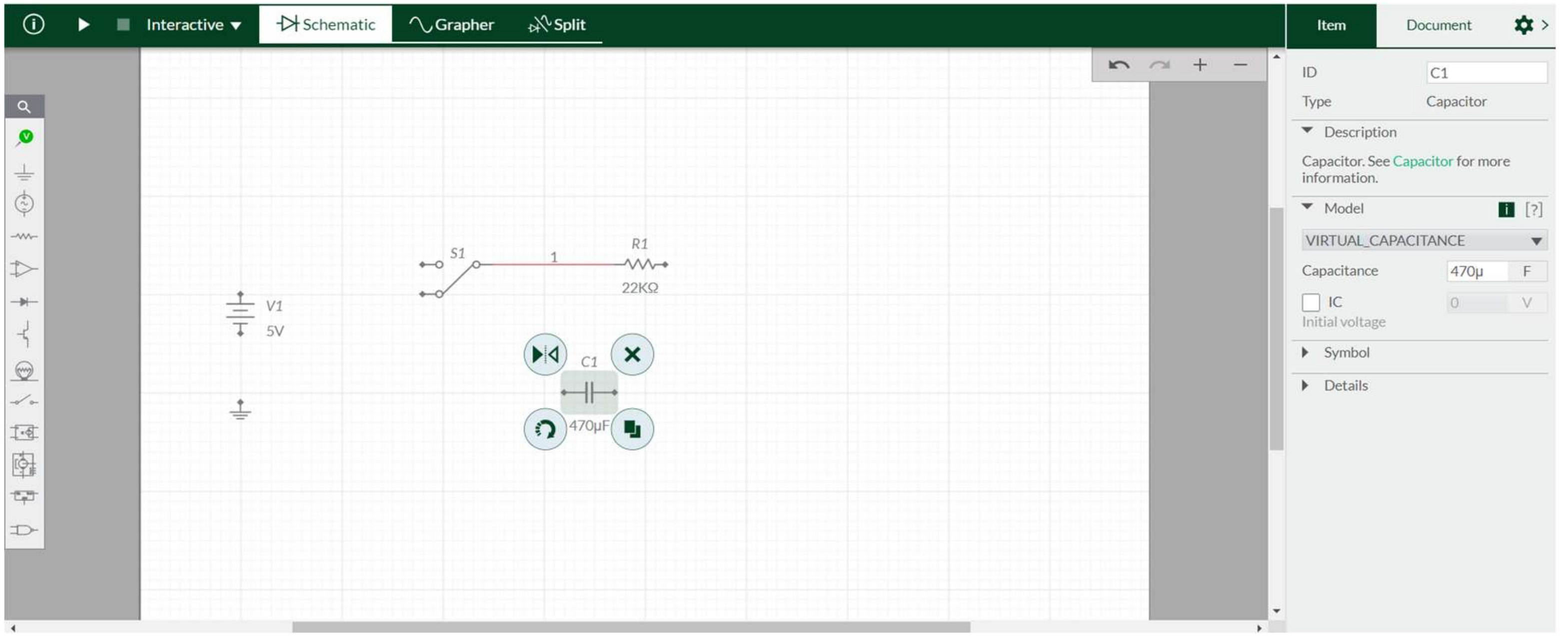
Hands-On With Multisim Live . . .

Add values to resistor and capacitor components.



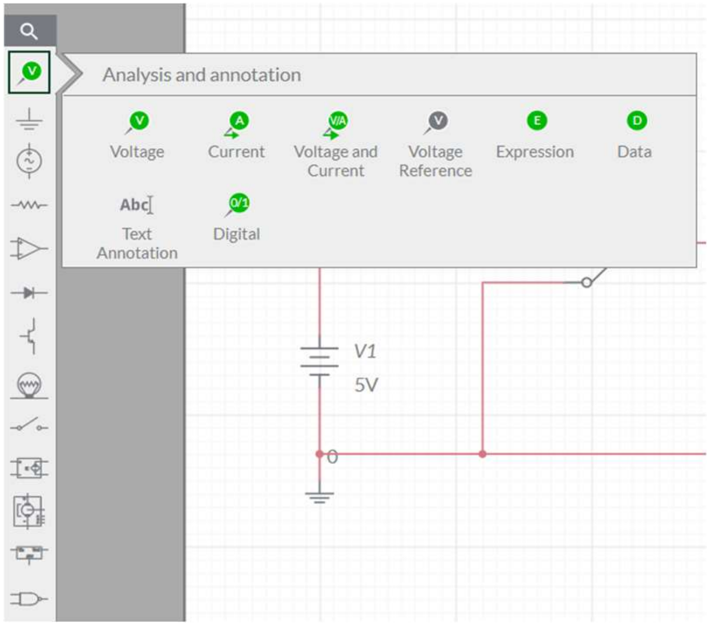
Hands-On With Multisim Live . . .

With the wiring tool, wire the electrical and electronic components to build the RC Timer circuit.



Hands-On With Multisim Live . . .

