Introduction to Eagle

UDComp Lab Spring Break Research Experience - 03/23/2021

Instructor: Jessica de Souza

About me

- ECE Department, Medical Devices and Systems;
- Graduated in Telecommunications Engineering;
- I love working with electronics and embedded systems;
- Let’s grab some coffee: jdesouza@eng.ucsd.edu

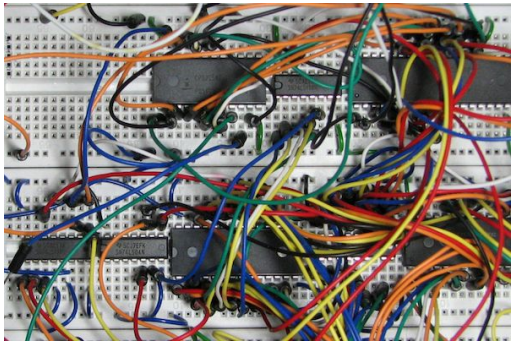
What to Expect from Today

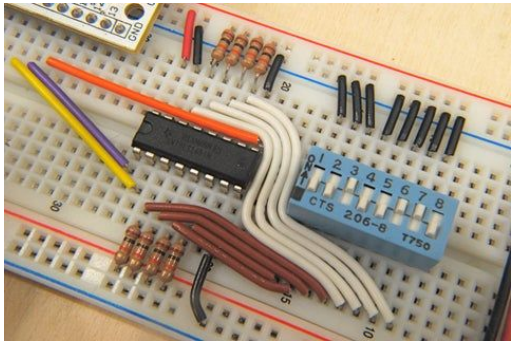
Today we will:

- Learn why it is important to design circuits;
- Learn about Eagle from Autodesk;
- Design our first schematic and board;
- Learn some design tips;
- What to do if you want to manufacture;

Why should I do that?

You want to keep your project neat and easy to identify components and the circuit flow:

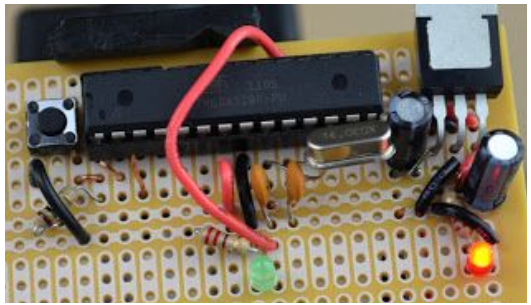


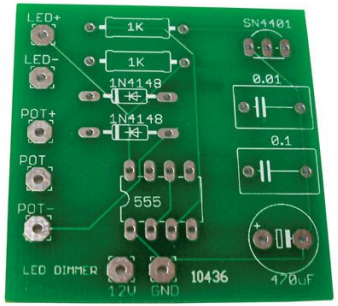


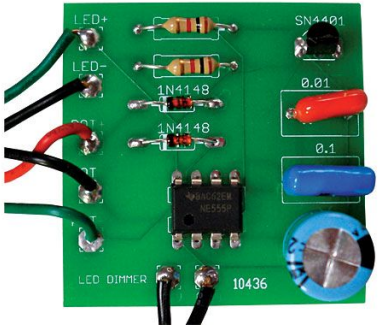
Which one is easier to debug?

Why should I do that?

In research, is common to upgrade your circuit with more professional and robust designs, especially if you will be working with the circuit for a long time.







Which is why we like to build PCBs!

Why should I do that?

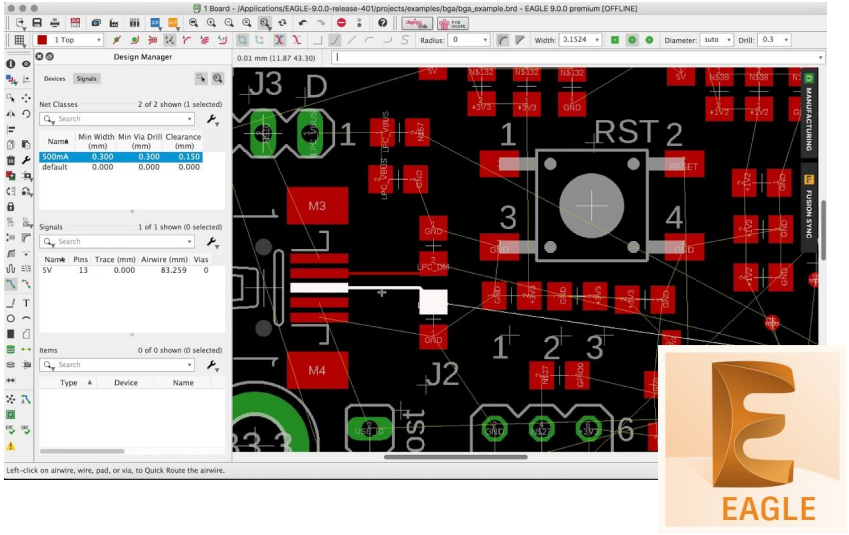
When working with hardware, the following things will make your life easier:

- Easier board debugging;
- Proper soldering;
- Parallel build + testing;
- Keep your multimeter close;
- Good quality materials = - interference and - board errors;

About Autodesk Eagle

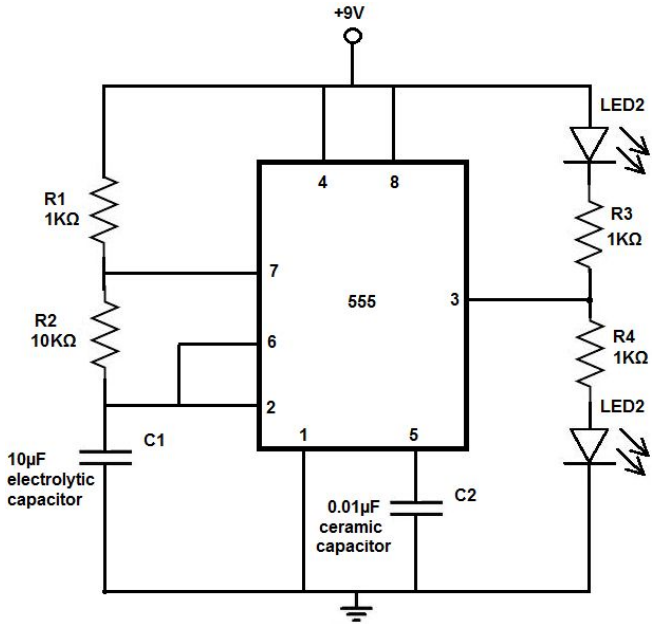
Eagle is an Electronic Device Automation (EDA) software that allows you to:

- Design Printed Circuit Boards (PCBs);
- Create schematics of your circuit;
- Place components;
- Generate the gerber file and have it ready to manufacture.



There are many other options for circuit design, I left a list of other software options at the end of this presentation.

