

# Basic Tutorial of CircuitLogix

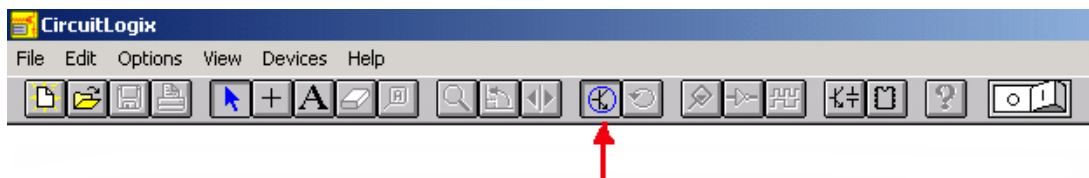
## Introduction

The purpose of this tutorial is to provide a basic overview of the electronics simulator [CircuitLogix](#), and to demonstrate some of its core functionality.

## Starting a New Design

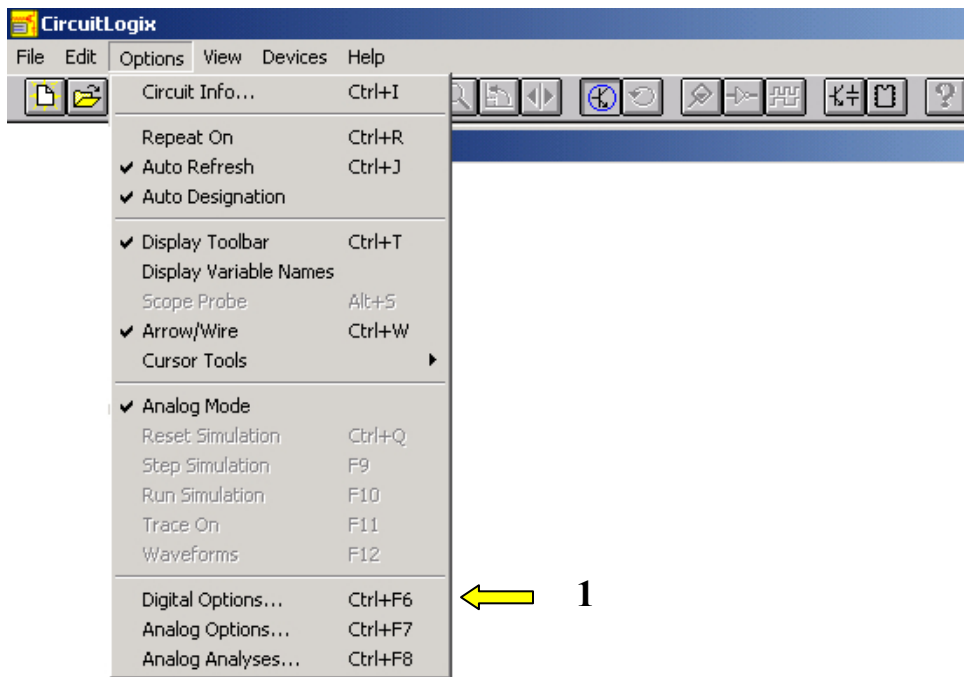
This program is user-friendly and well designed, and is intended to be a tool that you can pick up and start using. The program is intended to allow for trial and error on part of the user, and requires a minimal amount of instruction to build and test circuits. This tutorial will provide you with some pointers in order to help you get started and to avoid some common problems.

1. The first thing to observe is that CircuitLogix can be used to design and simulate both analog and digital signals. (Wherever you see "Spice" in the help system, it's referring to analog simulation.) You can build your circuit with out concern for this fact, but when you simulate it (run it), you must be sure to do so in digital mode. If you don't, you'll get an error message, probably telling you that you need a ground node "before a valid spice Netlist can be generated". To avoid this problem click on the toolbar button with the symbol for a transistor on it (shown in the next figure).



Just get in the habit of clicking on the transistor button (it turns into an AND gate) every time you start running CircuitLogix.

