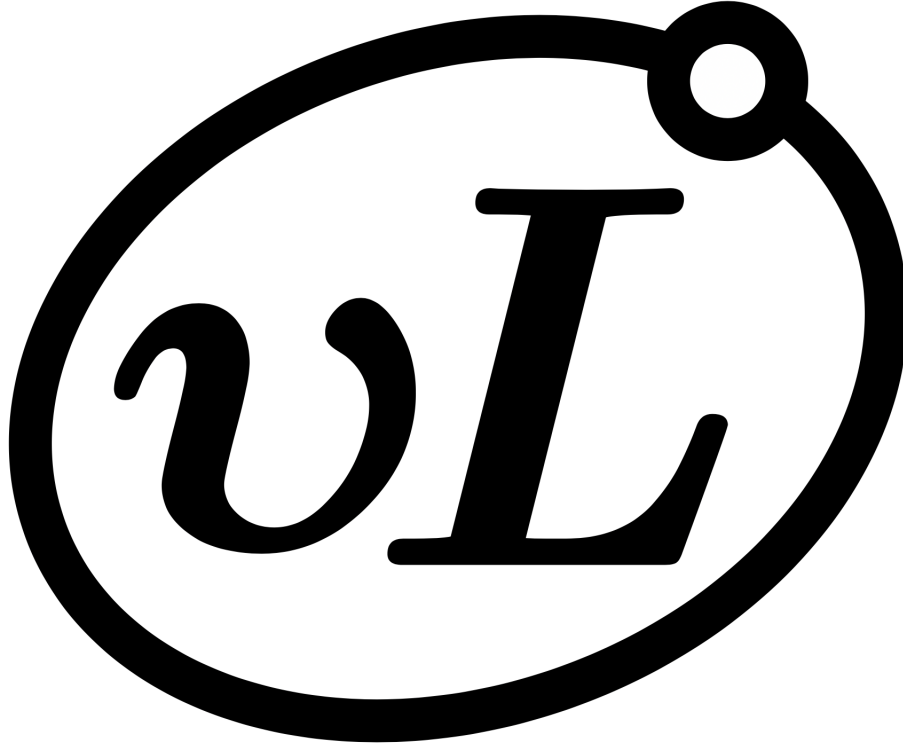# Upsilon Lab *Designing PCBs with EAGLE*
an Official UCLA Physics & Astronomy Department Sponsored Organization

Kayla Jane Rodriguez, Independent Author
Edited by Jacob Saret, Founder-President
April 23, 2018

"A lab for undergraduates, by undergraduates."

1   Summary

This guide will help the reader understand the basic actions of EAGLE with the objective of designing a printed circuit board (PCB) which can be realized through numerous third-party PCB printers. All of the stages of designing a PCB from start to finish will be covered in this guide. Several of the tools I found helpful when learning EAGLE are cited in Section 4 for your reference.

The process starts with designing a schematic, and learning basic functions like rotating, moving and coping, among other tricks. Next, your schematic will be converted to PCB format. At this stage all the wired connections you made in the schematic will be connected on the PCB. These connections are not solidified yet, because they are just "air wires" that show you what needs to be connected. You will want to place these components in such a way that minimizes the number of layers the board requires. This is more like a puzzle than anything, but their are tricks in simplifying the process of placing these components. This is followed by the process of routing-placing your printed wires on the board.

2 Creating the Schematic

2.1 Adding Components

2.1A Searching for Components

To search for components, click the small component icon on the side bar, this icon is shown labeled Table 1. Here, you search for any of the component parts you will need for your board. It is helpful to search a couple letters rather than the entire part number, for example, "OP" if you are looking for some specific op-amp. The whole part number is usually too specific, several different options are always useful. If you are looking for more generic parts such as resistors or capacitors, then simply search the entire part name. Do not get overwhelmed by the number of available generics, because the differences are usually unimportant at this level. If you want a compact or high-wattage resistor, those are available.

When I search for resistors, I usually pick the R-US option which gives the standard US symbol for the resistor, and from there I pick my PCB footprint. These are usually holes with varying distances and I will normally compare those distances with the size of the resistors I have in my lab. Again, for resistors, the footprint does not matter too much unless it's a potentiometer or has an attached heat sink. For capacitors, I go through the same process, and I make sure that the holes are not unreasonably big or small.