The Circuit: LED flasher with 555 IC



Source: http://www.learningaboutelectronics.com/Articles/LED-flasher-circuit.php

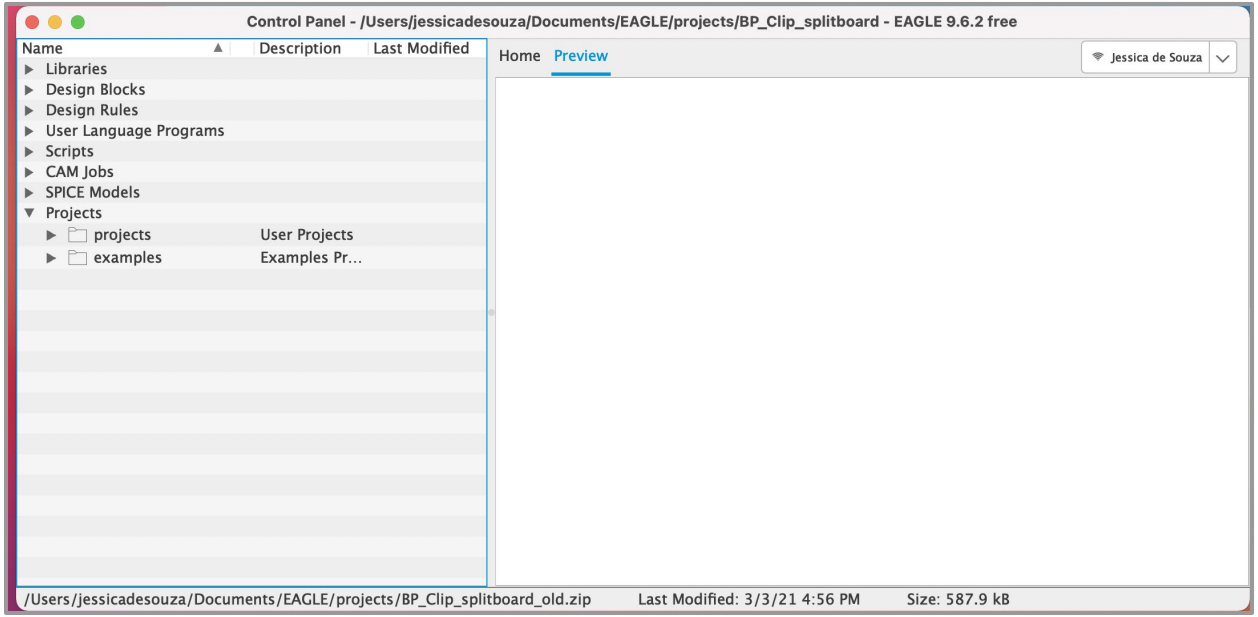
Component list for Eagle:

- 555 timer IC: ICM7555 (1 unit)
- LED: LED5MM (2 units)
- 9V battery: AB9V (1 unit)
- 1KΩ Resistor: R-US_0204/7 (3 units)
- 10KΩ Resistor: R-US_0204/7 (1 unit)
- 10µF Capacitor: CPOL-EUE5-10.5 (1 unit, elect.)
- 0.01µF Capacitor: C-EU075-052X106 (1 unit, cer.)

Let’s get started!

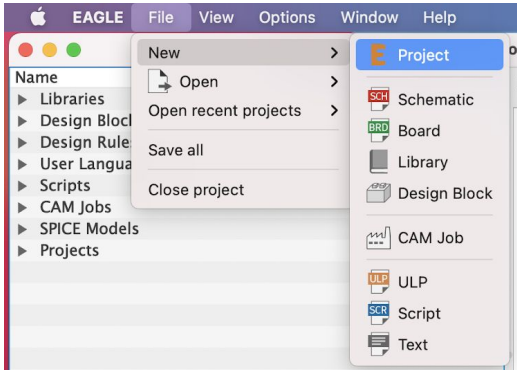
1. Open Eagle, you will see the following screen

2. Control Panel View

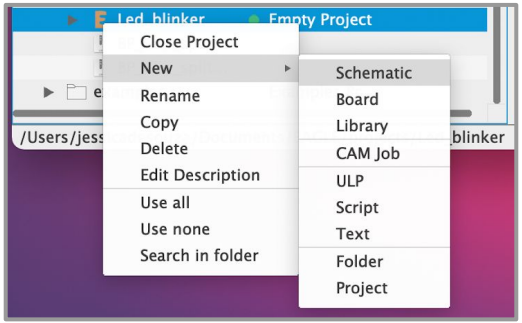


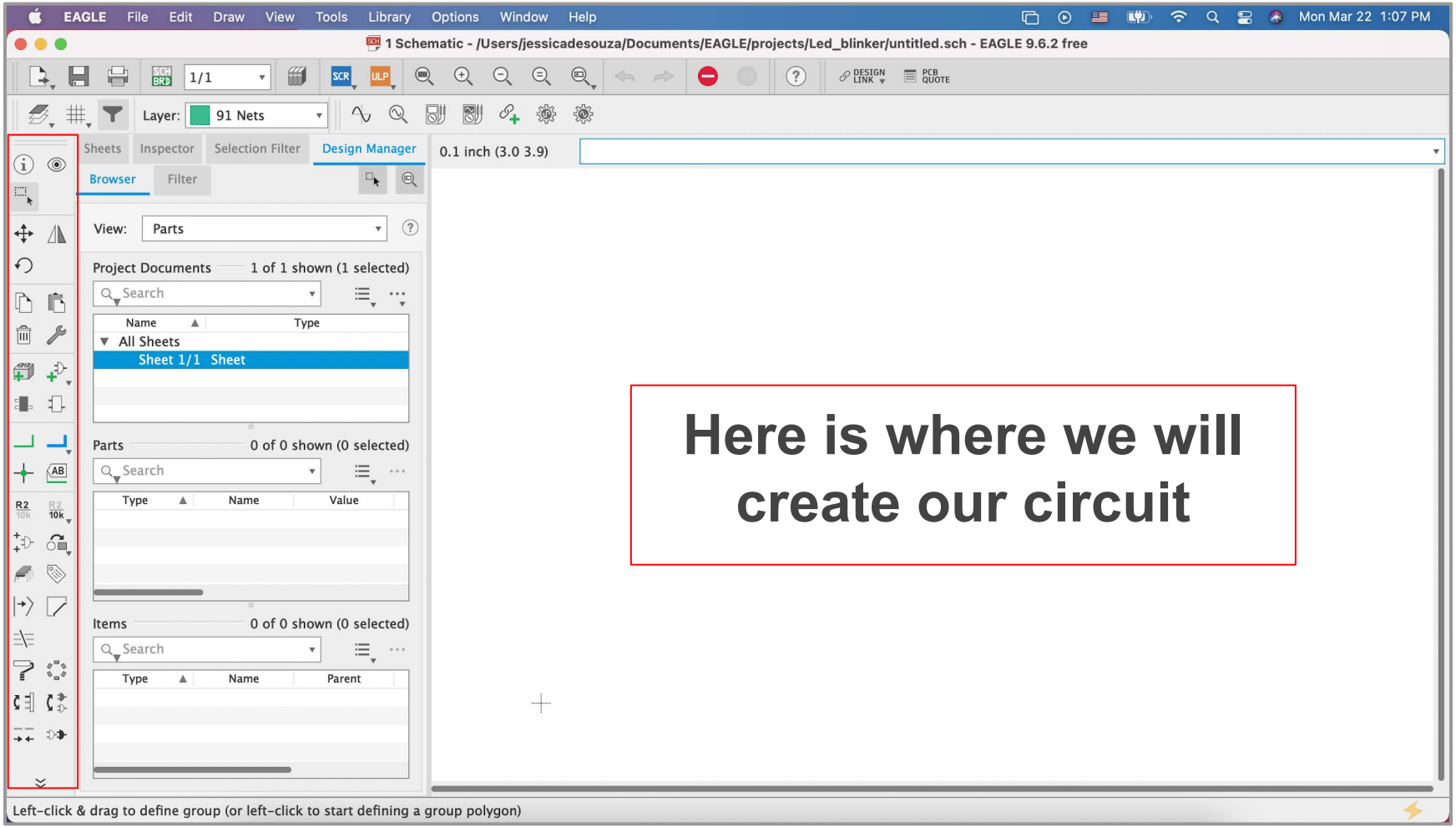
3. Create a new project:

