

# Making Your First PCB

## Before You Start

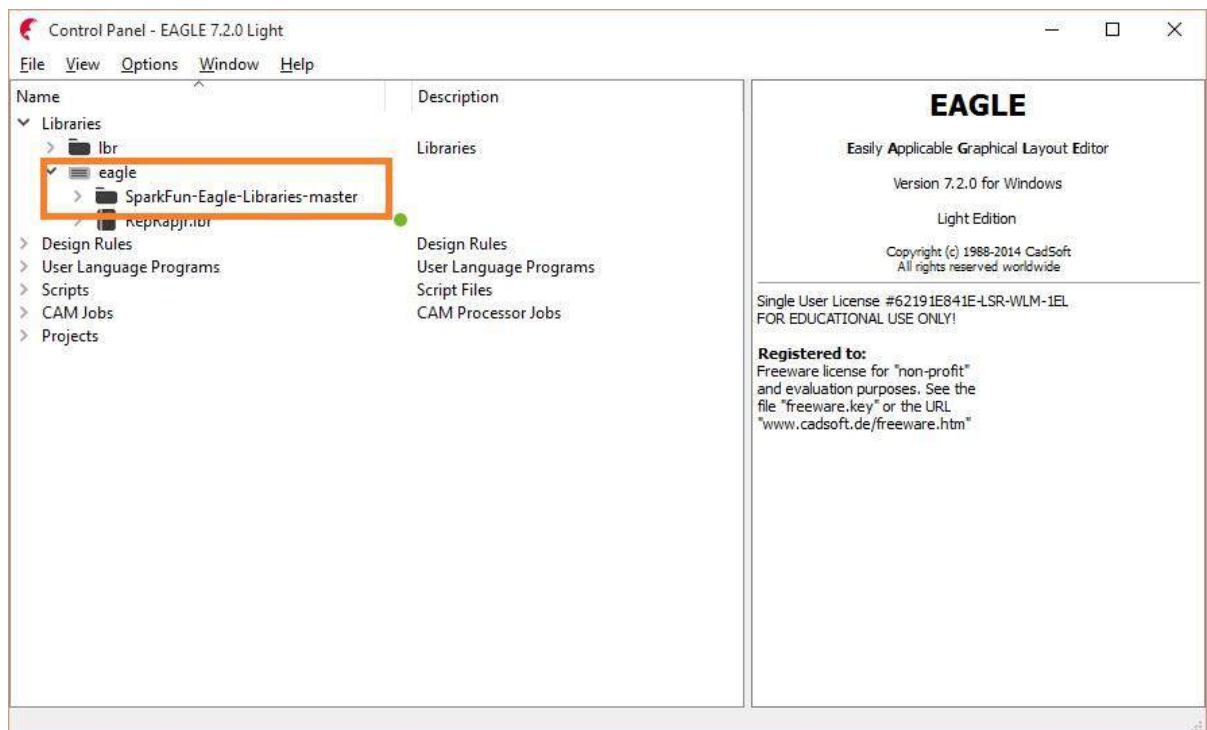
Before you start, you should have covered and completed everything in the tutorial below. This includes installing Eagle, importing the Sparkfun libraries, and navigating throughout the program. At the very least, you must have eagle installed and the Sparkfun libraries imported.

<https://learn.sparkfun.com/tutorials/how-to-install-and-setup-eagle>

We are going to be using the Sparkfun libraries because they are much simpler to use than the default libraries that Eagle is installed with. Learning to navigate the default libraries is a task that can wait until you have a solid grasp on the basics of the program.

## Starting a New Project

1. Open EagleCAD. Ensure that you have the SparkFun libraries available by expanding the “Libraries” tab on the left side of the interface.