It is important that you search "GND" on the search bar and connect your parts to that GND as necessary. EAGLE may ask you if you want to name something GND, or if you want to change that name, make sure all your grounds have the same name so they will all be connected. This will be discussed further in Section 3.3.

2.1B Importing Component Footprint into Your Custom EAGLE Library

Many times, the part you need to add to your schematic simply is not in EAGLE's built-in library-you'll need to add these yourself. Of course, first check by searching the library for your component as described earlier. If it is unavailable, there are several options:

1. Find a component with the same shape as the one you are looking for. However, when you do this you need to check that it has the exact same dimensions as well as pin assignments.

2. Import a new library or component footprint.

3. Make your own custom component through EAGLE. This is very tedious and requires the exact dimensions of your part.

Most circuits are available only using the library and the first two options. Thus, I will only discuss option 2 in detail- option 1 only requires a datasheet comparison. However, I will provide external sources which discuss option 3. There are several ways you can import component parts, either importing a library file from github or from importing a specific part from snapEDA.com or ultralibrian.com.

To add any new parts to your library, go to the EAGLE folder, and find the "lbr" (library) folder. All your new libraries will go here. It may be under your Applications folder (Mac) or your downloads folder (Windows). Any new libraries you find you will be drag to be in the lbr folder.

You can add these components through the sites I mentioned earlier, or simply searching the web for the desired component.

If you download a library from github (e.g the Adafruit or Spice library), you need to download a zip file of the library to add to your lbr folder. When you download specific components to your library, it is important to keep track of these components. A folder titled "My Library" inside the lbr folder will suffice- make sure to place components you find in this folder.

2.1*C*   Example: Finding Parts on snapEDA.com

Once you've found your part on snapEDA, click the "Download Footprint" (Figure 1) or the "Download Symbol and Footprint" (Figure 2) and add this to your lbr subfolder. If neither option is available, you'll need to find another listing of the part. snapEDA suggests related symbols you can use if only a footprint is available, and you can download the schematic and footprint for this related part, and use this footprint. For example, I searched for an OP270, looking for the DIP-8 package; however the schematic symbol was not available. To solve this problem, I clicked the related symbol link and downloaded the symbol and footprint and put them all under the lbr subfolder, and now I can use my newly downloaded OP270 DIP-8 package! The "related symbols" link is shown in Figure 3.

Figure 1: Download footprint on snapEDA

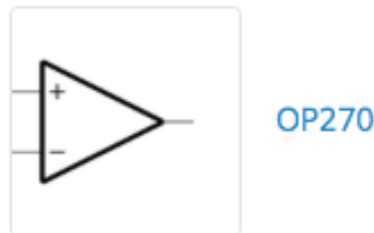Figure 2: Download footprint and schematic on snapEDA

Figure 3: Download related schematic symbol on snapEDA

Now, none of this really makes a difference if you cannot access these libraries. To access the

libraries, go to your control panel, go under Libraries tab, and click lbr. Without any downloaded libraries you will see just "Managed Libraries", this tab is the built-in library that comes with EAGLE. If you followed the steps from above, you should see a tab with the same name as your lbr subfolder, or any other libraries you may have downloaded from one of these repositories. Note there are gray dots next to every new library- this indicates that these libraries are not being used, meaning you have no access to them. You need to click all of the corresponding gray dots to enable them Those dots will turn green, and the libraries will become available. This is shown in Figure 4.
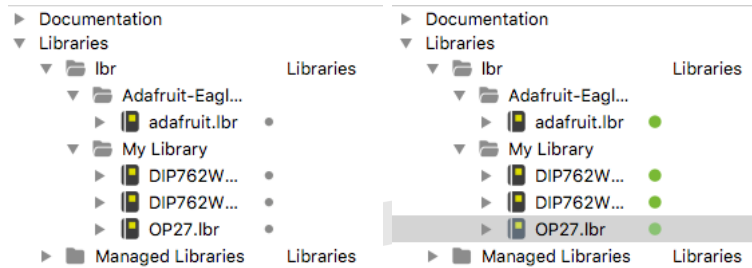

Figure 4: Libraries: a. inactive and b. active.

2.2 Basic Functions

Once you make the schematic, EAGLE has several basic functions that every user should become familiar with, including: properties of a component, copying, pasting, deleting, and routing; shown in Table 1. All these icons are the same picture on the brd file, except for the routing icon.