Project name: Led_blinker

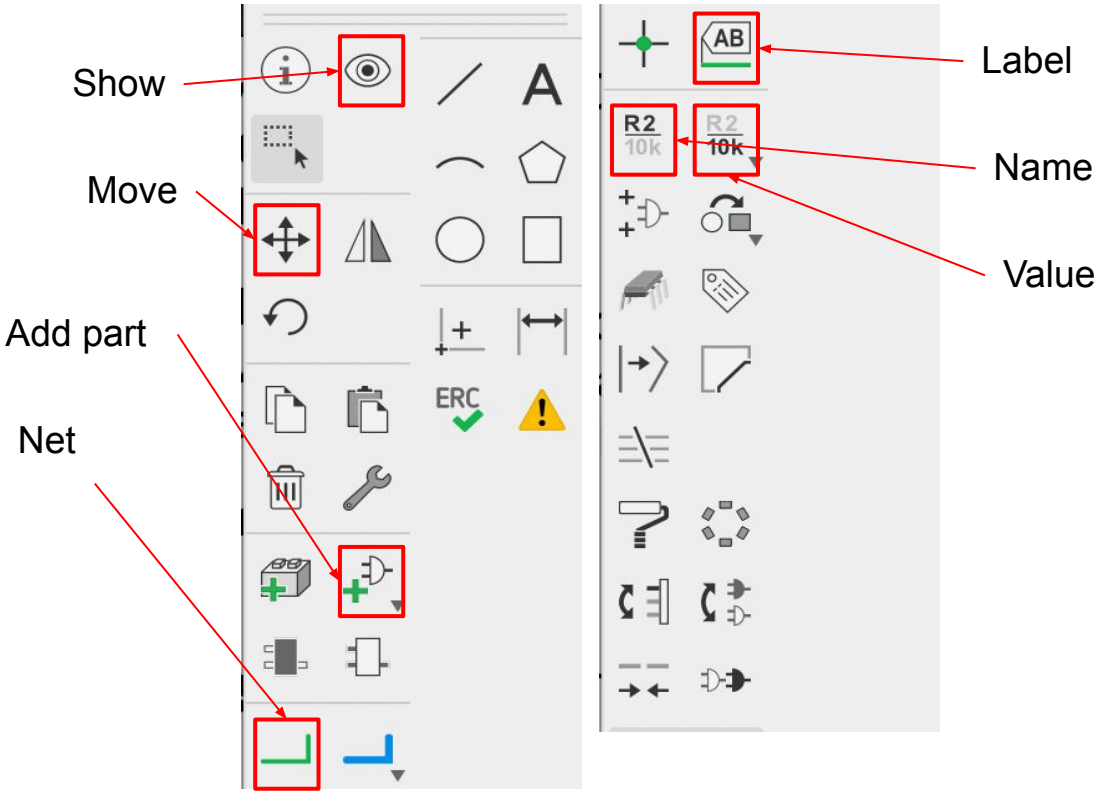


4. Inside your project, create a schematic:

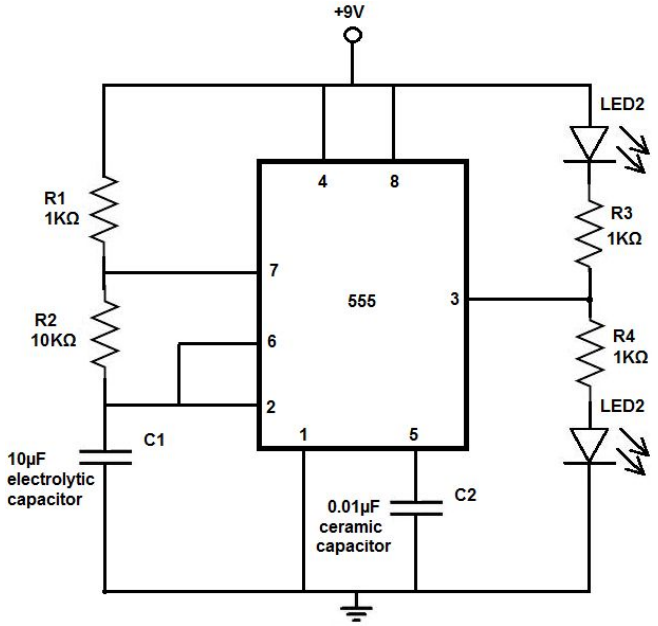



EAGLE File Edit Draw View Tools Library Options Window Help
Mon Mar 22 1:07 PM
1 Schematic - /Users/jessicadesouza/Documents/EAGLE/projects/Led_blinker/untitled.sch - EAGLE 9.6.2 free
Layer: 91 Nets
Sheets Inspector Selection Filter Design Manager
Browser Filter
View: Parts
Project Documents 1 of 1 shown (1 selected)
Search
Name Type
All Sheets
Sheet 1/1 Sheet
Parts 0 of 0 shown (0 selected)
Type Name Value
Items 0 of 0 shown (0 selected)
Type Name Parent
0.1 inch (3.0 3.9)
Here is where we will create our circuit
Left-click & drag to define group (or left-click to start defining a group polygon)

Tools That we will be using for schematic:



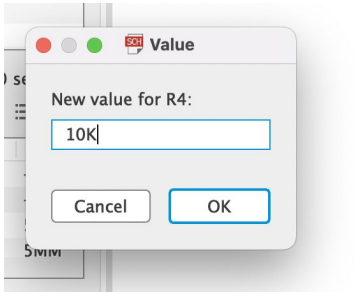
Getting the Components we need



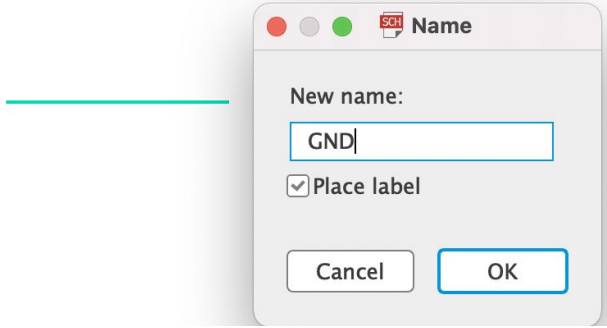
Component list for Eagle:

- 555 timer IC: ICM7555 (1 unit)
- LED: LED5MM (2 units)
- 9V battery: AB9V (1 unit)
- 1KΩ Resistor: R-US_0204/7 (3 units)
- 10KΩ Resistor: R-US_0204/7 (1 unit)
- 10µF Capacitor: CPOL-EUE5-10.5 (1 unit, elect.)
- 0.01µF Capacitor: C-EU075-052X106 (1 unit, cer.)