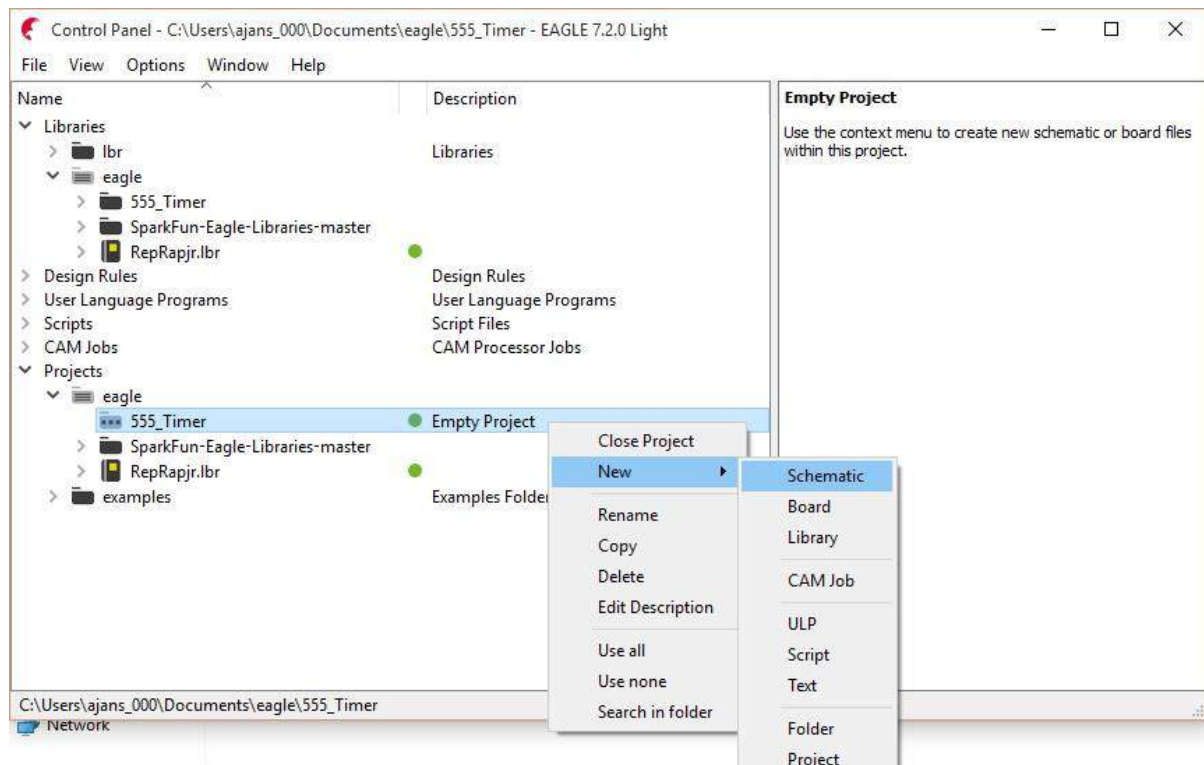



2. Right click on the “lbr” folder, and select “Use None”. Next, right click on the “SparkFun-Eagle-Libraries-master” folder, and click “Use All”. We have just told Eagle that we don’t want to use the default libraries, and we want to use *everything* in the SparkFun library for our project.

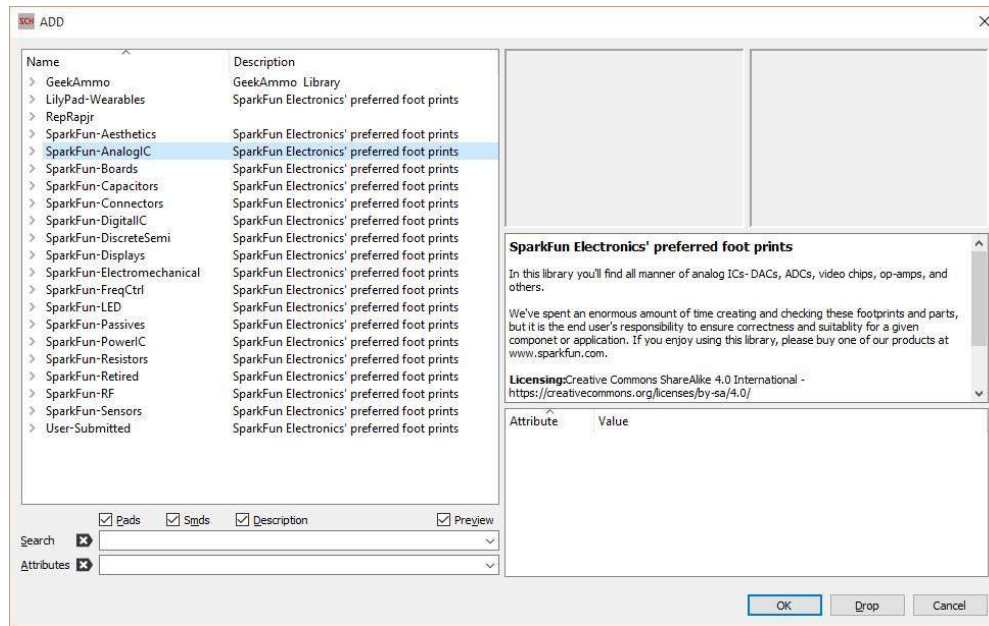
3. Create a new project (File > New Project). Name your project "555\_Timer".