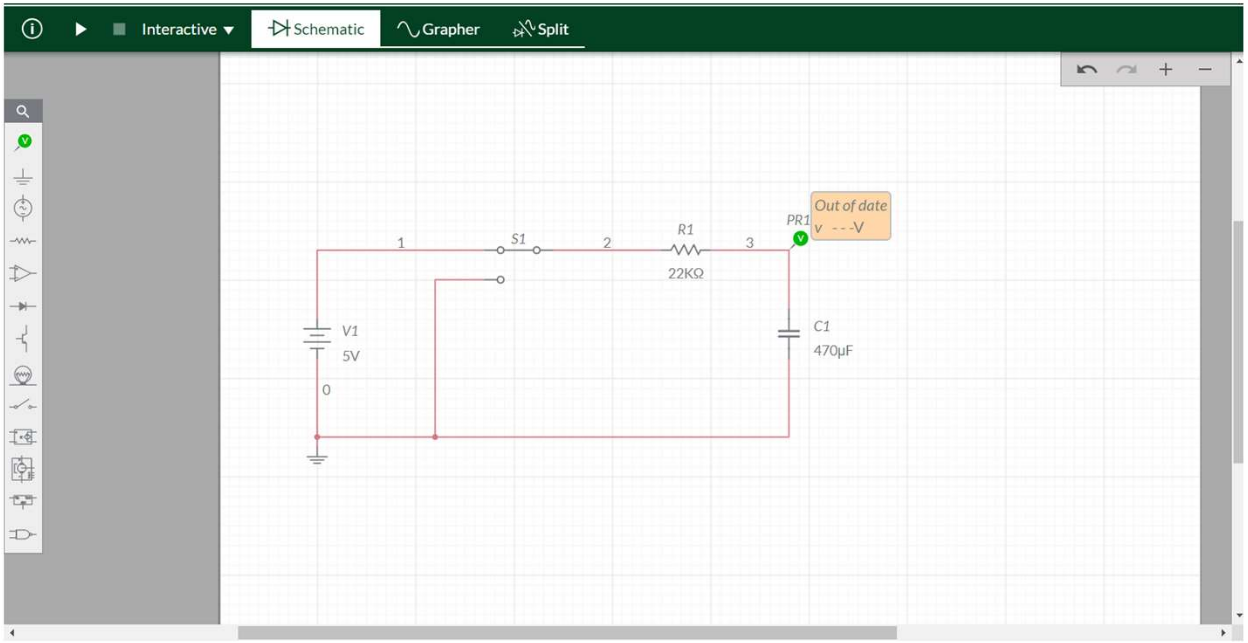
With the RC Timer circuit completed, add a voltage probe to monitor the charging and discharging state of the C1(470µF) capacitor.



Hands-On With Multisim Live . . .

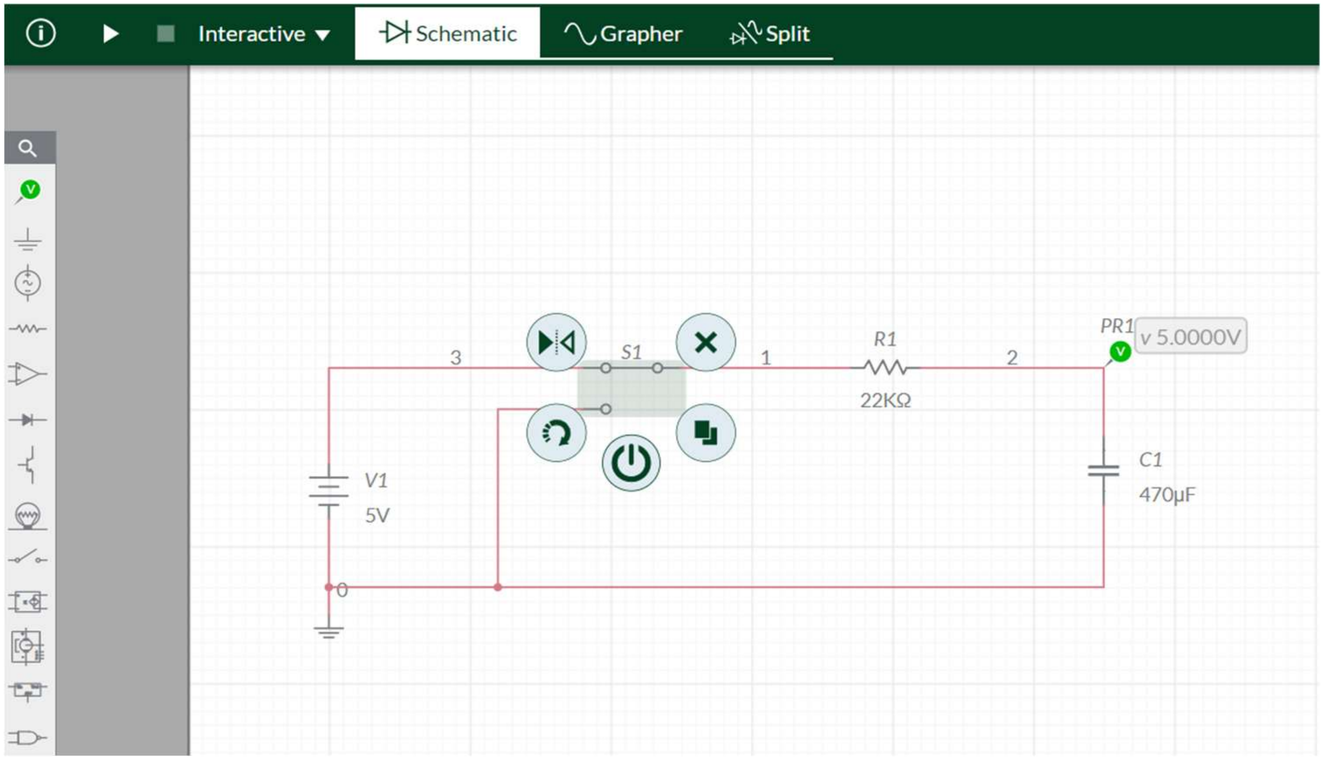
A simulation session can be initiated by clicking the play button.

Click the Play button. The simulation will run in Interactive Mode!



Hands-On With Multisim Live . . .