Changing the R values to resistance

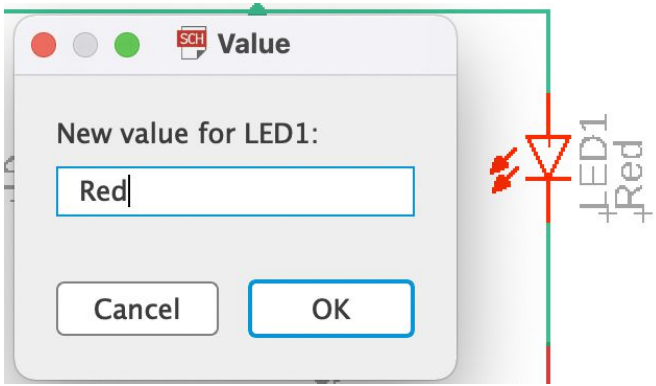


Naming the wires

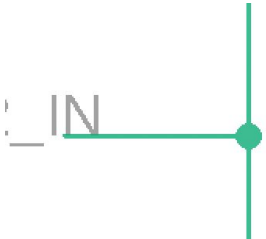


Verify the connections using the show button

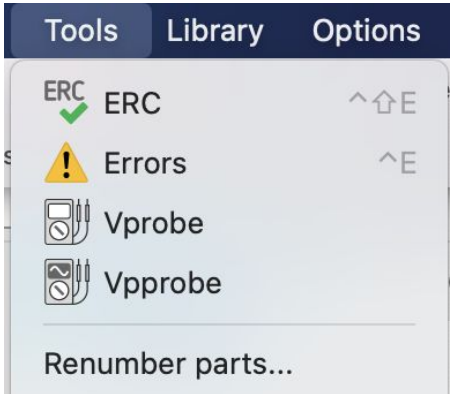
Changing the LED values to its color



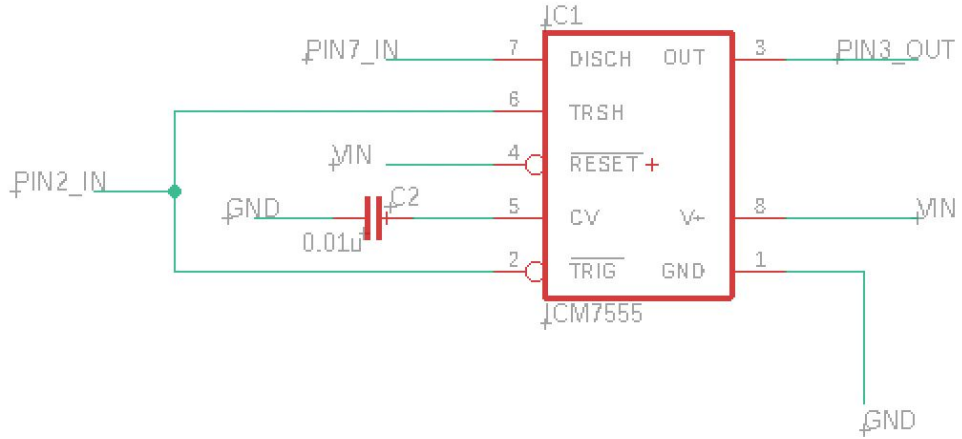
Junctions are okay in schematic

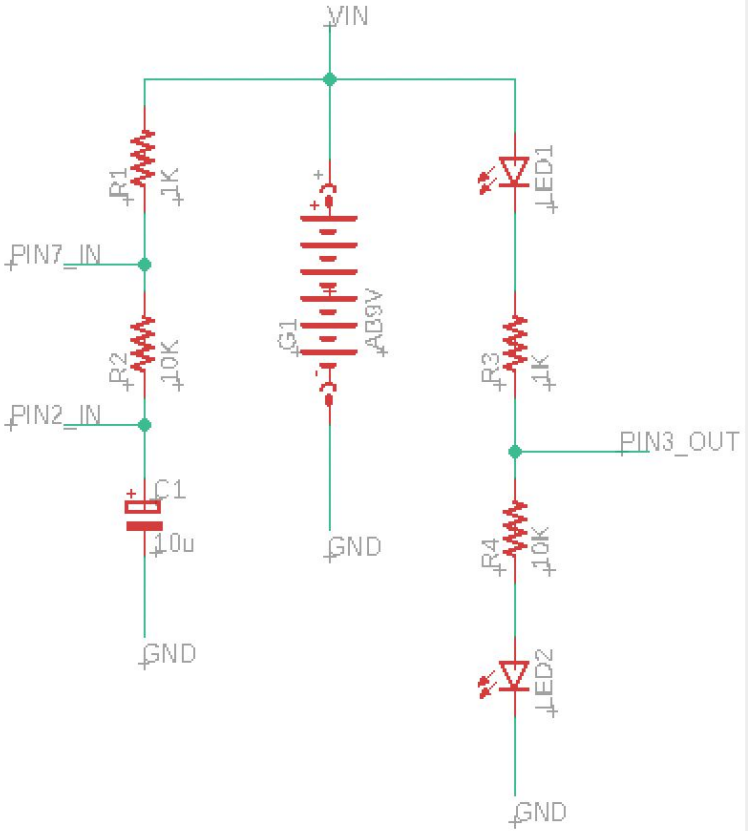


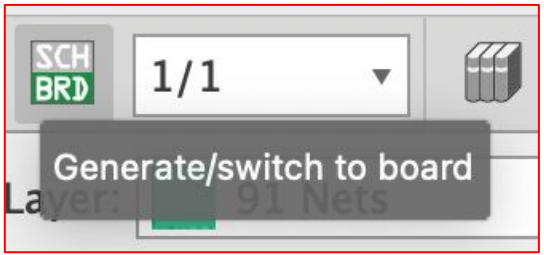
Verify if there are any errors



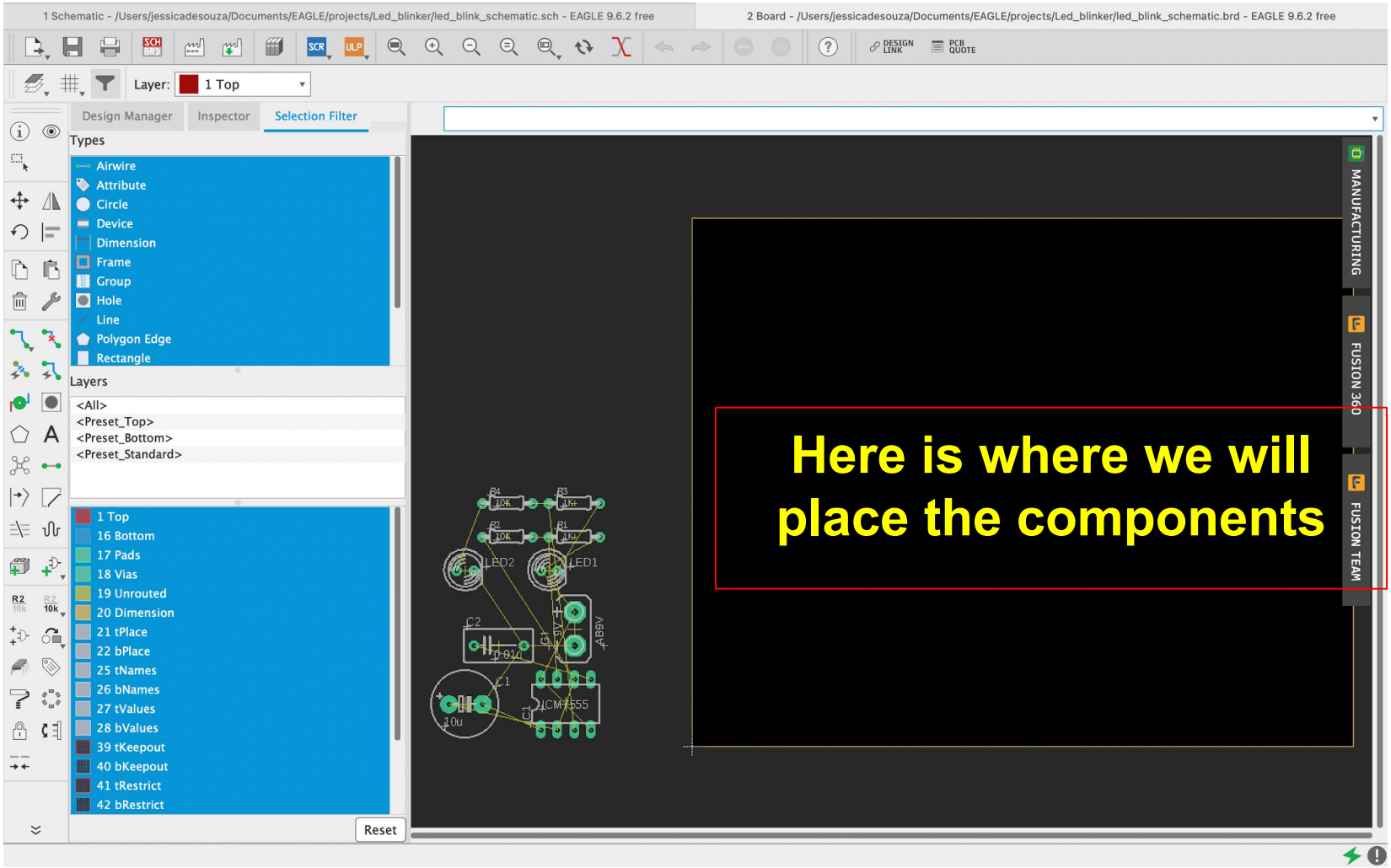
The final result:



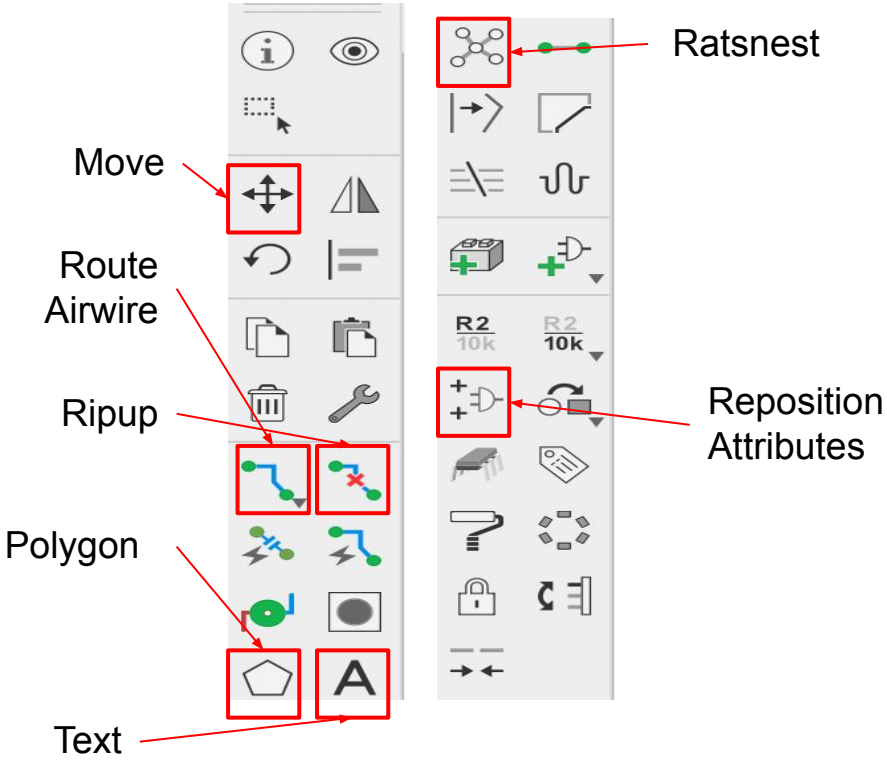




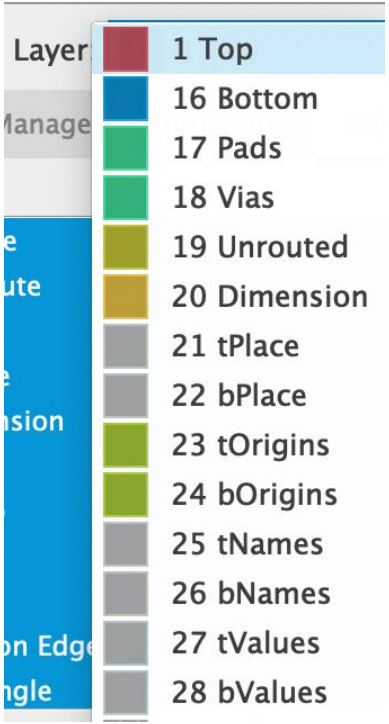
Let’s assemble the board


1 Schematic - /Users/jessicadesouza/Documents/EAGLE/projects/Led_blinker/led_blink_schematic.sch - EAGLE 9.6.2 free
