

Eagle Commands

Description

As part of Gadgetry you will be using Eagle; Eagle is a computer-aided drafting (CAD) software tool to allow creation of printed circuit boards from circuit schematics. Unlike most such tools which cost thousands of dollars, Eagle has a limited freeware version. You can obtain this for any OS at:

<http://www.cadsoft.de/download.htm>

There are three components to Eagle; the *Schematic View*, the *Layout View*, and the *Library Editor*. This document concerns just the schematic and layout views. Editing libraries will be covered later on in the course.

The schematic view allows you to create a logical representation of your circuit. It shows parts as simple electrical symbols, and connections between parts as green wires. The connections between parts are known as *signals*. Signals are how power and data are transmitted within the circuit. When you have completed an electrical schematic in the schematic view, you can then move on to the layout view. The layout view is a physical representation of your gadget, showing all the parts in proper proportion to one another. The wonderful thing about an integrated tool like Eagle is that all of the signals and parts described in the schematic view are automatically generated in the layout view. Signals look like yellow wires connecting different parts together. Furthermore, if you make a change to your schematic, it will be automatically altered in the layout. Thus you can use the schematic as a way to reason about your circuit, and the layout as a way to create a manufacturing and assembly drawing for it.

To use Eagle, you will need to learn to effectively navigate the user interface. Both layout and schematic views have a number of buttons on the left side; clicking on one of these buttons will put you in a *mode*, for example, if you click on move, you will be able to move any part, wire, or text that you then select (by left clicking on it). You can also select modes by typing in the command line prompt directly above the viewscreen; you just need to type the name of the mode you wish to access. This is by far the faster of the two ways to switch modes, so it is worth learning the names of commonly used modes. The rest of this primer describes how to use these modes.

Schematic View

use – Type this command in order to add a library (like gadgetry.lbr) so that you can use the parts in the library. When you type it, a dialog box will open – just navigate to the folder in which you placed the library and then click on it. Once added you shouldn't need to re-add a library to the same schematic (unless you move the schematic between different computers).

add – Adds a part to your schematic from any of the loaded libraries. Eagle loads a large number of standard libraries, so you'll see a large list. One of the libraries should be gadgetry – click on the '+' next to gadgetry to expand the library and see which parts are available. Some parts have multiple instantiations – those with '+' to the left of them; you can click '+' to expand and choose the instantiation you want.

move – Type move to go into move mode, then left click on the part, name, or wire you wish to move.

rotate – While actively moving a part, you can rotate it by right clicking. You can also go into rotation mode by typing 'rotate' and then left clicking on the item you wish to rotate.

wire – Type wire to go into wire mode; left click once to start the wire, then move the mouse in the direction you want the wire to go – you'll see a shadow wire following your mouse movements. Once you are satisfied with the position of the wire, left click again to complete the wire.

junction – If you are connecting two wires in a cross, you must place a junction to electrically connect them, otherwise Eagle assumes they're just running on top of each other. Type junction and then click on the spot the wires cross to place the junction.

name – You can use name to change two things: To change the name of a **part** or to change the name of a **signal**. To change the name of a part, type name, then click on the part – you'll be able to name it (note that eagle capitalizes all letters of a name automatically). To change the name of a signal, type name, click on the wire associated with the signal, and then name it. To connect two wires in the schematic which are not connected by wire, you can name them both the same name.

label – The label command causes the name associated with a wire to be displayed. Type label, and then click on the wire whose name you want to display.

value – Changing the value of the part is like changing its name. Type value, click on the part, and then type the correct value for the part.

group – To modify multiple parts and wires at once, you can type group. Once you are in group mode, click and drag the mouse so that the displayed box completely envelops all of the parts you wish to modify. Once you have created a group, you can go into a different mode (move is a popular one). In this different mode, if you wish your actions to apply to the whole group, **right-click** anywhere in the group. This is especially useful for moving large parts of the schematic or layout around at once.

mirror – To create the mirror image of a part, use mirror and then click on the part you wish to mirror.

undo – You can undo your last action by going to edit->undo or by typing Alt-Backspace.

Abbreviating Commands – You do not need to type the full name of any command, just enough so that Eagle can identify it uniquely. In practice, if you type the first three letters of a command, that is enough to activate that command.

Layout View

Many of the commands that you used for schematic design also apply to layout. For example, move, group, undo, and rotate all work the same way. There are some important additions and modifications however.

auto: This magic command starts the auto-router, which will automatically route the entire board with copper wires. This is the quick and dirty way to route a board, and may or may not apply to your project.

IMPORTANT: Make sure the routing grid field is set to 5 (it defaults to 50)

route: This starts the routing tool, which allows you to manually route individual 'airwires', the thin yellow lines connect your nets together. To route an airwire, type route and then click on an airwire. Note that you can adjust the width of the wire as well as the size and shape of vias that are caused by your routing. You can also set the layer you are routing on to either "Top" or "Bottom". If you change layers in the middle of a route, Eagle will automatically place a via at the junction. All of these options are above the command line.

change width: Lets you change the width of your routed copper wires. You may wish to make the width of a certain wire wider than normal because it is supposed to supply power.

change layer: Lets you change the layer in which you are working – when it comes to routing wires, only two layers matter; 1 and 16, corresponding to the top and bottom of the board. It is very difficult to make a circuit board with wires on only one side, so you'll often be switching between layers when manually routing.

via: Allows you to place a via, or a hole in your board. A via allows your wire to move between top and bottom layers, which can be useful if your circuit board contains a lot of parts.

rats: This organizes the airwires so that they make more sense after you've moved parts around. Hard to explain, but you should probably issue a rats command every time you move a couple of parts around.

drc: This command runs the design rule checker, which checks your layout to make sure there are no errors. This is very useful to run once you believe your design is complete, as it tells you if there are problems which could prevent your circuit board from being manufactured.

Important: If DRC gives clearance and width errors before you start routing wires, talk to us

mirror: Mirror works somewhat differently in the layout view; in addition to flipping your part, it also moves it from the top side of the board to the bottom side (or vice versa). You should never need to use mirror unless you're making a board with components on both sides.

smash: Smash allows you to move the names of parts independently of moving a part – very helpful when positioning the names for connectors. To move a name, type smash, left click on the part, and then type move and left click on the name itself. Since this is kind of lengthy to do for each part, it is do a group command and select your whole circuit board, and then type smash and **right-click** to smash the entire group.