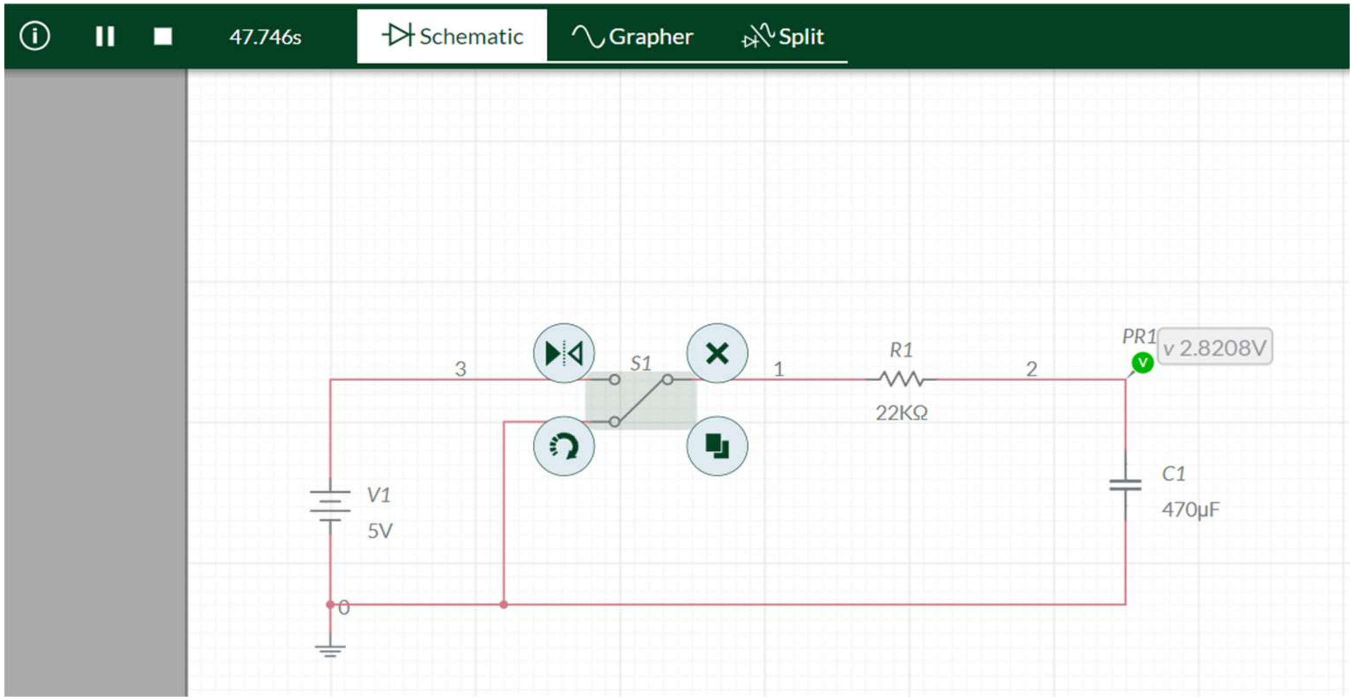
Place the selector switch in the Charge position.



The capacitor has charged up to the supply voltage V1

Hands-On With Multisim Live . . .

Place the selector switch in the Discharge position.



The capacitor is discharging.

Question 3

Who created the SPICE software?

a) Mike Engelhardt
b) Dave Patterson
c) Donald O Pederson
d) Forrest Mims

Hands-On With QSPICE

- Qorvo's SPICE (QSPICE) is the next generation of analog and mixed-signal simulation software for Electrical and Electronic engineers and designers.
- Mike Engelhardt, the creator of LTspice, created QSPICE to allow Electrical and Electronic engineers and designers the power and flexibility to evaluate their designs with high confidence.
- The simulation software is free and it supports digital logic designs.

https://www.qorvo.com/design-hub/design-tools/interactive/qspice

CEC Continuing Education Center

DigiKey

Hands-On With QSPICE: . . .

This mini lab activity will Get You Started on exploring the QSPICE online circuit simulator platform. The circuit that will be explored is an RC Timer.

- Download the software from the Qorvo website.
- Install the software on your development machine or system.

Click Here to get QSPICE!

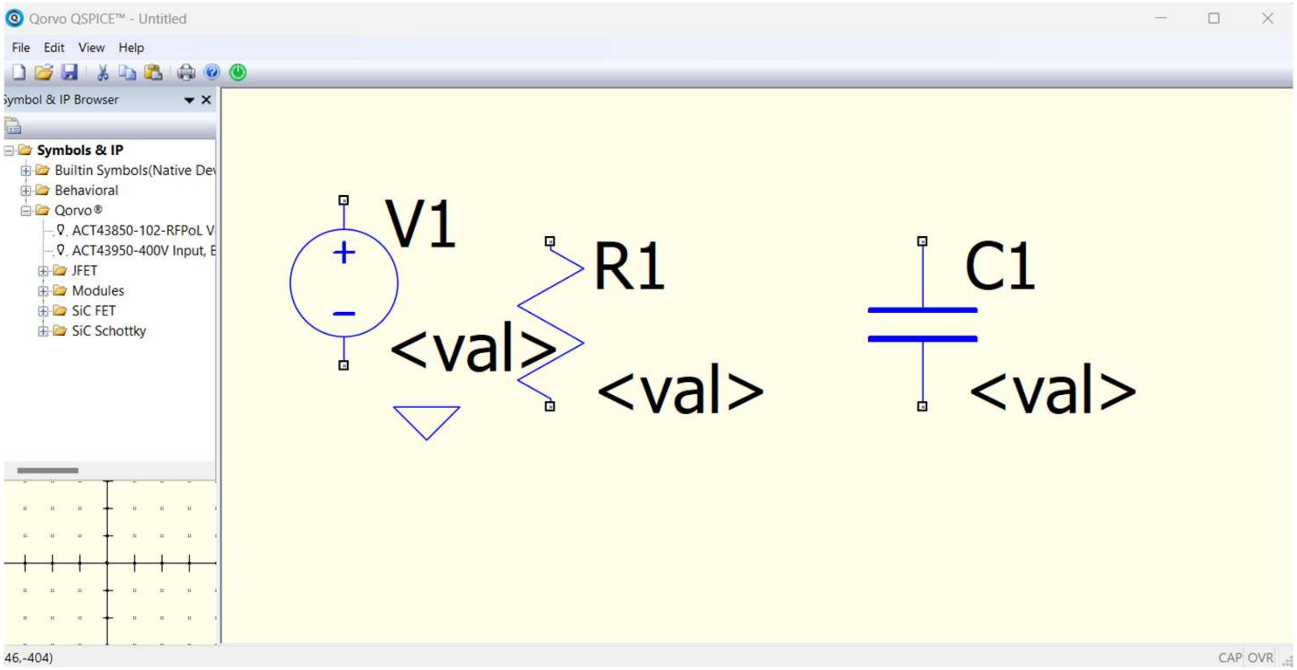
Hands-On With QSPICE: . . .

Open the software to view the development environment.

Hands-On With QSPICE: . . .