2 Board - /Users/jessicadesouza/Documents/EAGLE/projects/Led_blinker/led_blink_schematic.brd - EAGLE 9.6.2 free
DESIGN LINK
PCB QUOTE
Layer: 1 Top
Design Manager
Inspector
Selection Filter
Types
Airwire
Attribute
Circle
Device
Dimension
Frame
Group
Hole
Line
Polygon Edge
Rectangle
Layers
<All>
<Preset_Top>
<Preset_Bottom>
<Preset_Standard>
1 Top
16 Bottom
17 Pads
18 Vias
19 Unrouted
20 Dimension
21 tPlace
22 bPlace
25 tNames
26 bNames
27 tValues
28 bValues
39 tKeepout
40 bKeepout
41 tRestrict
42 bRestrict
Reset
MANUFACTURING
FUSION 360
FUSION TEAM
Here is where we will place the components

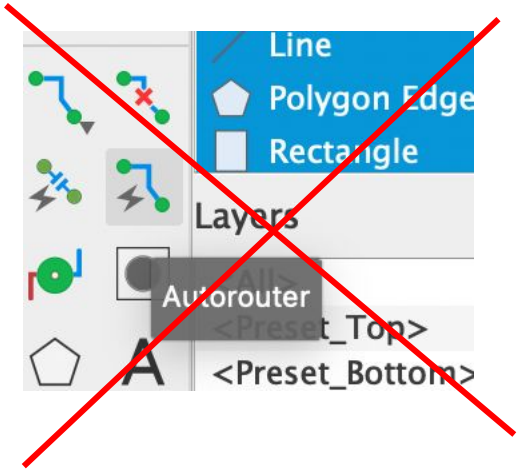
Tools that we will be using for the board