2. Click on the Options menu and observe the range of analyses options available.



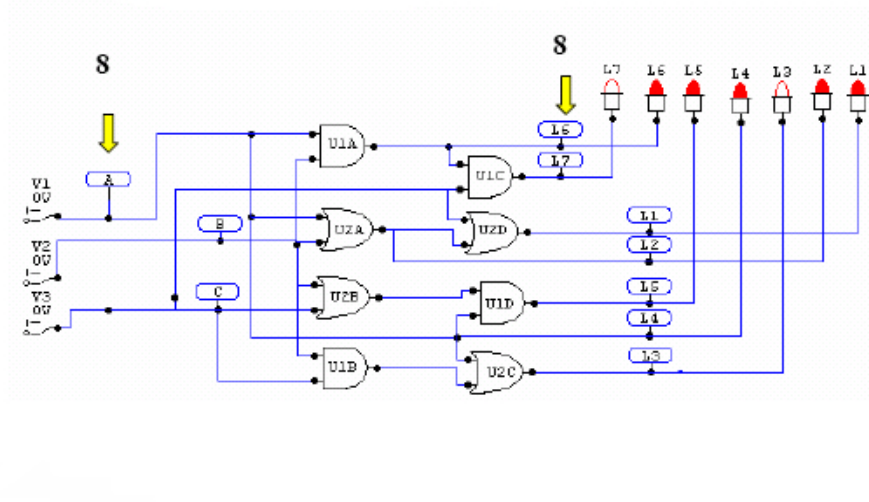
3. Use the *Devices* Library (2) to select gates, etc. to add to your circuit. Use the Wire Tool (3) to connect inputs and outputs together.



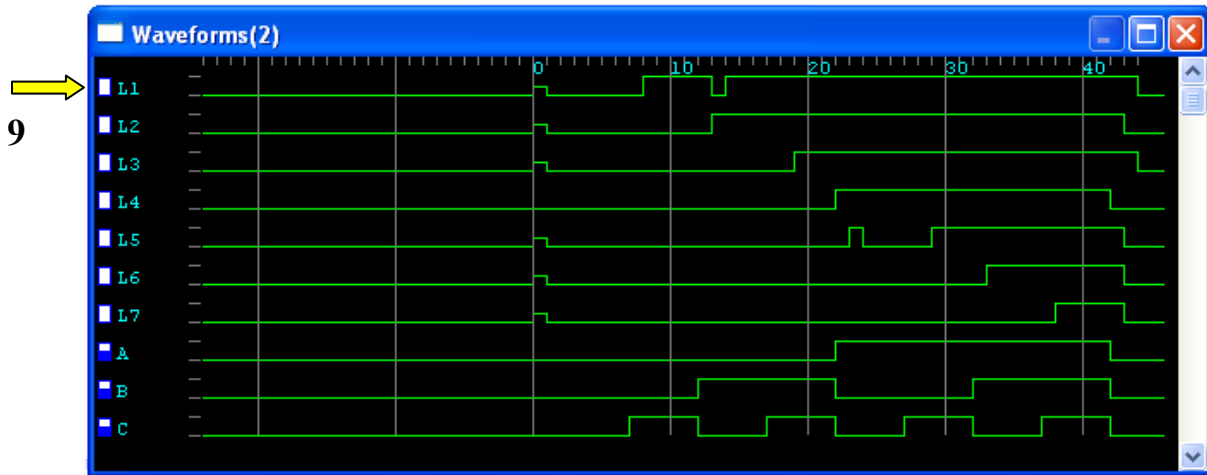
4. The most common input device is the digital logic switch, which alternates between outputting 0 and 1 each time you click on in. You can type a lower-case 's' to get one, or look for it in Hotkeys2 under the Devices menu, or select browse from the Devices menu, and select Digital -> Power.
5. You can simulate your circuit by switching the ON/OFF button (4) to the ON position. Use the logic probe (5) to display the logic value of any part of your circuit. The probe will show H ("high"), L ("low"), or Z (indeterminate) as you make it hover over various parts of the circuit.