## Making a Schematic

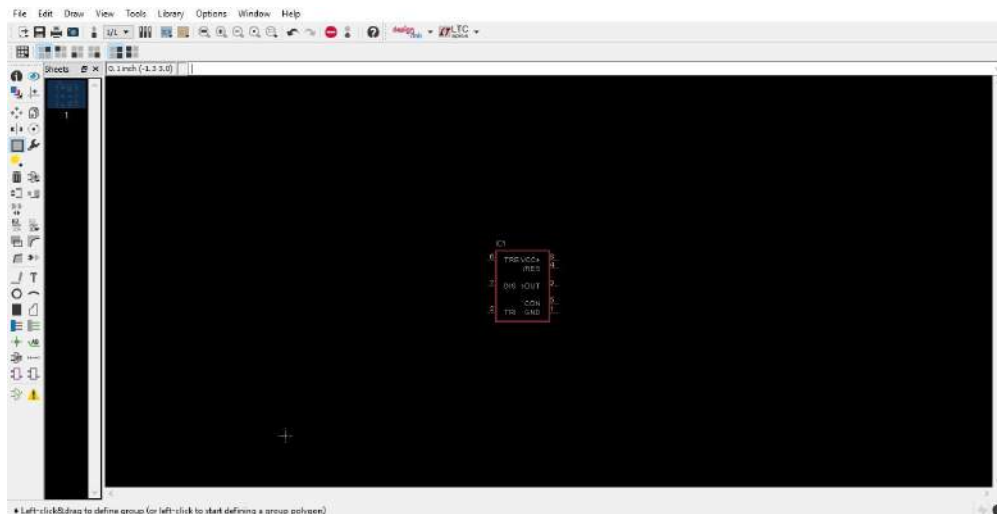
1. You should now have a project under the "Projects" dropdown in the left window. Now that we have a project to work with, we want to add a schematic. Go ahead and add a schematic now. (Right click on your project > New > Schematic).



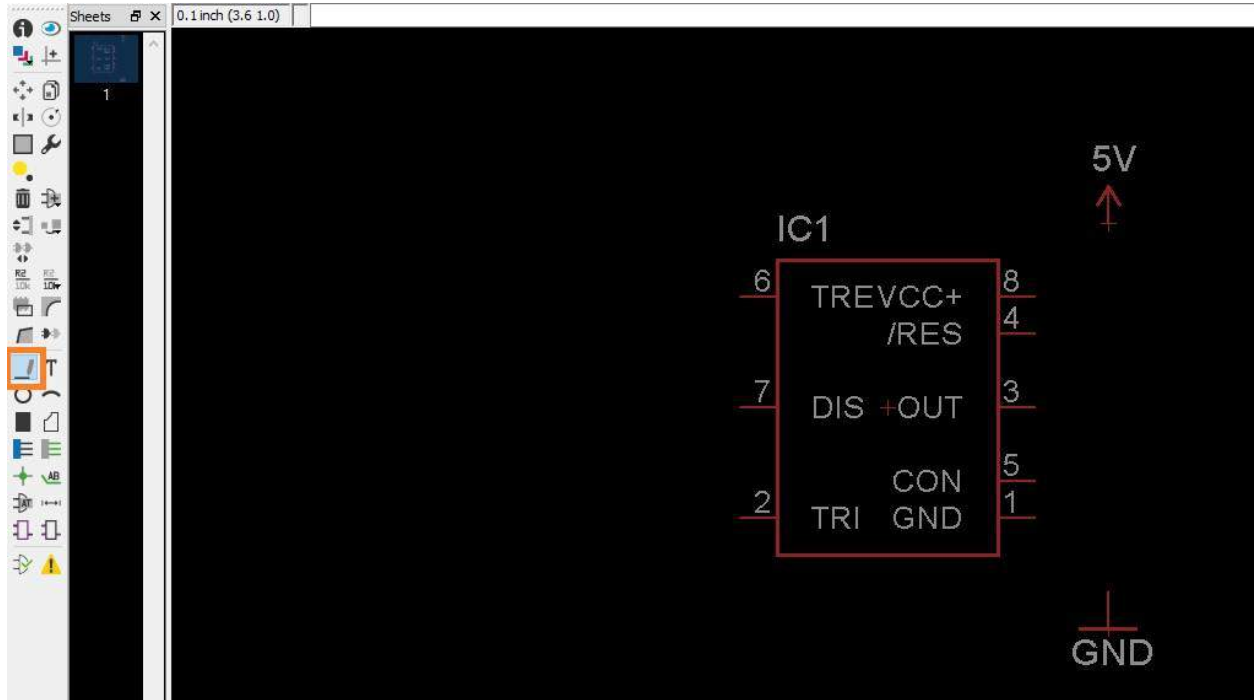
2. A new window should have opened. To make working in this window easier, make it full screen if it is not already. This is the schematic view for your project. In this view you can create a circuit schematic from components. Save this schematic as "main\_schem".
3. Let's add our first component, the NE555 IC. To add a component, click the  button in the toolbar on the left side of the window. Alternatively, you can use the menu at the top (Edit > Add).
4. If everything with your libraries has gone correctly, the library view will open. From here, you can select a component from any library you have chosen. You should see the SparkFun libraries in this view. Note in the figure below the SparkFun libraries are available. If you do not see these libraries, refer to the initial steps as well as the linked SparkFun tutorial to ensure that Eagle knows the location of the libraries, and you are currently using them.