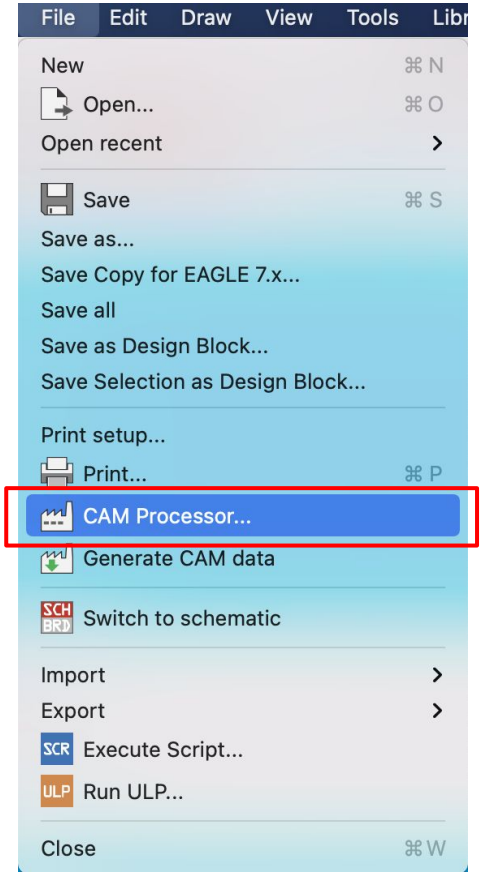
For multilayer development



Avoid using Autorouter



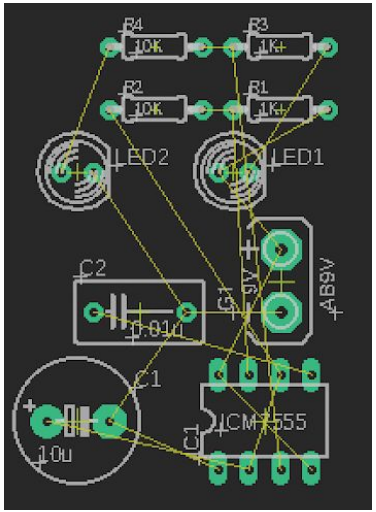
Generating Gerber File

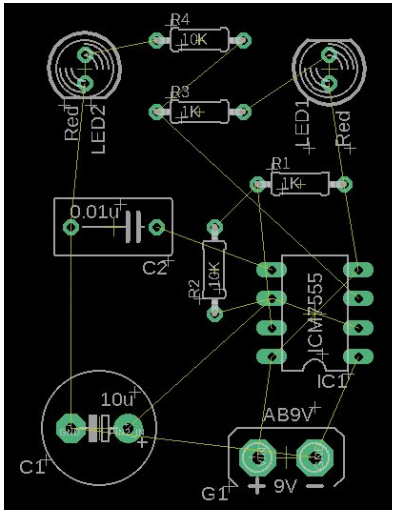


Play with the components arrangements

- Avoid too much crossed lines;
- Keep your components close if you can;
- It is always a good idea to keep one identification of the component in the board;

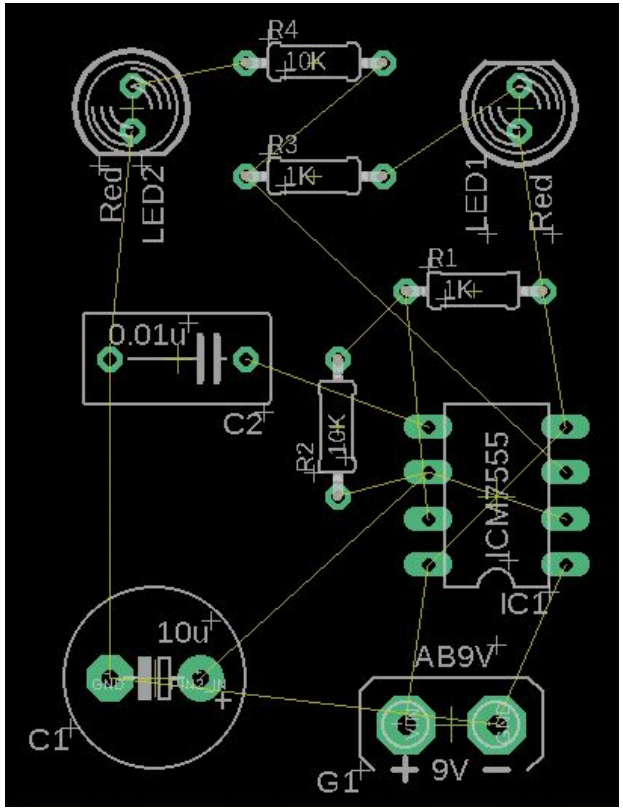
Bad



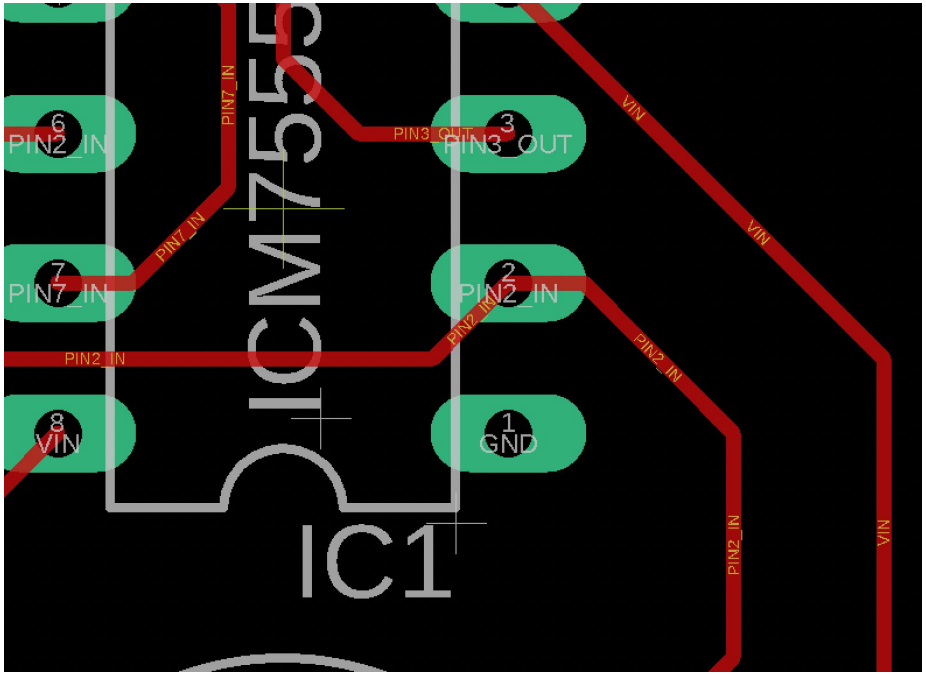


Good

This was my arrangement:



Placing the wires

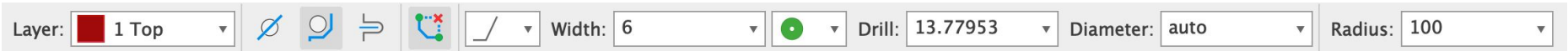


Avoid 90 degrees connections for better electron flow;