Place the following components onto the screen.
Press the following keys on your keyboard:

<G>: Ground
<R>: Resistor
<C>: Capacitor
<V>: Voltage



Question 4

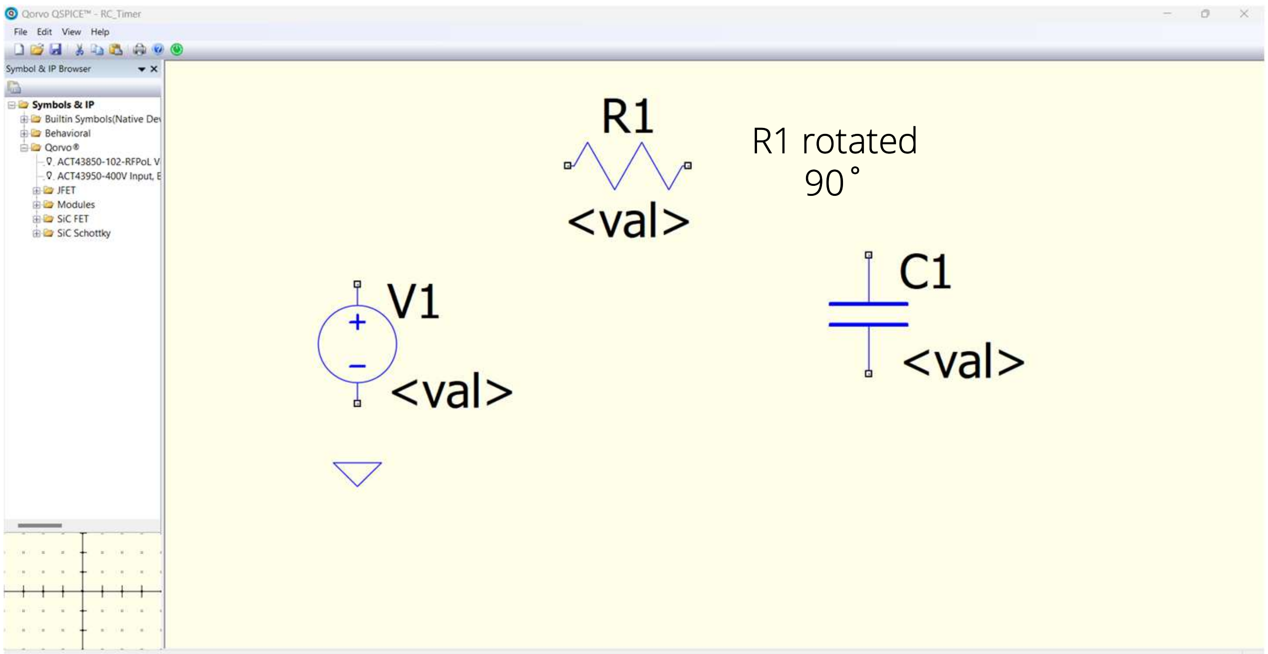
What method is used in QSPICE to add components to the circuit design screen?

- **a) pressing designated letter keys on a keyboard**
- **b) selecting components from a library**
- **c) creating a script**
- **d) none of the above**

Hands-On With QSPICE: . . .

To rotate components, right-click with the mouse, select “Orientation”, “↑ Rotated 90˚”.

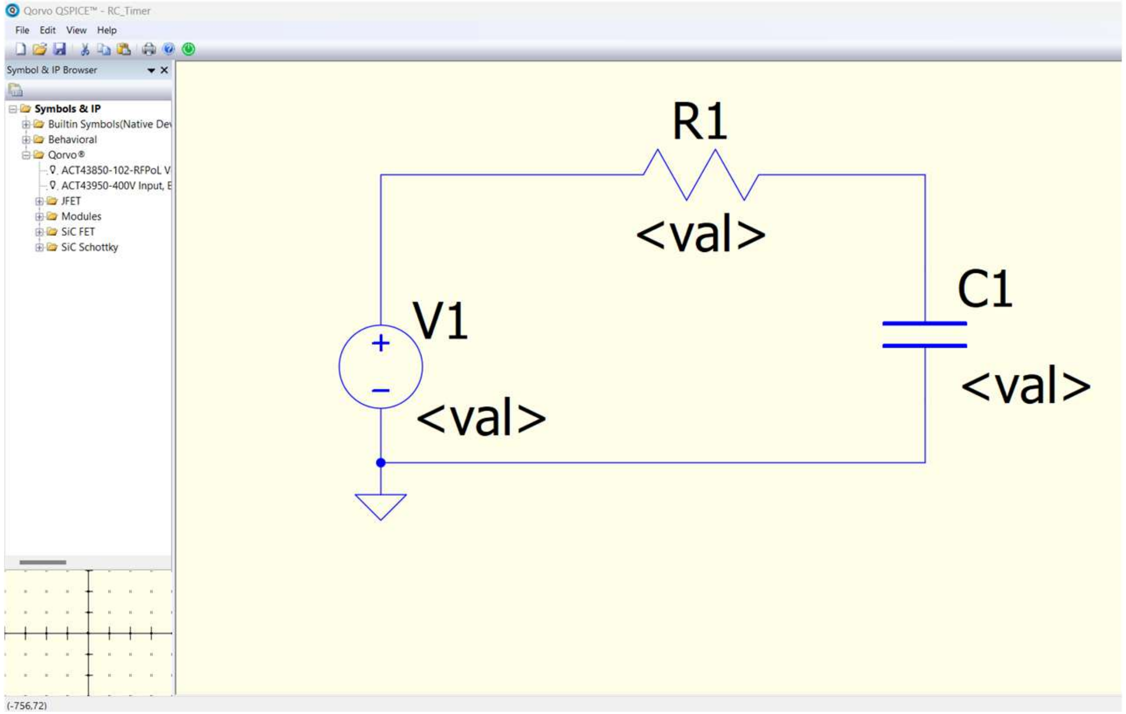
After each task, press the ESC key!



Hands-On With QSPICE: . . .

To wire the circuit, right-click with the mouse, and select "Draw the wire".