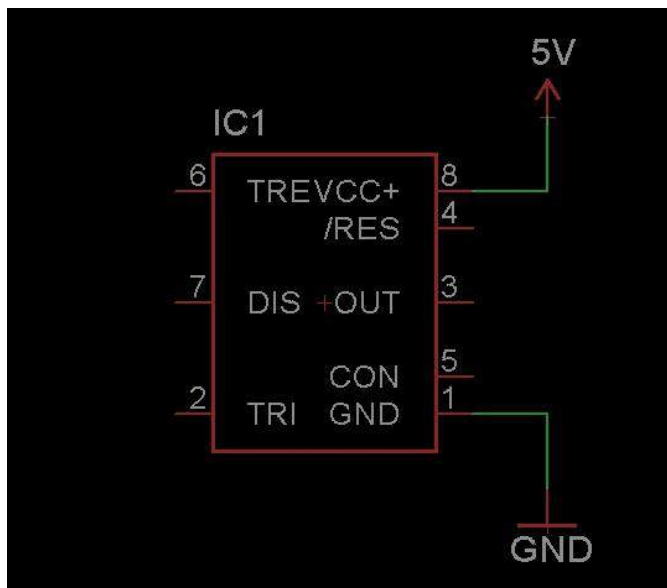
5. Navigating through these libraries can be intimidating, but the SparkFun Libraries are well laid out, and many of the categories are obvious. Expand the “SparkFun-AnalogIC” library, and find the NE555. Expand the NE555. Here you can see 3 different options. Notice that as you cycle through the options, the schematic and footprint are displayed. Each of these options presents the same component, but with a different physical footprint. The second is “NE555KIT”. Select the “NE555KIT” component, and click OK. The component is now attached to your mouse. Place the component in the middle of the schematic view. Once you have placed the component, click on the “Group” tool in the toolbar  to remove the component from your mouse pointer.
6. That was a lot of steps, so let’s make sure that everything worked out. At this point, your schematic view should have an NE555 timer IC schematic in the center, and you should have the “Group” tool selected. (See below for what your window should generally look like).



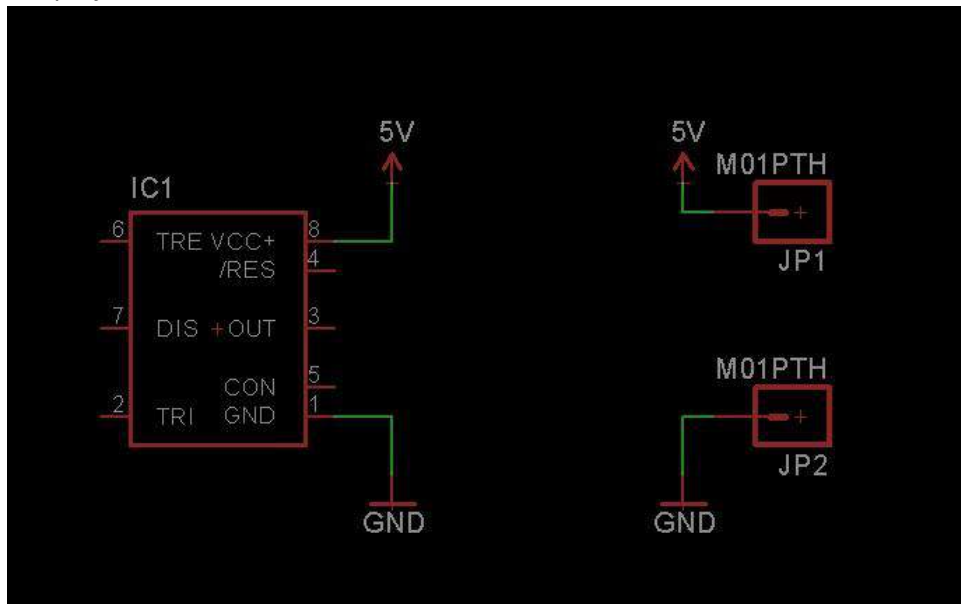
7. Now you should have familiarity with how to add components to the Schematic View. Using the same process for adding the NE555, add VCC and GND to the project. (These can be found under SparkFun-Aesthetics > 5V for VCC, and SparkFun > Aesthetics for GND).
8. Once these components are placed in the Schematic View, we need to connect them to the IC. Select the “Wire” tool from the toolbar (See below, on the left in the orange box).



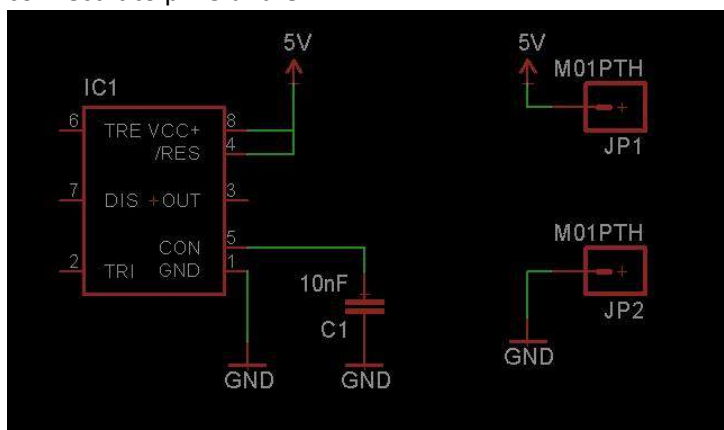
9. Once you have selected the wiring tool, click on pin 8 (VCC+), and then click on the connection for 5V. Press ESC to finish drawing the connection. You should see wire now connecting VCC+ and 5V. Also connect pin 1 (GND) to GND.



