Circuit Simulator User Manual

Table Of Contents

Part 1: Interface

Simulator
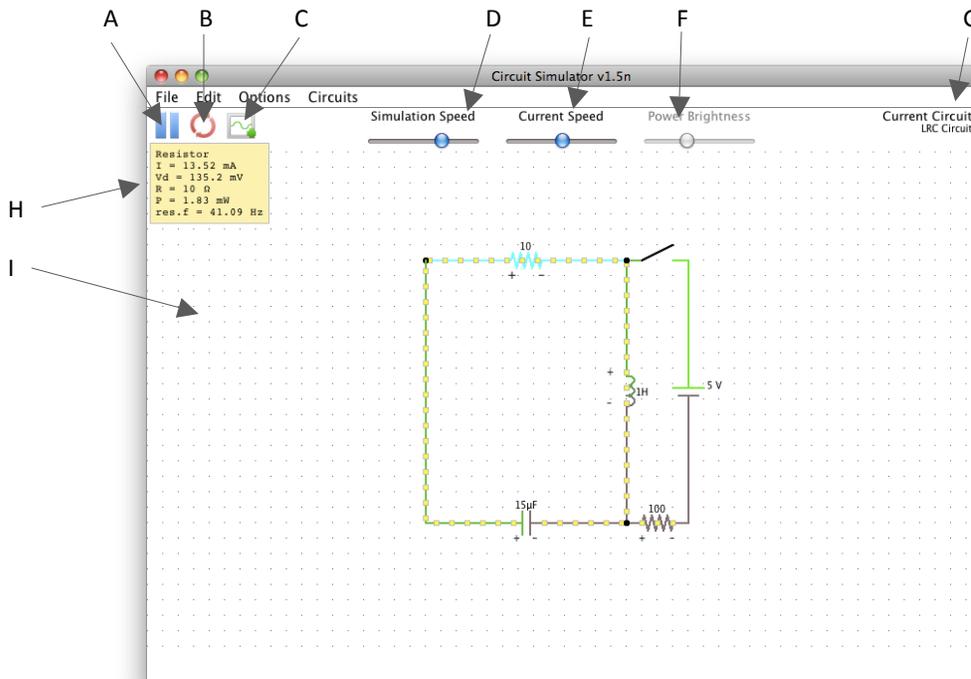
A. Pause / resume simulation
B. Reset simulation
C. Create new scope window
D. Adjust simulation speed
E. Adjust current speed
F. Adjust power brightness
G. Name of currently loaded circuit
H. Instantaneous values of selected element
I. Circuit board

Oscilloscope
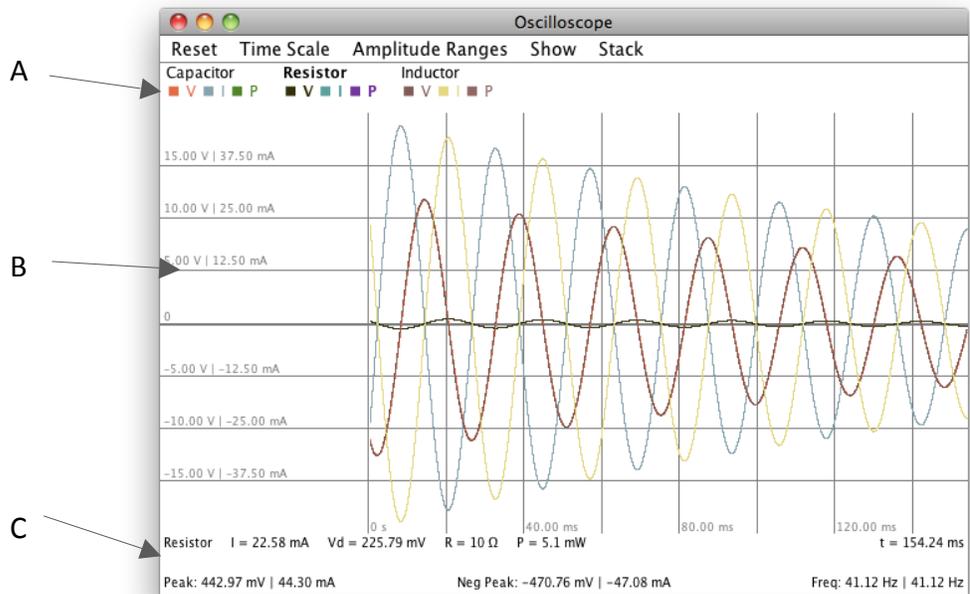
A. Displayed elements (selected element in bold)
B. Waveforms
C. Instantaneous values of selected element

Part 2: Adding / Moving Elements

Adding Elements

To add an element, right click on the circuit board. The following menu will appear.
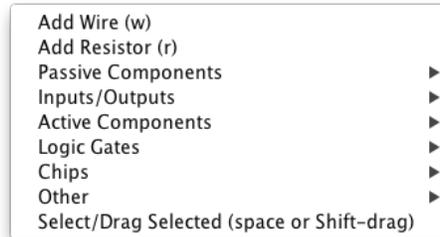
To add an element to the circuit, select the desired element from one of the submenus, then click and drag on the circuit board to place the element. When drawing, elements snap to the grid shown on the circuit board. For a smaller grid, check the Small Grid option in the Options menu.