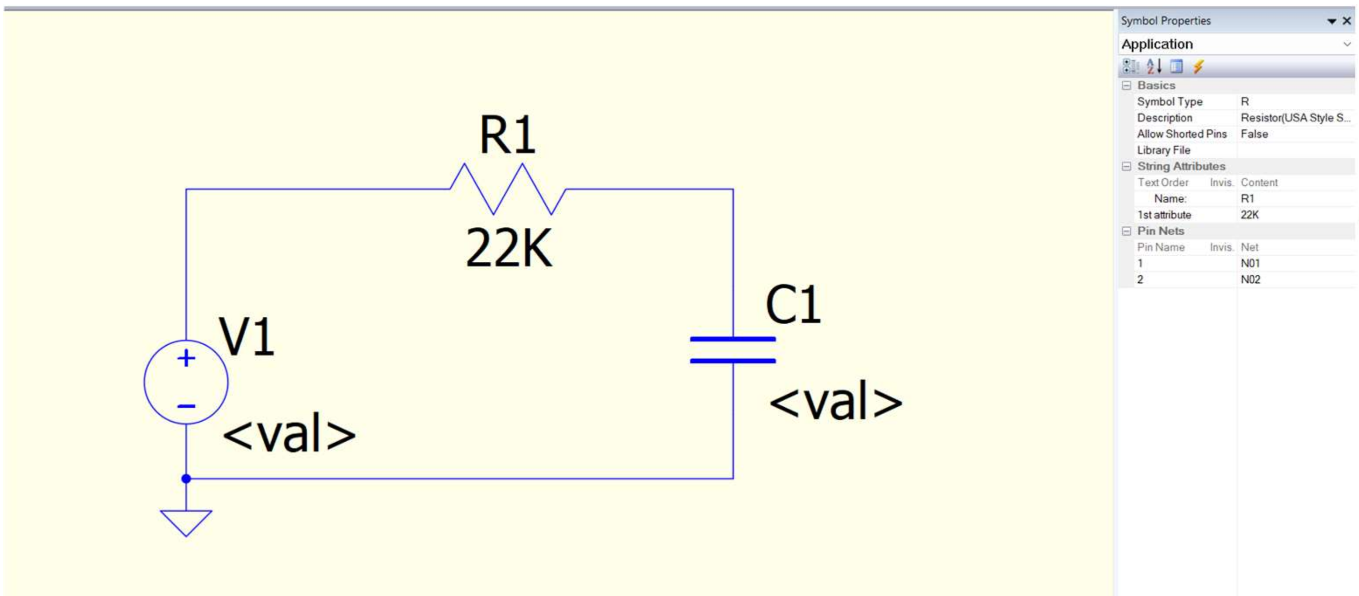
After each task, press the ESC key!



Hands-On With QSPICE: . . .

Select and right-click with the mouse. Click “Show Symbol Properties” with the mouse. Edit/change component value.

Repeat the task for the Capacitor! The ground 1st attribute will be 0V

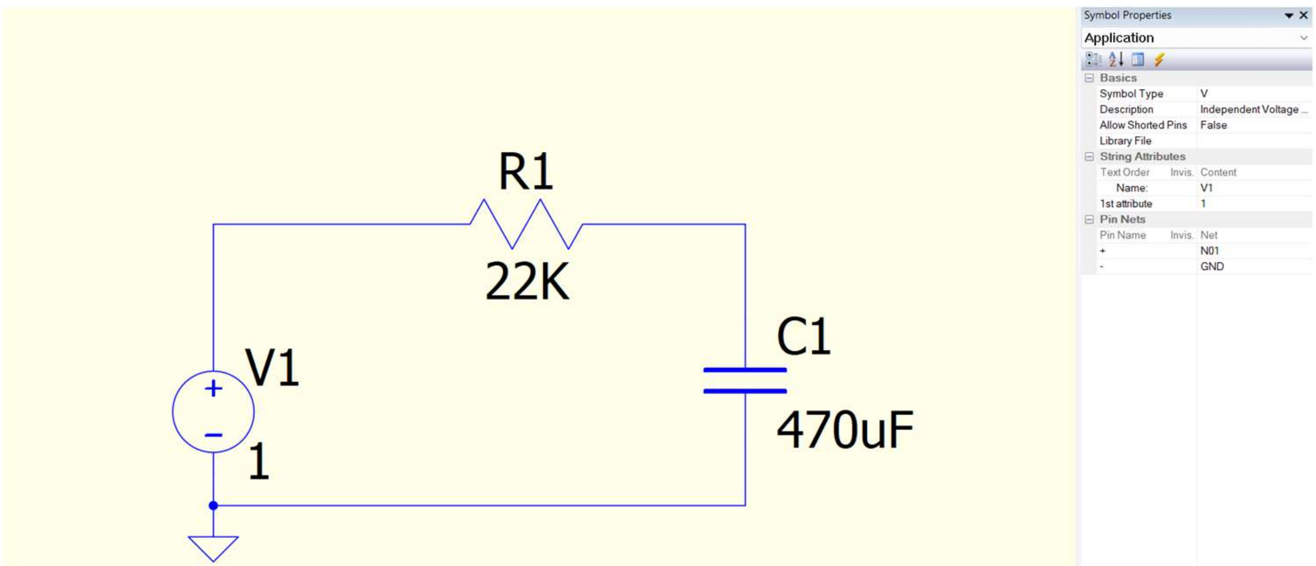


Change component value (1st attribute) here!

Hands-On With QSPICE: . . .

Select and right-click with the mouse the V1 component. Click “Show Symbol Properties” with the mouse. Edit/change component value.