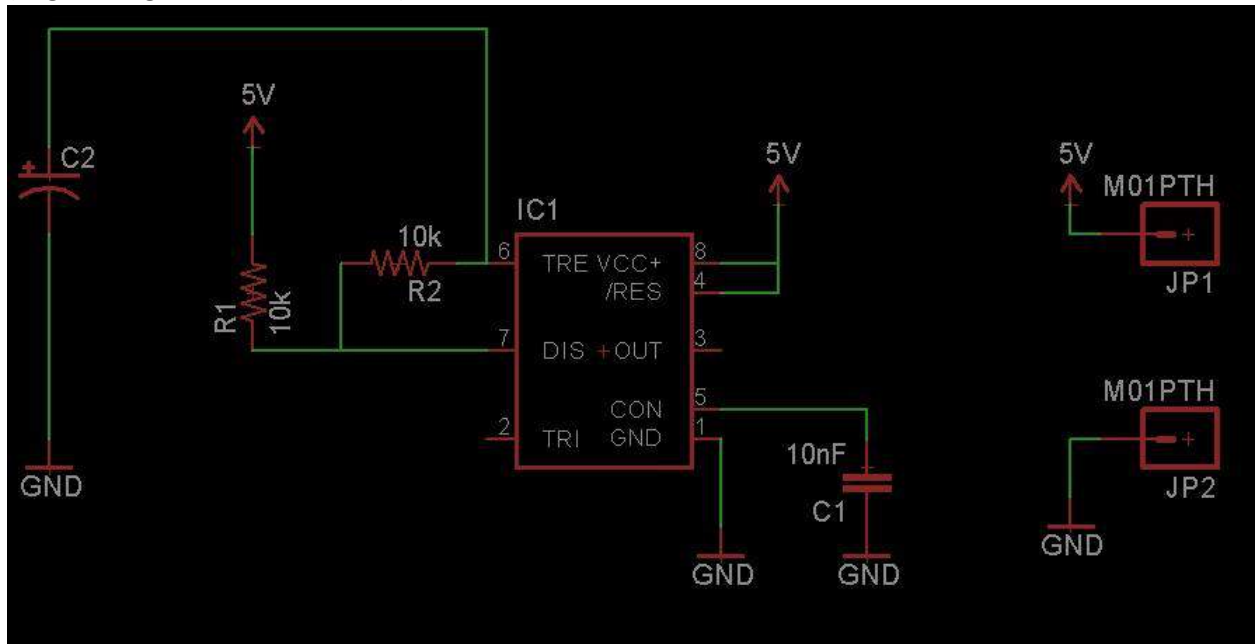
10. You now know how to make a schematic in Eagle! This same process of selecting a component, placing it, and then connecting it to the rest of the schematic is all that is needed to create any Eagle schematic. Let's finish adding the rest of the components that are going to make up our board.
11. 5V and GND do not have any footprint associated with them. They are symbolic, and make it easier to connect things together. For example, if you were to make a different connection to a different GND symbol, Eagle would recognize that the GND's are connected, and it would connect them automatically. We need to add something to physically connect our 5V and GND wires to. We will add two pins. (SparkFun-Connectors > M01 > M01PTH). Add two of these to the project. Connect one to 5V, and one to GND.



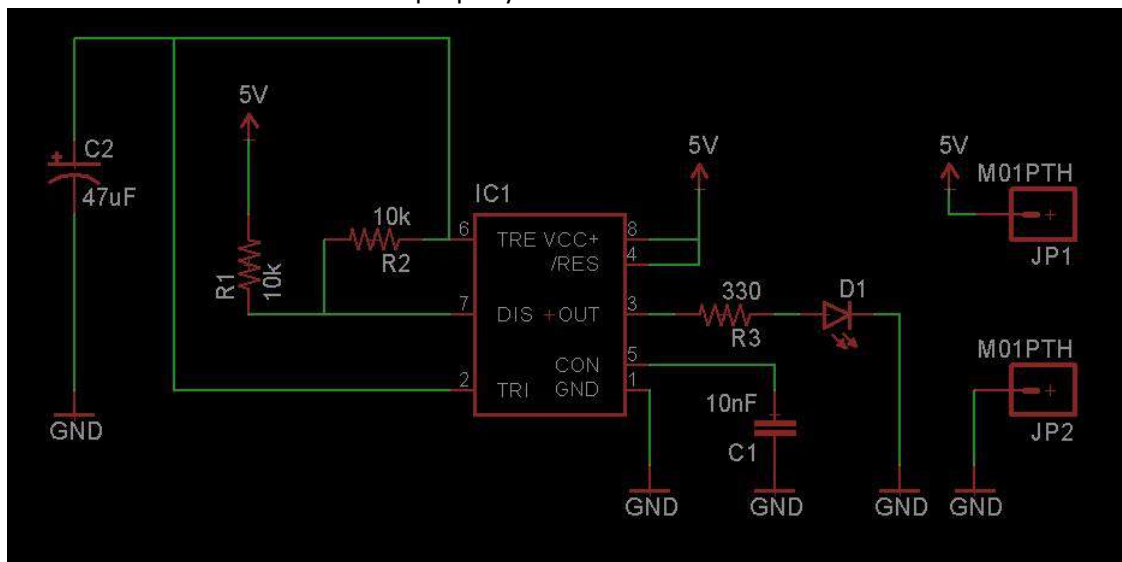
12. Now we can add the rest of our components. Pin 4 needs to be connected to VCC. Pin 5 needs to be connected to a 10nF capacitor, and the other side of the 10nF capacitor needs to be connected to ground. (SparkFun-Capacitors > CAP > CAPPTH). When we add this capacitor, it is a generic footprint. This means it has no value associated with it. To add a value we use the tool ("Value" tool). Click on the capacitor with the value tool selected, and enter "10nF". Then connect it to pin 5 and GND.



