Cross reference: use the label command, then click the xref button or go to properties of the label and check xref.

Show @ referencename (cmd) – has EAGLE point you to the specific cross reference label to find it.

Use test and wires to document all the important information in your schematics!!!

Power connectors overwrite the net name with whatever they are called!

If you connect ground to power and allow it to connect, it will rename ALL OF YOUR POWER SUPPLIES! Don’t do this. It would be awful.

Ctrl+d for DELETE!

EAGLE BOM

Run > BOM > check list type values, not names, which will sum resistors of same value into one row, etc.

Attribute (cmd) > NEW > add attributes to individual parts so when you make the Bill Of Materials (BOM), these attributes will be columns in the spreadsheet!

**Making a part!**

NE5532 for example

Looks at datasheet – it’s an SOC, it’s got 8 pins, it’s a dual package op amp

Go to control panel > file > new > library

Libraries contain three things

Device

Footprint

Symbol

Symbol + footprint = device

Steal a symbol: Library TOC > Symbol column > Add symbol > import >

Pad (cmd) – gives thru-hole pad for component

Smd (cmd) – gives smd pad for pin

Can change grid size to distance between pads!! Then it’s super slick for adding the eight pads, as the grid will automatically separate them by the proper distance.

Pro tip: type coordinates into the line command: [0 0] [1 1] [0 1] Gives line coordinates…

Need a pin 1 marker!

Use text: >NAME (WILL FILL IN AUTOMATICALLY THE NAME OF THE DEVICE IN YOUR SCHEMATIC! No space between “>” and “NAME”!) Use the following layers:

* tname layer on the package
* tplace layer on layout

Add device

Add the components for the schematic

Under the package sidebar, select new, then “Add local package”, then SOIC8

Prefix: U

Connect: Assign ports to the appropriate pins

You’ve made a new Library: Now use (cmd) > Available > Browse (and select the new library you made). Then got to In Use (or Available), and select the library, then click USE. (if it says REMOVE where it should say USE, it’s already in use!)

**IF YOU HAVE MORE THAN ONE COMPONENT IN A DEVICE:** (such as the NE5532, which has two op amps and a set of power pins), click repeatedly to place all the components of the device (three times for the NE5532 to get the two op amps and the power supply)

If you make changes to a library midway, you must update it!!

Update (cmd) select the library, then updates the library you’ve made changes to.

Shift+delete to delete entire polygon (ground plane gnd)

Polygon (draw around board), then type ratsnest (cmd) which will fill the ground plane!

Rip @; (hit enter) It’ll rip up the entire ground plane, as in un-flood it.

Ripup @; (hides ground plane)