

Cathy's EAGLE Reference

Icons below can be found on the side toolbar unless specifically indicated otherwise. In EAGLE, you click on a tool, then the item to which it applies.

Watch the status bar at the bottom of the window for instructions on using the tools!

Starting a Project

Boot EAGLE. You'll get the Control Panel window (the one with the red eagle). File/New/Project. Give it a name.

Schematic

From the Control Panel, File/New/Schematic.

Placing Components

 Add -- select the desired item, then click OK (not Drop!); click to place (multiple times as necessary); right-click to rotate before place; ESC will go back to Add dialog (or click another tool). Some interesting libraries: rcl = Resistors, Capacitors, Inductors; R_US type 204 = 1/8 W, type 207 = 1/4 W; supply1 has +5V, GND, etc

 Move (F7) -- click icon, then click part, right-click rotates, left-click places in new position.

 Delete -- click the icon, then the item to remove.

Changing the View

The zoom tools are on the *top* toolbar:

 Fit (Alt-F2)  In (F3)  Out (F4)  Redraw (F2)

 Select -- click and drag; or click for the two corners and then click  Go (or  to cancel).

Connecting Components

Use  Net to make connections (don't use wire!) -- click on the connection point from a component, then on the next component (or at a location where you want a bend). Use right-click to change the angle of a bend (see the options in the second toolbar at the top). The net will end automatically when you click on the connection point of a component. Junction points will automatically be added.

To see where the connection points are, use the  Display tool and turn on the Pins layer. You can also move a component to verify that the nets are connected to it. (F9 is Undo.)

Use  Move,  Split, and  Delete as necessary to get a nice layout.

Additional Stuff

Use  Name and  Value to identify components (including naming nets).

Use  **Electrical Rule Check** to verify your circuit. It will show things like missing connections. The status bar at the bottom of the Schematic window will show the number of errors and warnings, and if they're not 0, a new window will show a list of the problems.

To move a group of items together, use  Group to select (click-drag a rectangle or multiple click for polygon), then select Move and use ctrl+right-click to select the group (and left-click to place it).

 Copy -- click on the item to copy, then in the location to place the copy.

To copy a group, use  Group,  Cut,  Go, then  Paste.

For components that are part of a package (e.g. 7400 NAND gate), you will need to add connections for power/gnd using  Invoke.

 Pinswap lets you swap equivalent pins (e.g. to make board layout easier)

 Gateswap lets you swap two equivalent gates in the same package (e.g. to make board layout easier)

 Bus -- creates a bus to represent multiple signals.

 Label -- places a label showing the name of a net/bus.

Tip: Instead of physically connecting lines, you can give the same name to signals.

Use  Net, click the connection point for the first signal, move horizontally and click again to make a line, then hit ESC to terminate that net. Repeat with the other signal. Use  Name to identify each net. You will be asked to confirm the connection when you name the second net. *It is a good idea to use  Label to label both signals to make this connection clear to anyone looking at the schematic.*

Board

From the schematic, File/Switch to Board. Choose Yes when asked to create .brd file.

The Basics

Place items on the board using  Move (right-click to rotate). Move the board edges to be an appropriate size.

To create a ground plane, use  Polygon (not  Rect!) -- click the tool, make sure the desired layer is selected in the toolbar at the top (e.g. 16 Bottom), then click to create a rectangle near the edges of the board. Use  Name (click on the line you just drew) and call it GND. Say Yes to the confirmation to connect signals.

Click  Ratsnest to show the count of airwires remaining. This also cleans up airwires (e.g. if a signal moved closer) and the ground plane (e.g. after moving/rotating a component).

Use  Route (F11) to lay down traces. After selecting this tool, choose the layer in the toolbar at the top. You can also set the width of the trace and type of via.

Use  Ripup (F12) to remove traces.

Use  Move (F7) and  Split (F8) as necessary to adjust traces.

Use  **Design Rule Check** to verify that your board doesn't have clearance problems or other issues.

 Errors -- show the list of errors/warnings from the most recent DRC.