13. Now we need to build something a little trickier. We need a  $10\text{k}\Omega$  resistor from VCC to pin 7, another  $10\text{k}\Omega$  resistor from pin 7 to pin 6, and then a  $47\mu\text{F}$  capacitor from pin 6 to GND. The two  $10\text{k}\Omega$  resistors can be found in SparkFun-Resistors >  $10\text{KOHM}-1/4\text{W}5\%(\text{PTH}) > 10\text{KOHM}-1/4\text{W}5\%(\text{PTH}) \text{ AXIAL}-0.3$ . The capacitor can be found under SparkFun-Capacitors > CAP\_POL > CAP\_POLPTH2. You will need to assign this capacitor the correct value ( $47\mu\text{F}$ ) just as you did with the  $10\text{nF}$  capacitor. Also, this capacitor is polarized. Ensure that is connected correctly (negative sign connected to GND).





14. Now for the final connections. Connect pin 2 to pin 6. Then connect pin 3 to a  $330\Omega$  resistor. Connect the other side of the resistor to the anode of an LED, and connect the cathode of the LED to GND. You can use the same resistor as before, and then just change the value to “330”. The LED is found under SparkFun-LED > LED > LED5MM. The LED needs to be placed in the correct direction for it to function properly.

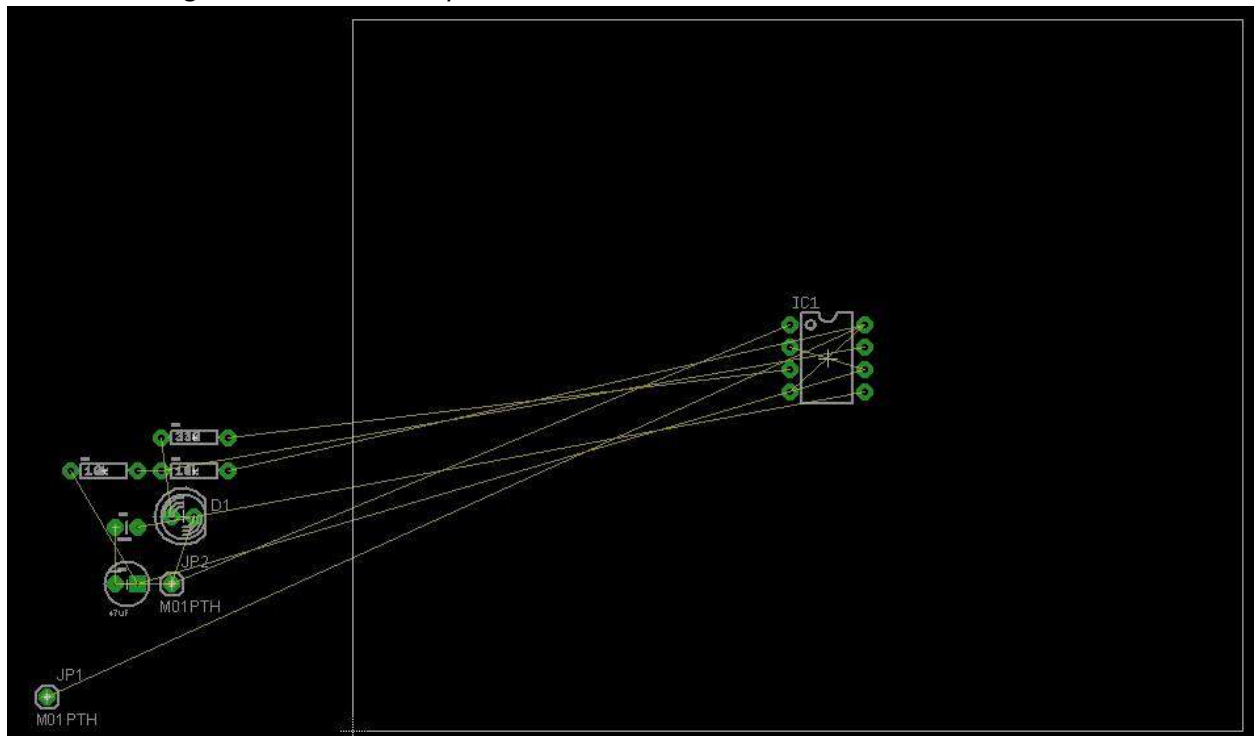


This concludes the schematic portion of the tutorial. You should feel comfortable adding and connecting components, as well as changing their values. You should understand that all GND's are connected, and all 5V supply connections are also connected. This principle extends to naming the pins and wires coming from another part of the schematic as well. If they share a name, Eagle will connect them.