Repeat the task for the Capacitor! The ground 1st attribute will be 1. This value is the supply voltage value

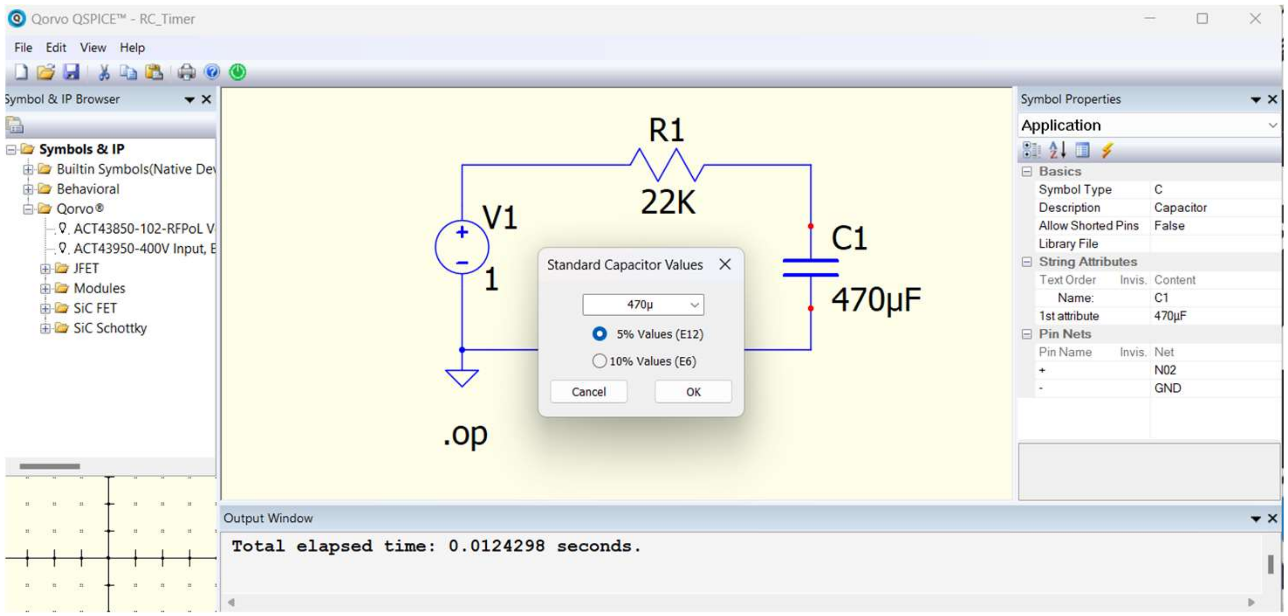


Change component value (1st attribute) here!

Hands-On With QSPICE: . . .

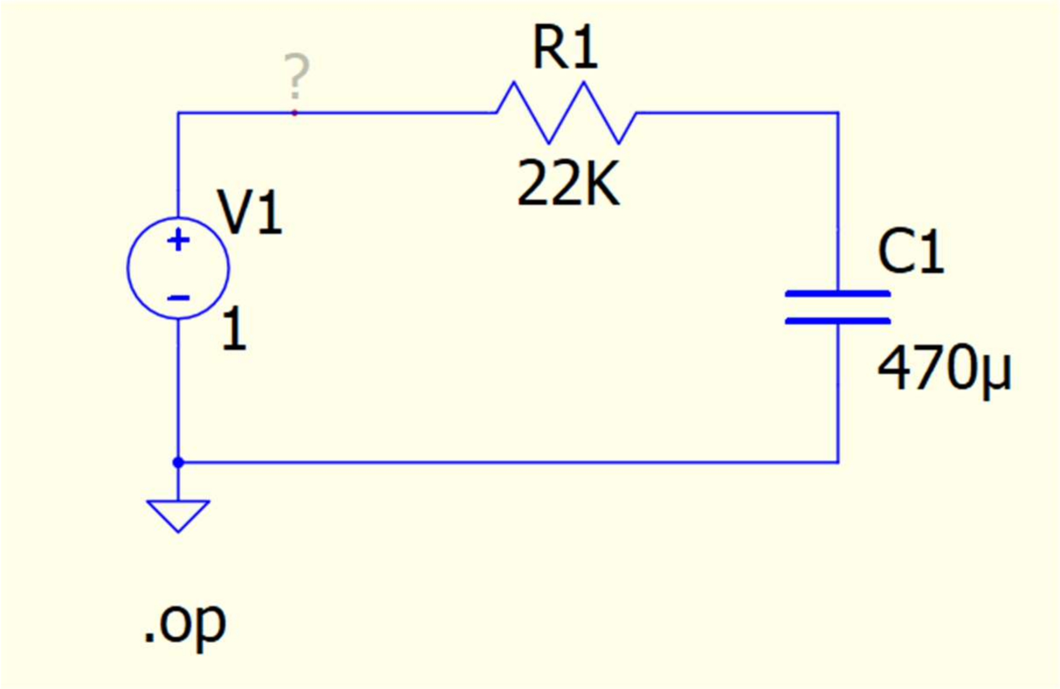
Select and right-click with the mouse the V1 component. Click "Standard Value" with the mouse. Select the 5% for the capacitor to obtain 470u for the simulation session. Click the OK button.

Repeat the task for the 22K resistor.



Hands-On With QSPICE: . . .

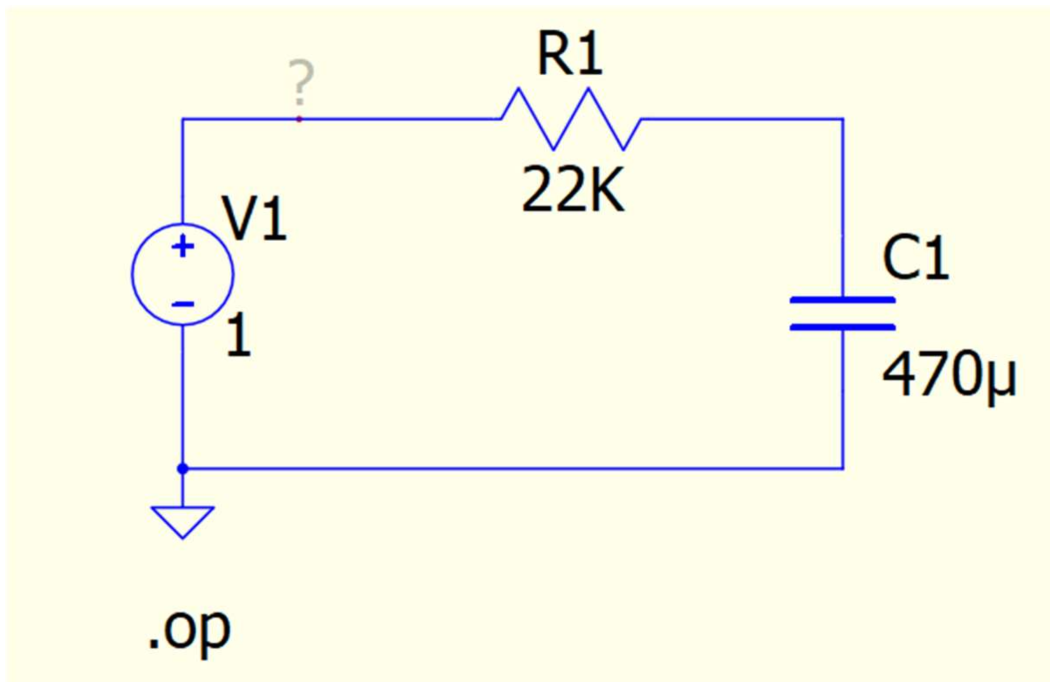
The new value will be populated/displayed on the circuit model.