Additional Stuff

 Change does lots of interesting stuff.

 Smash separates the text (name and value) from the component so they can be moved independently.

Use View/Grid to change how the "snap" works. (Hold Alt to get the finer increment when moving items.)

The board outline is done using  Wire, typically in layer 20 Dimension.

 Display -- use to set layer visibility.

 Show -- click a signal to highlight all its traces and airwires.

 Info -- click an item to get information such as its layer.

 Via -- happens automatically if change top/bottom layer during routing.

 Hole -- e.g. for mounting (for 4-40, use 0.115 or 0.120 (screw is just over 0.100)).

Set the gap between ground plane and components using  Change/Isolate.

 Text -- for additional labeling.

 Mirror -- to place an item on the bottom of the board.

Don't Forget...

Include these in the schematic / board as appropriate:

- header for connecting power
- power switch
- voltage regulator
- programming header
- reset button
- testing tie points (to measure V, A, etc)
- mounting holes
- silkscreen (or copper) text with board name, version, copyright

Files for PCB Fabrication, Solder Stencils

From the Board window, File/CAM Processor

File/Open/Job, select board or stencil CAM file, Process Job

It's a good idea to load the resulting files into a viewer to ensure that they look correct.

Library

From Control Panel, File/New/Library (or File/Open/Library, or right-click/Open in Libraries list).

(I put all the stuff I've created into new libraries so I can upgrade existing libraries without risk of losing my components. It also makes it easier to get my customized stuff onto another machine.)

Libraries have 3 sections:

Symbol -- the image that appears on the Schematic.

Package -- the image that appears on the Board.

Device -- a symbol with corresponding package(s), and the pin mappings that link them.

After saving changes to a library, go to the Schematic or Board window and choose Library/Update.

Layers are important! In addition to the obvious coloring, they also define how elements will be handled when manufacturing boards.

Symbol

Library/Symbol, enter a name in New, click OK.

To start with an existing symbol:

(File/Open, select library.)

Library/Symbol, select the symbol to copy, OK.

Turn on all layers.

Use Group to select everything.

In the Command box just above the window, type: `CUT (0 0)`
(this really just makes a copy)

Switch back to your library (e.g. with File/Open Recent).

Library/Symbol, select your new symbol.

In the Command box just above the window, type: `PASTE (0 0)`

 Pin will add connections. (The green "ball" is where the connection is.) Icons

on the top toolbar will select decorations like Clk and Dot. Use Name to label the pins. Use Change to set pin purpose, pin/gate swap level.

Layers...

94 Symbols	Wire/etc	0.010 Width	Outline, symbol, etc.
95 Names	Text	0.070 Size	>NAME
96 Values	Text	0.070 Size	>VALUE

Package

To copy an existing package:

Find the library in the EAGLE Control Panel, expand its contents, scroll down to the packages

Right-click the desired package, choose Copy to Library

For a new package, Library/Package, enter a name in New, click OK.



Pad: Drill = 0.032, Shape = Octagon (or 0.040, Long); (Thermals / Stop = On)



Smd -- surface-mount pad.

Layers for visual indicators...

21 tPlace	Wire/etc	0.006 (to 0.010)	Silkscreen markings (placement/orientation markings)
51 tDocu	Wire/etc	0.005	Layout aid, not for silkscreen (button outline, board edge)
25 tNames	Text	0.050	>NAME
27 tValues	Text	0.050	>VALUE

Use  Info to check on things like layer assignments and pin names.

Layers for solder mask ("Stop" layers) and solder paste stencil ("Cream" layers) are handled automatically by Pad and Smd.

Device

Library/Device, enter a name in New, click OK.

Use Add tool to select a symbol. Use Name to name it.

To add a package, click New button (in lower right of window), select a package, (give it a name if there will be multiple variants, OK.)

Select the package in the list and click Connect; map the pins.

Edit/Description to add text description; will show in Info and (Schematic) Add.

(Click Prefix to set a prefix, e.g. R for resistors.)

(Right-click item in Package list to edit variant name.)