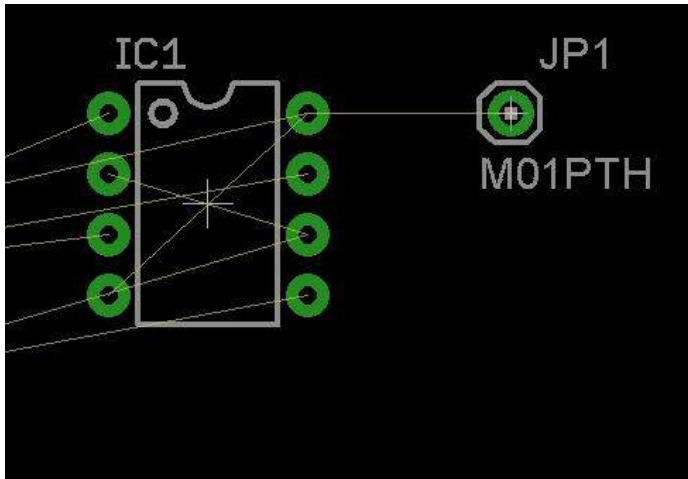
## Making a Board Layout


At this point you should have a schematic ready to go from the schematic portion of the workshop above.

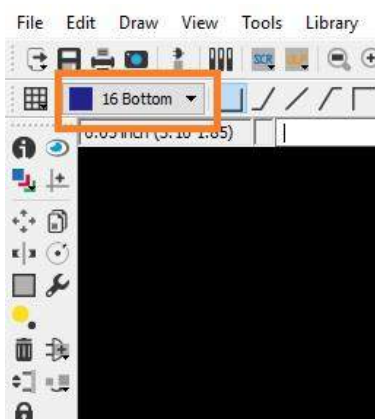
1. In the toolbar at the top, click the  button (Generate/switch to board). Alternatively, find this under File > Switch to Board. Click yes when it asks if you would like to generate a board.
2. A new window will have opened. There should be a mess in the one corner. That mess is all of our components footprints, or how they will look on the board. The rectangle in the center is the outline of our board. Finally, all green lines that are criss-crossing between the components are connections that have not yet been made. Select the move tool (). Each component has a cross in the center. When selecting in board view, this cross is where to click.
3. Using the move tool, click on the NE555 footprint. Move the NE555 to the middle of the board. Right click while the component is selected to rotate the component. When finished, you should have something that looks like the layout below.



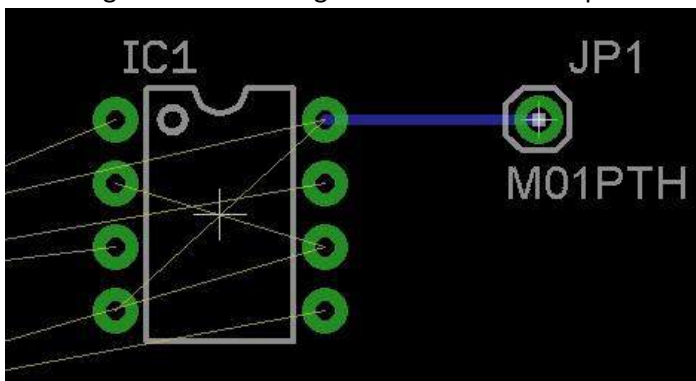
- Now let's make some easy connections between components. Select the single jumper (JP1 or JP2) which is connected to pin 8. Move it, as you did with the NE555 IC footprint, to a position closer to pin 8.



- Now, click the  button (Route) in the left toolbar. Up at the top left, ensure that you have the bottom layer selected.



- Now let's route! Click on the jumper that you placed, and then click on pin 8. You will see the route follow your mouse. If a successful connection has been made, an auditory cue is given, and the green line showing an unconnected component will disappear.