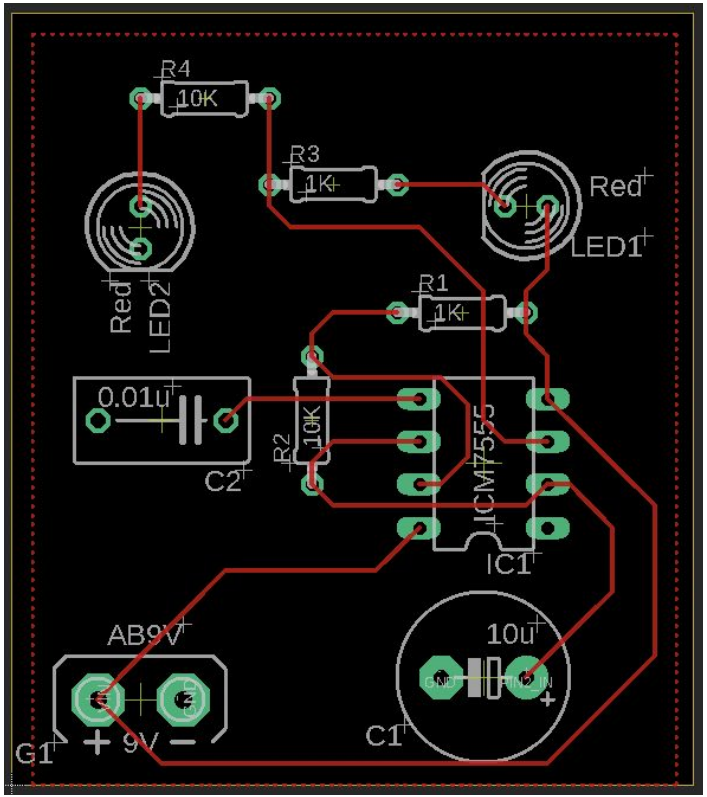
If your components allow, is okay to draw wires through them;

Wire width around 10 mils is fine, but your application may require more (for larger current);

If you need to cross a wire through other, you may need to use a via (for multilayer boards)



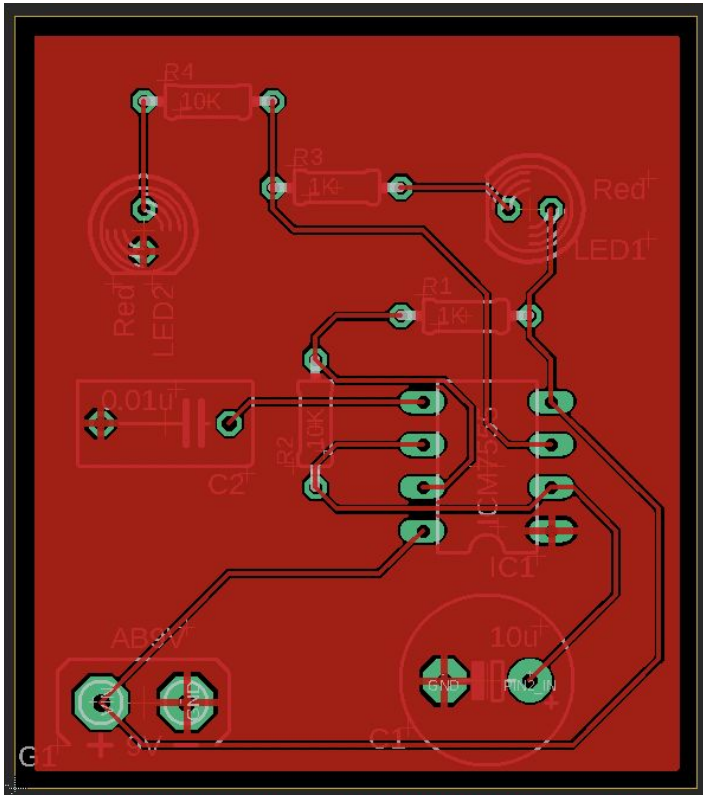
This is the final result:



Why is the ground not connected to anything?

Note the dotted rectangle around the circuit, that is our ground plane.

This is the final result:

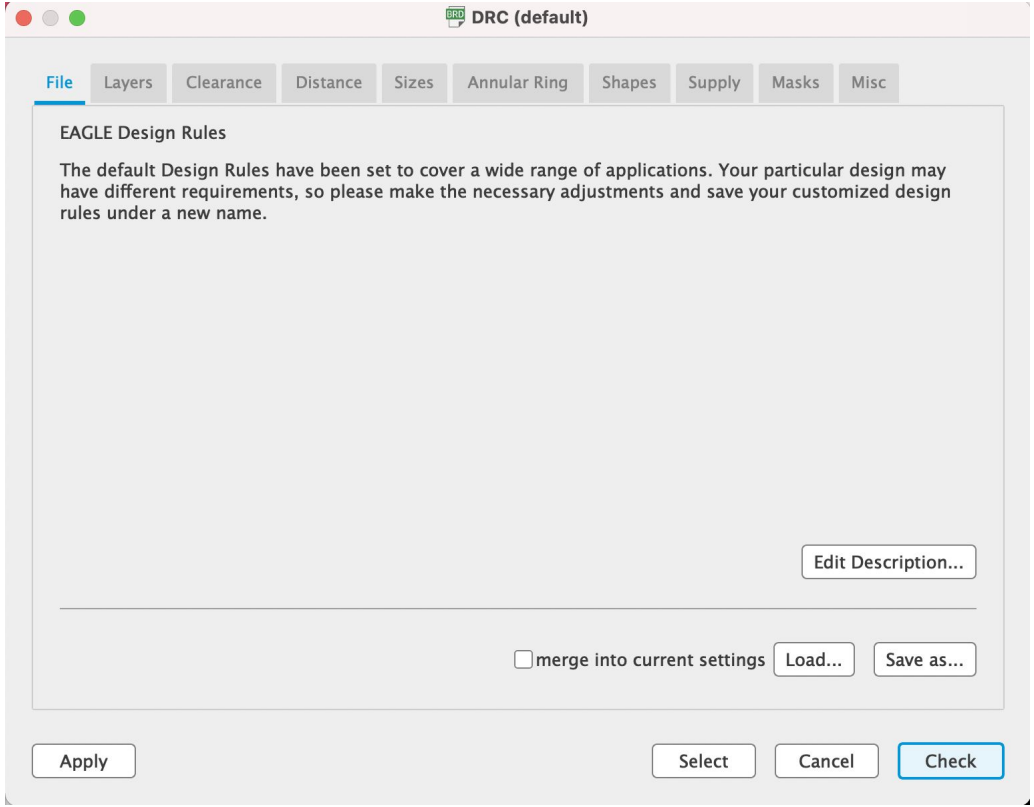


Why is the ground not connected to anything?

Note the dotted rectangle around the circuit, that is our ground plane.

Press ratsnest after creating the rectangle and assigning a signal to it, and the entire board will get red.