To access the properties function, click the top left corner of your tool functions and find the button with a mouse icon dragging a box. The properties function allows you to add values to your components (e.g indicating that a resistor is a $10k\Omega$), using a function called "smashing", this allows you to move the text that comes with different components (i.e the value, the part number). Without checking and applying the smash tool, you will not be able to move this text. This becomes important when your circuit becomes large or complicated and you want everything to be more readable. To show a value, you would type in this value in the value box, and check show for the text to appear. It is important to note that to access the properties of any given component, you must click the plus sign associated with the component- **this is true for all of the other basic functions!** Otherwise, EAGLE cannot decipher which component information you need.

Copying and pasting are useful when you have many identical or similar components in your PCB. Once you've clicked the copy button, click the plus sign of the component you are copying. To paste, click on another area of the schematic to place this new component. The process is similarly simple to delete components by clicking on the trash can icon, and then clicking on the plus sign of the component you want to delete; or rotating by clicking the circular arrow icon. To move parts around you would need to click an icon that has four arrows (two going up and two going outward). In a similar fashion, you can also add text to your schematic. This text will not be converted on your PCB, it is just for your reference on the schematic. Adding text is different than the built in text you can have shown through the properties functions, the text added through properties will be shown on your PCB. The text is meant for the schematic only.

The same icon used to check the properties of a component can also be used to make the same changes learned above, except to a group. If you use your mouse (while making sure the properties icon has already been selected), to click and drag across multiple components. This selects these components and you can now move, copy, paste, trash, rotate, etc with the entire group, rather than a single one.

To route your components, or wire them, click the green corner icon. Once you click this icon, click one end of a component (circles should appear) and connect it to the other end of a different component. If you cannot just connect two components with a single line, then click the part of the schematic where you want your routing to switch directions. These connections are important because the way these are routed are the way your PCB will wire your components. If your schematic is done correctly, then your PCB is wired correctly. However, when considering these connections on a PCB, it is important to consider whether the placement of a component makes sense, not whether it is wired correctly.

A less important basic function is the different mouse shortcuts. When rotating a part, you can click the rotate icons, or you could right click your mouse to have the component rotate. Also, zooming in and out can be done with a scroll of the mouse. This section obviously does not apply if all you have is a laptop trackpad. All the basic functions will work without these mouse functions.

3   Converting and Creating the PCB

To convert your schematic to a PCB, you would click a button towards the top of the screen that reads "SCH" on top with green surrounding all of the top half and "BRD" on the button with gray surrounding the bottom half of the icon (on the PCB board it will be the same similar text except the top half of the icon will be red instead). These icons are shown labeled Table 1. This converts it to a PCB and automatically creates a link between the two files. This link allows you to go back to your schematic and make changes and convert it back to the PCB in the same file.

EAGLE allows consistency checks- if there is a failure, you'll need to re-examine your schematic and PCB. All the basic functions you can do on a PCB are used the same way as the functions described in Section 2.2, **except the routing**.

Once converted to PCB, all your components are wired by something called "airwires". These wires will not actually exist, but are there to make sure you correctly wire your components. The only components you can route on your PCB are connections on the same airwire (or wire with the same name). If you want to connect things that are not connected by an airwire, you must go back to your schematic and make those changes.

3.1   Placing Components

On any one layer, your wires cannot touch. This can be difficult when trying to get all the components on just one layer so there are some tricks to keep in mind. The cost of PCBs increases greatly with each additional layer, so optimization is a crucial skill. If you are an undergraduate or otherwise amateur, you should be able to fit most of your components on just one layer, with other layers holding few components. There is an autorouting feature in the software, but it is not that efficient, so it is usually best to place the components yourself. **It is extremely important that you do not route your components until the end, when none of your airwires are**

touching. Also, if you know certain airwires are GND and you plan on making a ground plane later on, then these wires may cross others without consequence. A trick on placing components would be to place components near each other that are within the same section on your schematic, and overlap certain component pictures if you know it does not matter if they overlap in real life (i.e two resistors). Furthermore, I recommend putting these wires through those pictures might also be easier.

