

# LOGICSIM MENUS

## Apple menu

About LogicSim...

Displays information about the price & the author.

## File menu

New

Create a new document.

Open...

Open a document (from any 2.x version, not 1.x).

Close

Close a document.

Save

Save a document.

Save as...

Ask for a file name and save a document.

Save as PICT...

Save the window contents in a PICT file. PICT files can be edited by standard drawing programs and can be imported into most word processors.

Save Spice Source

Ask for a file name and save a Spice node-list version of the current circuit.

Save as Spice Source...

Save a Spice node-list for the current circuit .

Page setup...

Display a dialog to set printing parameters.

Print...

Print a document (its circuit drawing).

Preferences...

Define actions associated with double-clicking components. Also whether stimuli are listed in the Info window.

Quit

Quit LogicSim.

## Edit menu

### Undo

Available to undo the actions of Cut, Clear or the delete key.

Copy, Cut, Paste, Clear, Select all  
Standard Macintosh commands.

## Circuit menu

### Redraw

Redraw the entire drawing window. May be useful after erasing some items.

### Component delays...

Show a dialog where the pin delays for the selected component can be entered.

### Component options...

Show a dialog where some parameters of the selected component can be changed. This dialog depends on the component or module. Most components have no parameters, so this command is not available. Clock components are a notable exception and this is the way to set their period.

### Component name...

Show a dialog where you can type a name to identify the selected component on the circuit. If the first character is '~' then the component name is written with a bar over it on the circuit, the conventional means of indicating negation.

### Create module...

Create a new module.

### Load module...

Load a module or component onto the end of the pallet on the left of the current circuit window.

### Display family names

Devices belonging to a family of delays can have their names displayed, or not.

### Create new family...

Create a new family of delay definitions

### Modify family...

Change the definitions of delays in an existing family

## Simulation menu

### Options...

Show a dialog where you can set:

- Delays: which delay mode to use for the simulation
- Initial node values: (this should be X under normal circumstances)
- Simulator clock rate: the speed of the simulation in the real world
- Auto advance delay: advance the signals record each time an input is switched

These options are explained further in the chapter on Advanced Features.

**Run**

Start the circuit simulation.

**Stop**

Stop the circuit simulation.

**Reset**

If the simulation is running, restart it with the initial node values.

**Pause**

Stop temporarily or proceed.

**Show input values**

Display the values of input pins on the simulation drawing.

**Show output values**

Display the values of output pins on the simulation drawing.

**Show internal values**

When unchecked the real node values are displayed, when checked the internal values of components are displayed. The internal and external values will generally differ as a result of delays. However, if the simulation is stable (i.e. nothing is changing), the two display modes will give the same results.

**Record**

Stop/restart the recording of the LEDs/switches/buttons/clocks values in the Signals Window.

**Execute Stimuli**

Start executing the program of stimuli associated with the running simulation.

**Windows menu**

Each available window is listed and can be opened and brought to the front by selecting it.