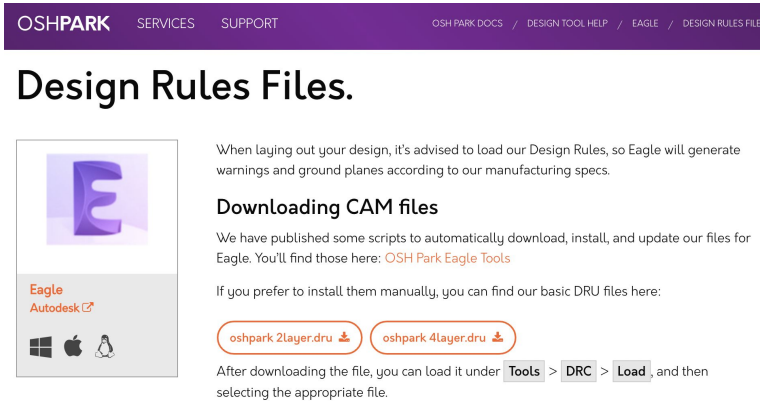
Next steps



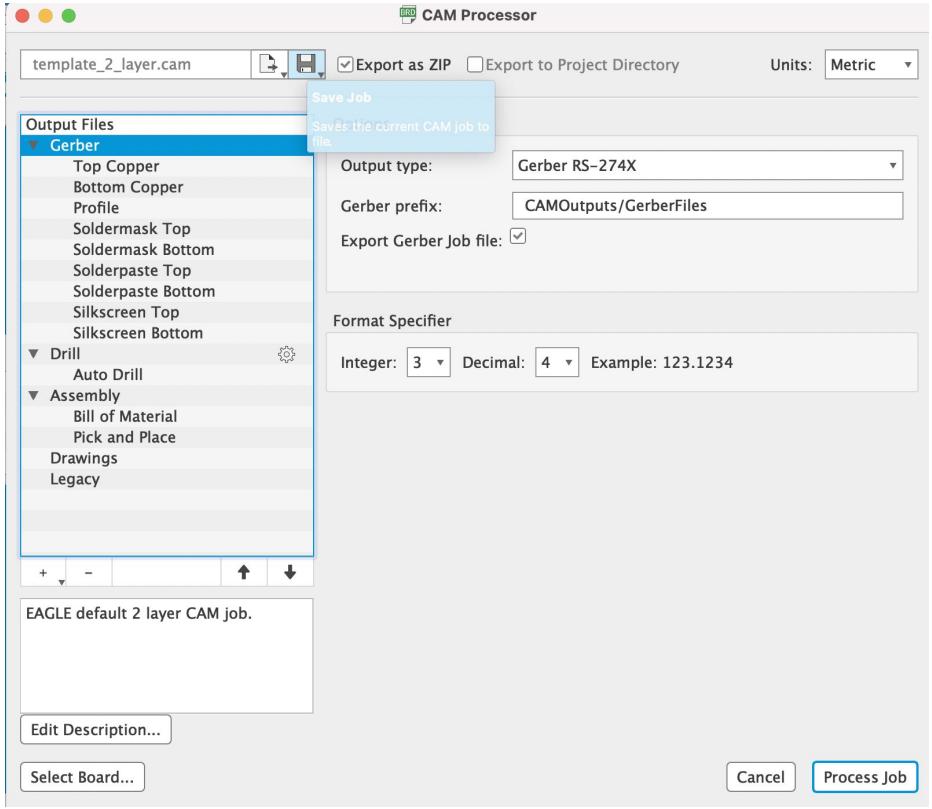
Design Rule Check:

There are several manufacturers who let you download specific design rules.

Otherwise it will run the general DRC and if there are errors it will let you know.



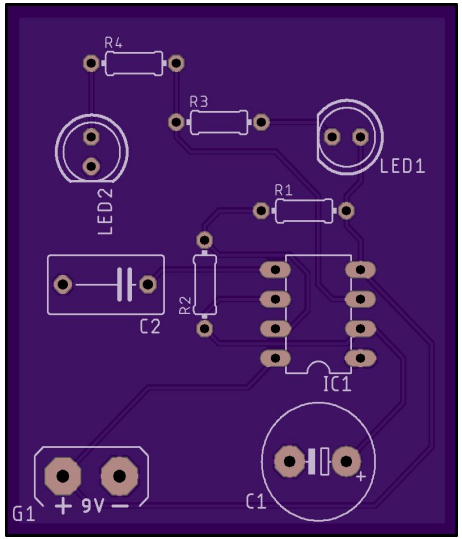
Exporting the Gerber file for Manufacturing

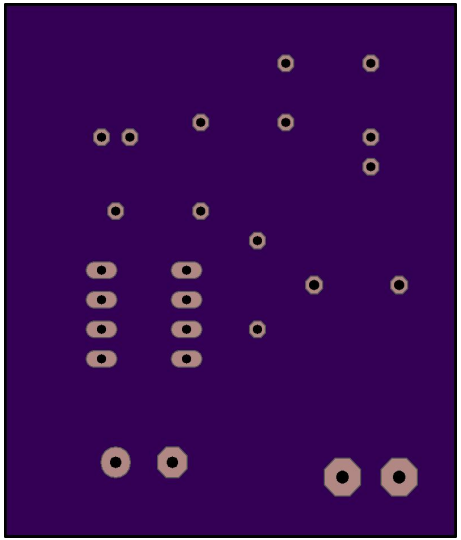


Extra: ordering your design

Once you upload your gerber, this is what will happen:

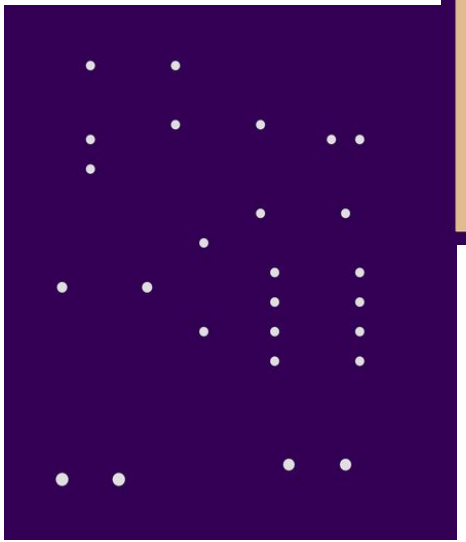
Top



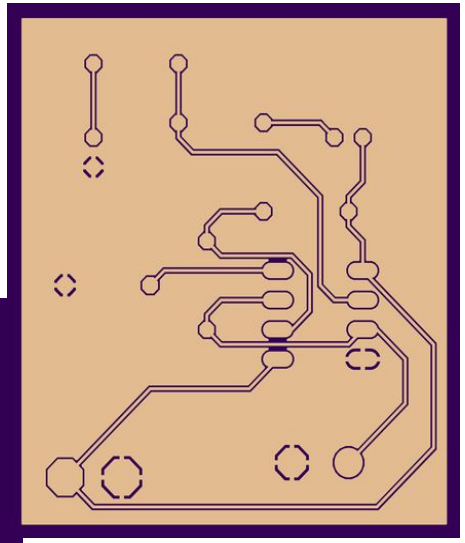


Bottom

Drills



Copper Layer



Resources

Circuit Design Software:

- Altium
- Eagle
- EasyEDA
- Fritzing
- KiCad
- NI Multisim
- Proteus

Board Manufacturers:

- Advanced Circuits
- OSH Park
- Hughes Circuits

Device Components and Libraries:

- SnapEDA
- Digikey
- Sparkfun