3.2 Routing

There are four primary types of PCB routing: straight, 45 degree, 90 degree, circular, but you can use any variation of the three. If the speed of your circuit matters (or the frequency at which it operates), wiring and component placement becomes significant. Typically, electrons in your wire will be able to travel the fastest in circular or straight wiring with no kinks. However, on a rectangular board, circular wiring is not usually practical. The next best and professional standard is the 45 degree option. If speed is not a concern, 90 degree wiring will suffice, and your board will be neat and organized without much effort. You can choose your wiring type easily by looking at the top tool bar of the PCB file, or by right clicking your mouse. To route different components on your .brd file, you need to use green line icon which looks like Table 1.

If you wish to revert to an airwire, use the "ripup" function. The trash function will **not** return your airwire, and recall you cannot route things that are not connected by an airwire. This icon is labeled Table 1.

3.3 Adding a Ground Plane: Polygons and Using the Ratsnest Function

Once you have your circuit all laid out, you can now add a ground plane. Make sure that all your grounds share the same name if you want them to be connected- there are reasons you wouldn't, but if you don't know, you probably don't need multiple grounds. To make a ground plane, you need to first click on the make a polygon icon which is shown titled Table 1. Create this polygon over your circuit by making a rectangle around it. If you click under properties, you will see that the polygon's name is something that is not GND. This means that it is not a ground plane yet. If you try to change the name in properties, it will not work because you must use a different tool for this. Also, there will still be GND airwires, click the option that allows for the airwires not to show, and click apply to resolve this issue. These airwires are not needed as they will be connected to the ground plane, and this will help you find errors in the future.

You must use the NAME tool shown titled Table 1. Once you click this icon, click anywhere on your polygon and it will ask you what you want to rename- the signal or the polygon. To make a ground plane you would want to rename the polygon to GND to match the name of your grounded components. After doing so, then click the ratsnest tool which is shown titled Table 1. Now, if everything is right there will be no more airwires. All your components should have been routed before adding the ground plane, and if you made sure polygon properties do not show the airwires, then you forgot to connect something. If this is the case, you need to undo your ratsnest, and go back and connect the wire.

After you've set up the ground plane, you are nearly done!

3.4 Error and Consistency Checking

EAGLE's error checking function (Table 1) is quite useful. Here, EAGLE will show you some obvious mistakes in a list that you can either fix, or approve (i.e. ignore). In light of Section 3.3, you can approve are GND airwires crossing once you have the GND plane set up.

4 Kayla's Sources for Learning EAGLE

Video guides I found useful when learning EAGLE myself:
1. https://www.youtube.com/watch?v=1AXwjZoyNno
2. https://www.youtube.com/user/EAGLECadSoftComputer

EAGLE's official technical guide:
https://www.autodesk.com/products/EAGLE/blog/
library-basics-part-1-creating-first-package-autodesk-EAGLE/

5 Exercises

1. Make a folder of your own library parts and find the custom footprint of an 8-Lead PDIP (P-Suffix) Operational Amplifier with the part number OP270. Once this footprint is imported in your library, use it in your schematic and convert it to a PCB. Note: Many OPAMPs are the same size, even though they have different part numbers and often times those can be used instead. However, this specific op-amp is a dual gate, which has a different pin setup and this is important to get correct. This exercise is meant to test your knowledge on importing a library component into your own custom library.

2. Download a library such as Spice, or Adafruit from github and place one of its components onto your schematic and convert it to a PCB. You need to take the zip file from the github page, and be able to have access to this library when adding parts.

3. Add a resistor to your schematic and give it a value, does not matter, could be 5k, 10k, 100k, or whichever weird number you need. First, you need to make this value show on your schematic, and you need to be able to move this number around on another part of the page. This problem demonstrates your knowledge on the different basic functions of movement.

4. Practice different types of routing! Make a circuit with a BNC, an op-amp, and a resistor. Make it on a PCB board and try routing them with the different types of routing: 45 degree routing, 90 degree routing, and circular routing. Usually 45 degree routing would be best to create the fastest circuit, but sometimes others are necessary.

5. Make a inverting amplifier circuit with a BNC for each power supply of the op-amp, and an input BNC. Make sure your BNC is properly grounded and convert this circuit to a PCB. Once all your components are routed, make a polygon, fill using ratnest, and name this GND. This is absolutely necessary because without it, your circuit does not a reference to ground.

6 Menu Icon Glossary

Table 1: Menu Icons

Information		Sight
Layers		
Properties		Move
		Rotate
Copy		
Delete		
		Add Part
Name		
		Text
Junction		
		Polygon
		Routing
Routing		Ripup
Via		
Holes		
Ratsnest		
Error Check		
Warnings		