Available Elements:

Passive Components	Capacitor	SPDT Switch	Transmission Line
	Inductor	Potentiometer	Relay
	Switch	Transformer	Memristor
	Push Switch	Tapped Transformer	Spark Gap

Inputs / Outputs	Ground	Analog Output	Antenna
	Voltage Source	Logic Input/Output	Current Source
	A/C Source	Clock	LED
	Square Wave	A/C Sweep	Lamp

Active Components	Diode	MOSFET	Tunnel Diode
	Zener Diode	JFET	Triode
	Bipolar Transistor	Analog Switch	CCII
	Op Amp	SCR	

Logic Gates	Inverter	NOR	OR
	NAND	AND	XOR

Chips	D Flip Flop	Phase Comparator	DAC
	JK Flip Flop	Counter	ADC
	7 Segment LED	Decade Counter	Latch
	VCO	555 Timer	

*Note***:** Some elements (such as resistors, capacitors, and inductors) are displayed with +/- icons next to them. This is the element's oscilloscope polarity, and affects the sign of the waveform when the element's voltage or current is viewed in the oscilloscope. The sign of the waveform follows the passive sign convention. (Current entering at the + terminal is considered positive and a voltage drop from the + to the – terminal is considered positive). When placing an element, the first click will place the + terminal and the second the – terminal.

Moving Elements

To move an element that has already been placed, right click on the circuit board and choose Select / Drag Selected. Then click and drag on an element to move it. Alternatively, instead of choosing the Select / Drag Selected menu option, simply hold the shift key while clicking and dragging on an element.

To move only one post of an element, right click on the circuit board and choose Drag Post from the Other submenu. Then click on drag on an element's post to move it. Alternatively, hold the control key (command on a Mac) while clicking and dragging on an element's post.

Also in the Other submenu are the options to Drag All, Drag Row, and Drag Column. The Drag All option moves all elements in the circuit. Drag Row and Drag Column move all elements that are vertically and horizontally aligned with the element being dragged, respectively.

Part 3: Simulation

As soon as elements are placed on the circuit board, the simulation will start running. To pause simulation, click the Pause Simulation button at the top left of the simulator window. Click this button again to resume the simulation.

The Reset Simulation button sets the simulation time back to 0, clears any energy stored in elements like inductors or capacitors, and starts the simulation again.

To adjust the speed of the simulation, use the slider labeled Simulation Speed.

Show Current, Voltage, Power

By default, current and voltage are shown on the circuit as part of the simulation. Current is represented by the moving yellow dots. The speed of the dots can be adjusted with the Current Speed slider at the top of the window. Adjusting the slider is purely a visual change; it has no effect on the actual simulation. Elements at a positive voltage are drawn in green and elements at a negative voltage are drawn in red. In both cases, brighter colors represent higher voltage magnitudes. Current and voltage display can both be turned off by unchecking Show Current and Show Voltage in the Options menu. Also, power can be displayed on the circuit by checking Show Power in the Options menu. Note that voltage and power cannot both be displayed at the same time.

Like voltage, power is shown in green and red, with green representing power generated, red representing power dissipated, and brighter colors representing higher magnitudes of power. Also, the brightness of the colors used to display power can be adjusted using the Power Brightness slider at the top of the window. Adjusting the slider is purely a visual change; it has no effect on the actual simulation.

Part 4: Oscilloscope

Adding Elements to Scope

The simulator also includes an oscilloscope feature to visualize circuit behavior. To view an element in the scope, right click on the element and select the View in Scope option. If a scope window is already open, the element will be added to that window. Otherwise, a new scope window will be created and the element added to it. All the elements currently displayed in a scope window are listed at the top of the window. To remove an element from a scope, right click on the element's name and select the Remove From Scope menu option.

Multiple scope windows are also supported. The View in Scope menu option adds an element to the scope window that most recently had focus. Thus, to add an element to a specific scope window, click on the desired scope window to bring it into focus, then right click on the element and select View in Scope. An empty scope window can be created by clicking the New Scope button near the top left of the simulator window. The same element can be added to multiple scope windows.

Changing Displayed Values

The scope can display voltage, current, and power. These can be shown/hidden for each element independently, allowing for example one element's current to be compared to another's voltage.   To control which values are shown for an element, right click on that element's name at the top of the scope window. Then check/uncheck the Show Voltage/Current/Power menu options.

Under each element's name are the colors in which its voltage, current, and power waveforms are displayed. These colors are randomly set when the element is added to the scope, so sometimes a color is too light to be clearly visible or is too similar to another waveform's color to easily distinguish the two. To manually set the color for an element's waveform, right click on the element's name and select the Voltage/Current/Power Color option from the Change Colors submenu.