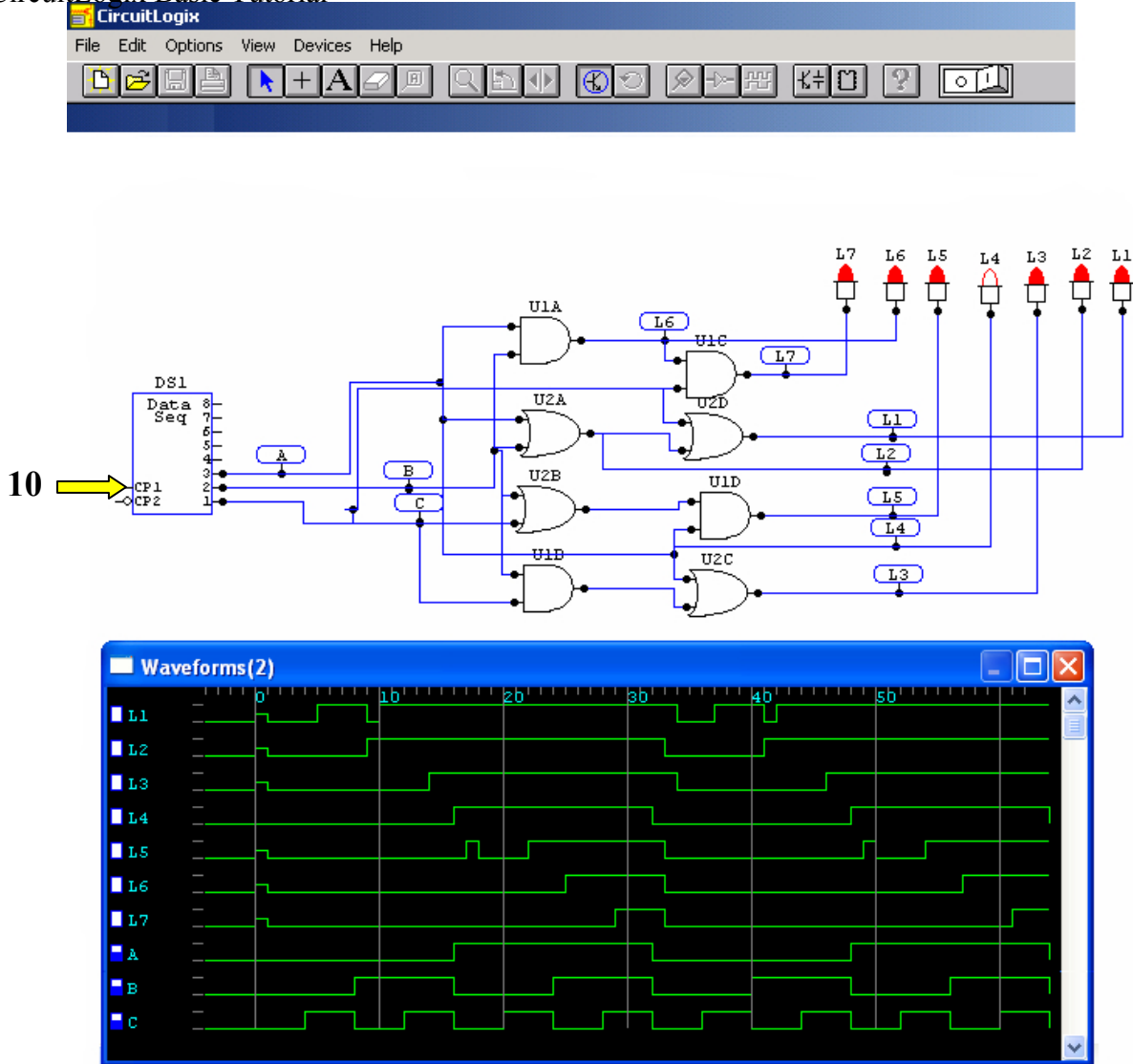
6. Sometimes, you will want to look at the waveforms for various parts of the circuit as you run your simulations. Use F12 to turn this option on (6), or press the waveform toolbar button (7). The waveforms are displayed in a separate window, so select "tile" from the Window menu to get them both to show on the screen.
7. The most common way to add traces to the waveforms window is to attach the Scope (8) tool to the wires you want to look at. Type a capital T to get a Scope tool, or select it from the Hotkeys2 menu under the Devices menu. To give meaningful names to your waveforms, double-click on the scope tool and type in the name you want to use.