Hands-On With QSPICE: . . .

Type ".op" for DC Operating Point simulation. To run a simulation session, click the power button with the mouse.

Click on the node (wire) between the V1 supply source and the resistor. The plot of the node will be visible in the Waveform Viewer Screen.



Hands-On With QSPICE: . . .

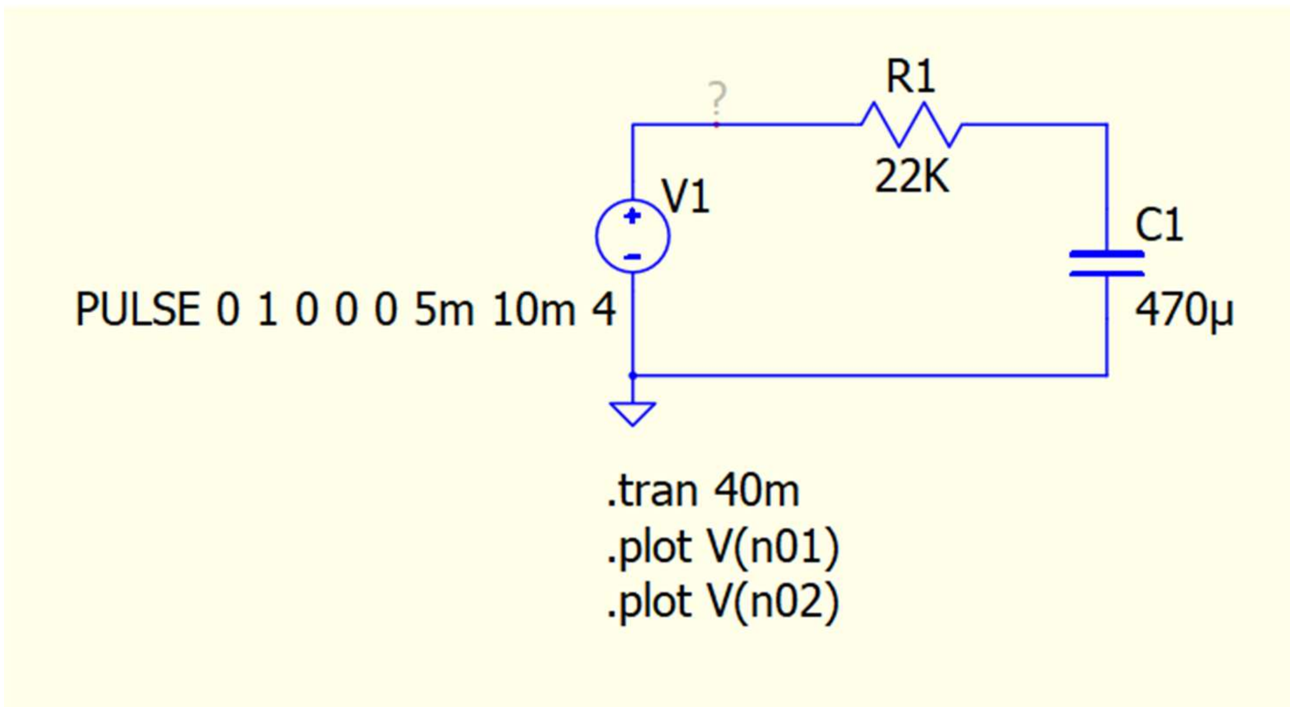
Waveform Viewer Screen.

The first Waveform is of the supply voltage. The second plot shows the voltage across C1.

Hands-On With QSPICE: . . .

Type the new circuit simulation parameters for injecting a Pulse into the RC Timer Circuit.

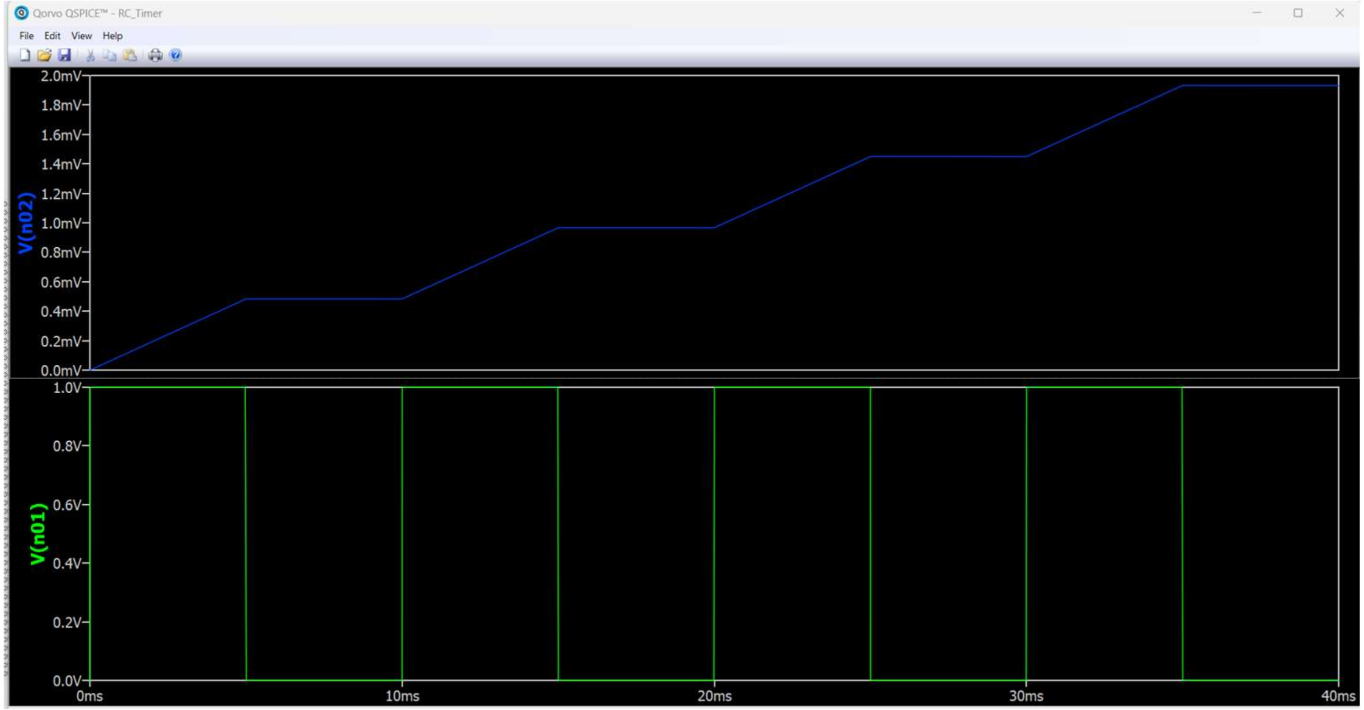
The PULSE Voltage source will apply a transient waveform. The capacitor charging will be displayed as discrete voltage values displayed in the Waveform Viewer.



