Introduction

Welcome to the Eagle Proto-board Tutorial! This tutorial is intended as guide on how to layout a circuit on a proto-board using the PCB CAD software Eagle. This guide is NOT intended to instruct on how to use Eagle, for a guide on using Eagle we recommend the SparkFun tutorials, which can be found here (note that there are two parts to the SparkFun tutorial, they don’t take long and it’s certainly worth the time).

Building the Schematic

So now that you’re an Eagle expert, start building your circuit in Eagle’s schematic editor, make sure to have included the same SparkFun parts library as used in the SparkFun tutorial.

Make sure to pay attention when selecting your parts!

- For ICs you want the DIP or DIL (dual in-line package) components.
  - Use the search feature when adding an IC. ex: search “microchip” when looking for the 6004 or 6002 Op-Amp packages.
- For passive components:
  - Resistors:
    - Use the ¼W resistors with the AXIAL-0.4 package
    - Tip: Use the ¼ W resistors with the AXIAL-0.1 package for when you plan to insert resistors in a vertical configuration to achieve a smaller grid footprint.
  - Capacitors:
    - Ceramic caps: CAP-PTH-5MM
    - Electrolytic caps: CPOL-RADIAL-XXUF (look for the appropriate capacitance value for your application)
- Everything else:
  - Diodes: DIODE-1N4001
  - Special components (e.g. LEDs, phototransistors, etc.): Do a search.

Now that the schematic has been built, have your partner review to make sure it is free of mistakes. Once everything looks good, it’s time to start planning out our board.

For the sake of this tutorial, the Mini Beacon circuit from Lab 0 is going to be used for demonstration.
Here is the schematic:

Board Layout

Now we switch to board view (in case you forgot the button, it’s in the upper left hand corner to the right of the cam processor button):
As we see, we have our parts and a board outline we would like to fit out circuit on. Before we can begin fitting parts, we need to make some tweaks to our grid and board outline. First we want to modify our grid, click the grid button in the upper left-hand corner of screen:

A window will pop up like so:

![Grid Settings](image)

Change the size to 0.1” (make sure it’s set to INCHES!) and the alt to 0.005”. Then select ok. What we have done here is change the grid so that it closely reflects the spacing of the holes on our proto-boards.
Next, we want to adjust the board boundary size, use the move tool and adjust the boundary size to reflect the height and width of your board. The board I’m using is 12 holes high X 13 holes wide; that dimension is reflected in the gray box in the upper left-hand corner of this screen:

Now start populating your board. Just as you did in the SparkFun tutorial try to layout your parts so that you minimize crisscrossing of the connections, it takes practice but with time a good layout will emerge. Use the alternate grid (hold the “alt” button to turn on the alternate grid) to space the ends of the components inside of the individual grid boxes. I’ve chosen to layout this board like so:
Once the parts have been put down to a satisfactory level, it’s time to start routing the connections. Before we begin, we should remind ourselves that this layout is for a proto-board, not a PCB, and thus we need to be bit a clever with our routing.

We start off by routing all of the bottom connections first and try to minimize the number of top connections that need to be made. Top connections are going to be made with 24 AWG wire and will need to be laid flat and neatly onto the board. Due to the parts on top of the board, your options to route wire on top are few.

Here is the routing for the bottom of the board, it probably could’ve been better but for the sake of this tutorial it’s fine.

As we can see, we have four more connections to make that couldn’t be easily made on the bottom, so they’ll need to be connected on top. This is where we need to pay close attention.

To route wires on top, we need to take into consideration that the ends of the wires will take up one grid space, start by routing the connections on the bottom and then use vias as you did in the SparkFun tutorial to transition to the top side of the board. These vias will represent where the ends of the top wires will terminate on the grid.
Here's an example of how a top wire was routed from the output of the diode to the IC, note placement of the vias and how the transition from bottom top and vice versa:

![Diagram](image1)

This was repeated for the last four connections, of which some required a bit of creativity. Here is our final layout:

![Diagram](image2)
Soldering

Now comes the fun part, soldering! Before we start soldering, we’re going to want to print the layout as it appears from the bottom. This is where you will be doing your soldering, so it’s good to have a reference to minimize errors in circuit connection. I recommend printing to PDF to preserve color but paper is fine too. When you select print the following dialog appears:

Make sure to select “Mirror” and “Upside down” in the options list, you will also want to adjust the scale factor so that your circuit covers the whole page. Also, you will probably want to change the page orientation, this circuit is pretty square so I left it in the “Portrait” orientation. The one downside to printing the layout in Eagle, is that the grid spacing isn’t included in the print, so you will have to rely on Eagle’s Board editor for the correct spacing of components.

Print the PDF and open it (it’s located in the same folder as your Eagle Project).

Now we can start inserting components into the proto-board, use the Board editor for this so that you can pay attention to the correct grid spacing. For bigger boards, I STRONGLY recommend starting with only a single circuit module (e.g. phototransistor stage, amplification stage, peak detector, etc.), so only insert components for that individual module. DO NOT insert all of your circuit components and solder everything at once, I assure you that you will have a bad time. We’ll treat this relatively small circuit as a single module.
Once you have the components to your first module insert into the board, CHECK, DOUBLE CHECK, and TRIPLE CHECK that your components are in the same grid holes as your virtual board in the board editor:
Now we can begin soldering, start by first “tacking” down your components one by one so they don’t fall out, don’t cut the ends at this time:

Refer to your layout on eagle and the upside down version of your layout on PDF/paper and think about how to make the connections. The preferred way is to use appropriate lengths of wire and solder, some use the ends of the components as their “wire” (hence, why I suggested to hold off on cutting the ends), and some just use solder. I use a combination of the three, but for this circuit I deferred to the latter two techniques.
For the top, you’ll want to use wire and routed it neatly like you did on the board editor (this is also the completed working circuit):

Conclusion

One thing that I didn’t add on this circuit, that should have been done and that we strongly encourage you to do, is to include test point terminals on the I/O of your ICs and other interesting parts of your circuit modules for easy debugging when you will inevitable encounter issues.
When you have finished a circuit module test it thoroughly to make sure it works as it did on your breadboard. In fact, it is helpful to keep your protoboard and completed breadboard circuit from Lab 1 side-by-side so that you can easily test and make sure you are observing the correct output from your modules. Once you’ve agreed that your module has been thoroughly tested, rinse and repeat the process for the rest of your modules making sure to test them along the way.

Also remember that your resistors can be stood vertically for a smaller footprint compared to horizontal orientation I used in this tutorial. Often a lot of space and routing headaches can be saved when swapping the resistor packages from AXIAL-0.4 to AXIAL-0.1.

This concludes the end of the tutorial, Happy Prototyping!