Viewing Instantaneous Values

To view instantaneous values for a specific element in the scope, left click on that element's name. That element's name will be shown in bold and its instantaneous values shown below the waveforms display. Also at the bottom of the scope window is shown the positive and negative peak values and resonant frequency for the selected

element. By default, only positive peak values are shown. The others can be shown/hidden by checking/unchecking Negative Peak Value and Frequency in the Show menu.

Stacking Scopes

In some cases, especially with logic circuits, it is more useful to view the scopes "stacked" one on top of another rather than all on the same coordinate space. To switch between stacked and normal views, check/uncheck the Stack All option in the Stack menu.

Stacked Scopes

Adjusting Scales/Ranges

Time scale and amplitude ranges can both be adjusted up or down by a factor of 2 from the Time Scale and Amplitude Ranges menus, respectively. Voltage, current, and power ranges can all be adjusted independently or all at the same time with the All Ranges menu option. Also, amplitude ranges can be fit to the currently displayed waveforms with the Fit Ranges option.

Note: Time is referred to as scale and amplitude as range for a reason. Time scale is measured in number of time steps per horizontal pixel. The amplitude of a waveform, however, depends on the size of the scope window, as the entire window is defined to be a certain "range." Thus, resizing the scope window vertically will make the displayed waveforms taller, but resizing the scope window horizontally will make the window display a shorter/longer time span.

Alternative Scope Modes

So far, everything described has applied to the "normal" scope mode: voltage/current/power versus time. There are two other modes available though: current versus voltage (I vs. V) and X versus Y. To switch modes, select the desired mode from the Show menu.

In current versus voltage mode, elements' voltage is plotted on the X-axis and current on the Y-axis.
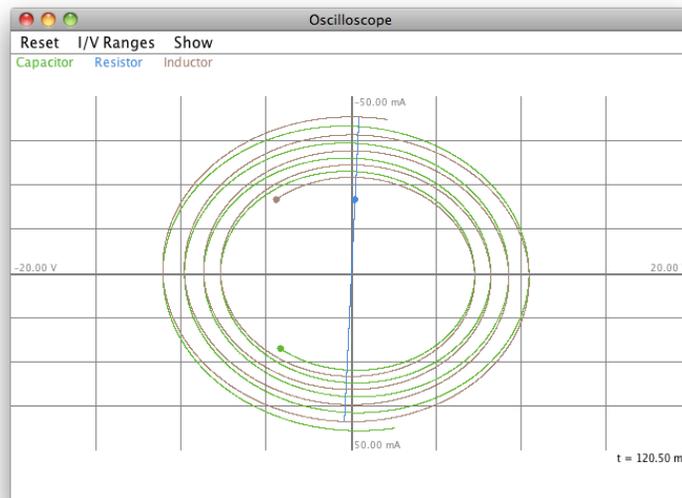
Current vs. Voltage mode

In X vs. Y mode, arbitrary values can be selected for the X-axis and Y-axis. This would allow, for example, plotting an inductor's voltage against a resistor's current.
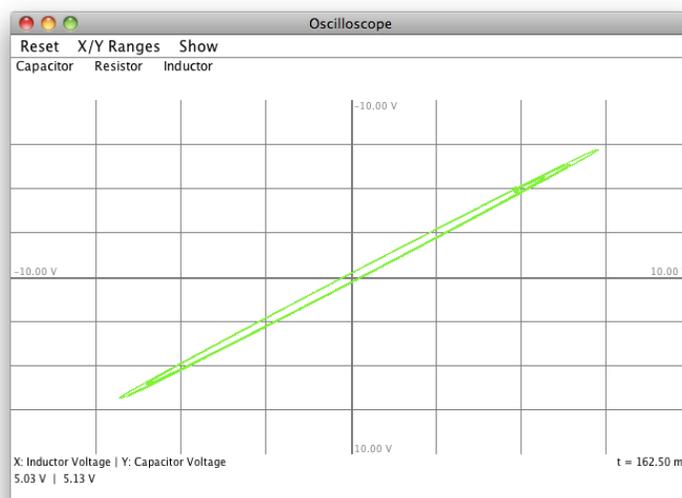
X vs. Y mode (Y = Capacitor Voltage, X = Inductor Voltage)

When X vs. Y mode is first selected, the waveform area will be blank. Before a waveform can be displayed in X vs. Y mode, the X and Y values must be selected. To do this, right click on an element's name at the top of the scope window and then select voltage, current, or power from the Select X or Select Y submenus. If an X or Y value has already been selected, selecting a new one will replace the old one and clear the plot. The currently selected X and Y values are also shown below the waveform area.

Part 5: Pre-built Circuits

The simulator comes with a number of common circuits pre-built. These can be accessed from the Circuits menu of the simulator window.