8. You can move waveforms up and down by dragging their names in the waveforms window. The white boxes (9) to the left of the names are used for setting breakpoints. Breakpoints can be used to stop the simulation when the certain conditions are met. You can learn how to use breakpoints from the on-line help.



9. To generate input values automatically or to have a clock signal, you have to generate them using a *Data Sequencer (10)*. Type capital G to get one, or browse to it under Digital/Instruments under the Devices menu. Here's how to set it up so that pins 1-3 on the right side can be used as an inputs signals.



Double click on the device, and you will get an "Edit Data Sequencer" dialog box. First of all, resist the temptation to turn on the "Use External Clock" option; it generates values at the proper rate for simulation (one propagation delay per step) all by itself. The data sequencer can generate up to 8 digital input signals with up to 32 different sequences. Each input sequence is stored in an address. For example, Edit the first 8 rows of the "Address - Data" box so that addresses 1, 2, 3, 4 contain 00 (hexadecimal), and addresses 5, 6, 7, 8 contain 01. Set the Start and Stop Addresses to 1 and 8 respectively.

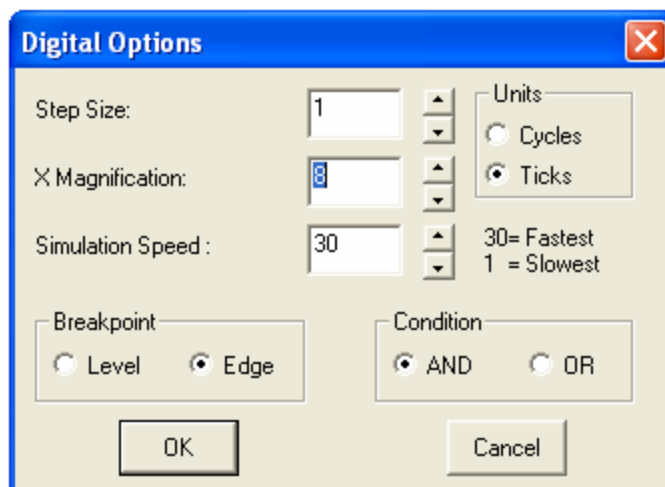
When the simulation runs, the sequencer will output "000" on pins Nos. 1-3 for four propagation delays, then will output "001" on the same pins for another four propagation delays, and so on. The process repeats as long as the simulation runs. For example, connect scopes to the signals you want to check out, and display their respective waveforms in the waveform window.

10. Use the Digital Options dialog box to control the size of a step when running the simulation in single step mode, to set the conditions for break points and to set the simulation speed. Choose Simulation > Digital to display the dialog box.

You can define the Step Size in either ticks or cycles. A cycle always consists of 10 ticks. A tick is the smallest unit of delay for the digital simulator. It takes one tick to perform a single step of the simulation for all devices.

Adjust X Magnification to view a larger or smaller section of the waveforms in the digital Waveforms window. By default, the magnification is set to 8. A smaller value zooms out, a greater value zooms in.

Use Simulation Speed to control how fast the simulation runs. This could be useful, for example, if the simulation is running too fast to view the states of a seven-segment display. Setting this field to a lower number slows down the simulation so you can view the changes of the display. Another method of slowing the simulation would be to run it in single step mode or set breakpoints.



Use the Breakpoint and Conditions options in conjunction with the waveforms window to set breakpoints. The following table illustrates the results of various combinations of settings.

Combination	Result
-------------	--------

Level-And	All break conditions must be met before the simulation stops.
-----------	---

Level-Or	Any one of the break conditions stops the simulation.
----------	---

Edge-And	The simulation stops when the proper edge occurs on all of the specified waveforms.
----------	---

Edge-Or	The simulation stops if a transition to any of the specified conditions occurs.
---------	---