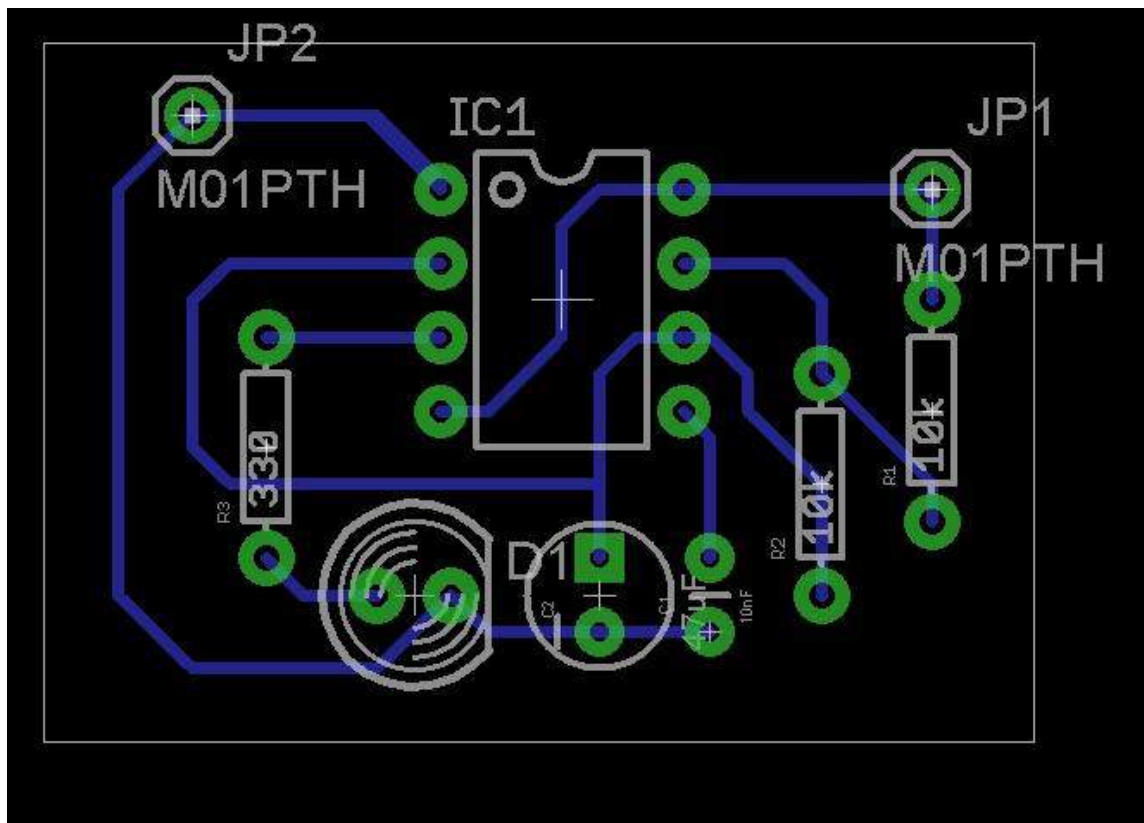


7. That is the process for laying out a board! Now all that is left to do is to layout the rest of the board. Now note, if two traces cross each other, **they will make an electrical connection on your PCB**. You need to take care that you do not have traces overlapping. With that being said, try to move the rest of your components so that you can connect everything without overlapping traces. Once you have positioned your components, try and route everything. Alternatively, you can look at the image below and route your PCB like that. Here are some tips and guidelines for routing a PCB in Eagle.

- a. When a route is in progress, change the bend of the route by right clicking. This can also be done by selecting these options at the top toolbar.



- b. Try to avoid 90 degree angles and sharp bends in the routing. Note the 45 degree angles in the example below.
- c. Always try and position your footprints before beginning to route. You will almost definitely move them around later, but it helps to have them laid out first.
- d. Always be thinking about how the component will fit once it is soldered into position. Make sure to leave enough room between traces and components to make assembly easy for yourself later on.



8. Once you are finished routing, using the move tool, select each corner of the PCB's rectangle, and bring in the corners so that they cover all the components and routing, with room to spare on the edges (This has already been done in the image above).

That's it for routing! The basics of routing are very simple. Just place your components, connect your components, and ensure there are no overlapping traces. However, routing correctly can quickly become complicated, and many things must be learned through experience. See the Appendix of this tutorial for more information on routing best practices.

## Generating Files for Fabrication

Now that we have finished laying out our board, it is time to send it to a manufacturer. To generate the files you need, first download the zip folder at the following link:

<https://cdn.sparkfun.com/assets/c/1/9/8/2/52056b19757b7f795b2a561c.zip>

Once you have downloaded the zip folder, extract it's contents. Inside the cam folder, there is a single file.

1. To generate the files we need, we need to open the CAM Processor (File > CAM Processor).
2. Next, we will use the file that Sparkfun has created. Create a new job by selecting File > Open > Job
3. When the window opens to select a new job, browse to the file that you downloaded before step 1, and open it.
4. Now, just click "Process Job" at the bottom of the window, and let it process. It shouldn't take very long. Once it is done, you have completed creating the Gerber Files!
5. Put the files with the following extensions into a zip folder: .dri, .GBL, .GBO, .GBS, .GML, .gpi, .GTL, .GTO, .GTP, .GTS, .txt

Talk to the technician to see how to submit your files and get your board fabricated!

## Conclusion

That's it for this tutorial. Now you should have a completed board laid out, and ready to be fabricated! Different companies have different requirements for what files to generate and give to them. Check with the fabricator to see what they require, and if they have any design rules you have to follow!