Hands-On With QSPICE: . . .

Waveform Viewer Screen.

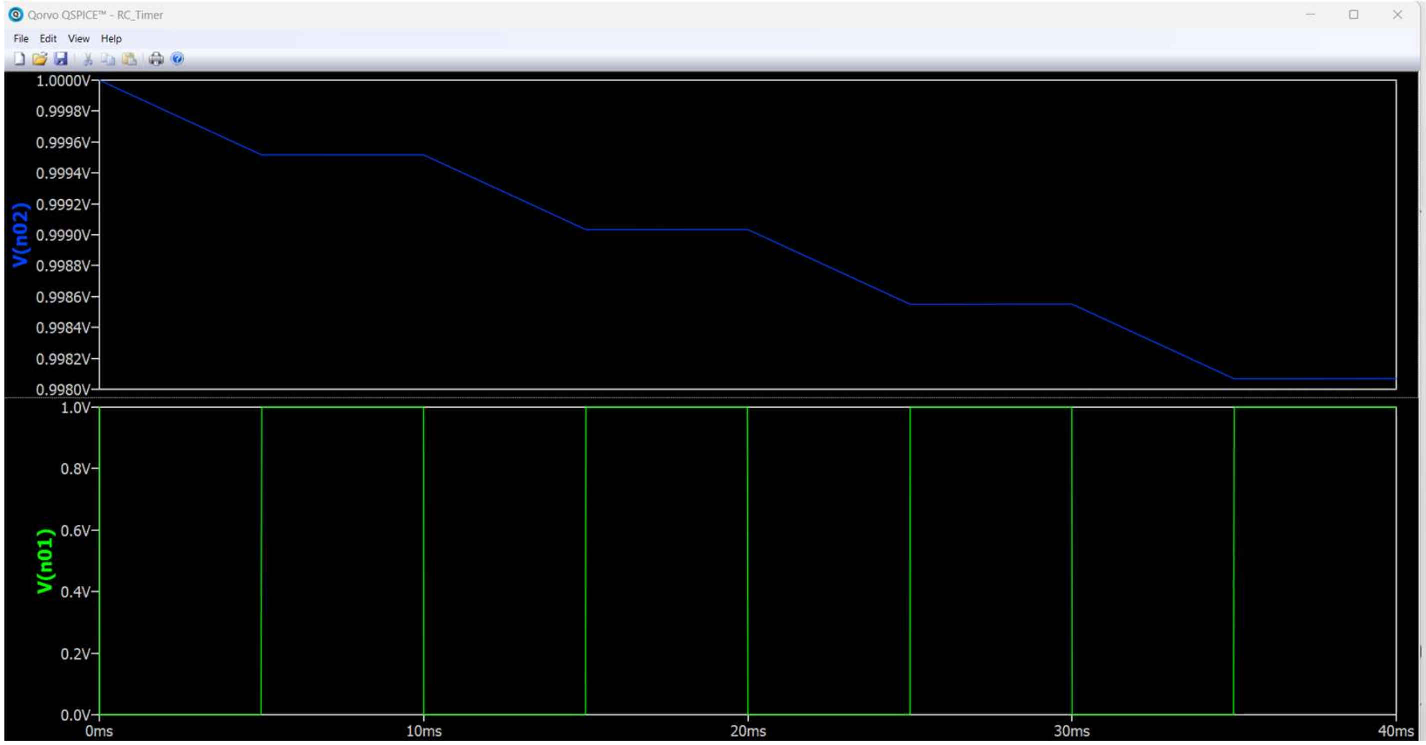
The Waveform Viewer displays the capacitor(C1) being charged.



Hands-On With QSPICE: . . .

Waveform Viewer Screen.

The Waveform Viewer displays the capacitor(C1) being discharged.



Hands-On With QSPICE: . . .

Definition of the PULSE Voltage Source attributes

Explore the PULSE Voltage Source by changing the attributes. Observe the results displayed on the Waveform Viewer!

PULSE 1 0 0 0 0 5m 10m 4

PULSE	1	0	0	0	0	5m	10m	4
	Voff	Von	Tdelay	Trise	Tfall	Ton	Tperiod	Ncycles

Question 5

In reviewing slide 45, what PULSE attribute has a value of 10m?

a) Trise
b) Ton
c) Tperiod
d) Tdelay

Thank you for attending

Please consider the resources below:

Pederson, D. O. (1984). A historical review of circuit simulation. *IEEE Transactions On Circuits and Systems, 31(*1*),* 103 – 111. https://web.engr.oregonstate.edu/~karti/ece521/dop.pdf

CEC Continuing Education Center

DesignNews

Thank You

Sponsored by

DigiKey

informa markets