Creating PCBs in Eagle (Fusion 360)

Learn the basics of creating PCB schematics, layout, and Gerber files with Eagle (Fusion 360).

Written By: Ho Yin Calvin Leung

This document was generated on 2022-07-21 06:04:29 AM (MST).

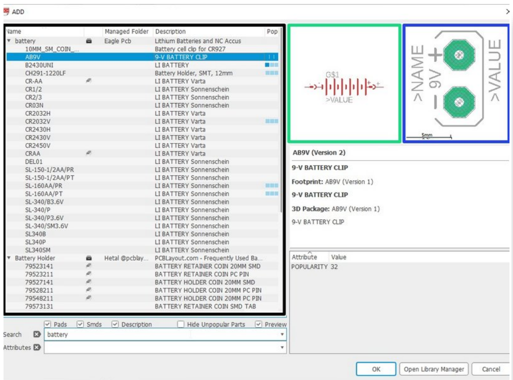
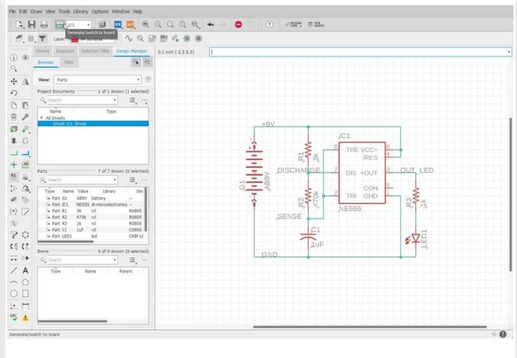
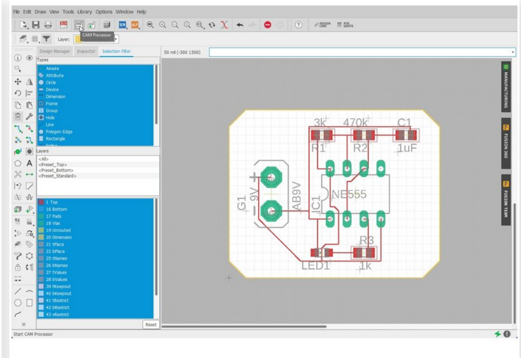
Introduction

EAGLE is electronic design automation (EDA) software that lets printed circuit board (PCB) designers seamlessly connect schematic diagrams, component placement, PCB routing, and comprehensive library content.

Install Eagle here:

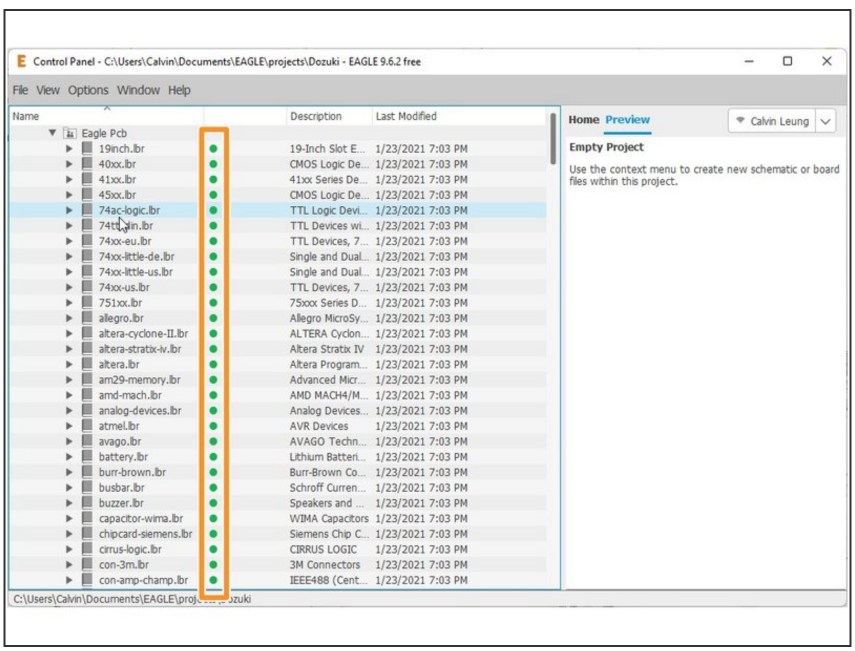
https://www.autodesk.com/products/eagle/...

Step 1 — Overview of PCB Design Process in Eagle

- Eagle schematics and layouts are made up of parts. Each part has a symbol as well as a footprint.
- Symbols are used for creating a schematic by connected components.
- Footprints are used for planning the PCB layout after a schematic is completed.

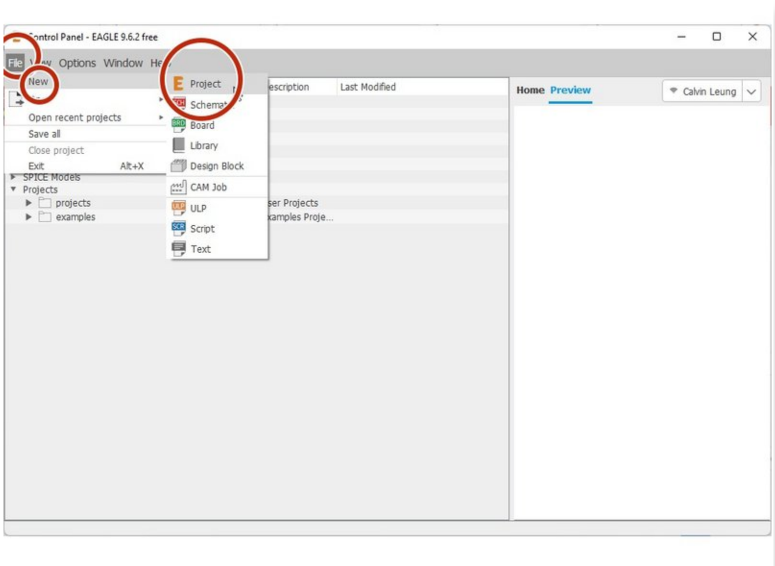
Step 2 — Overview of Eagle Libraries

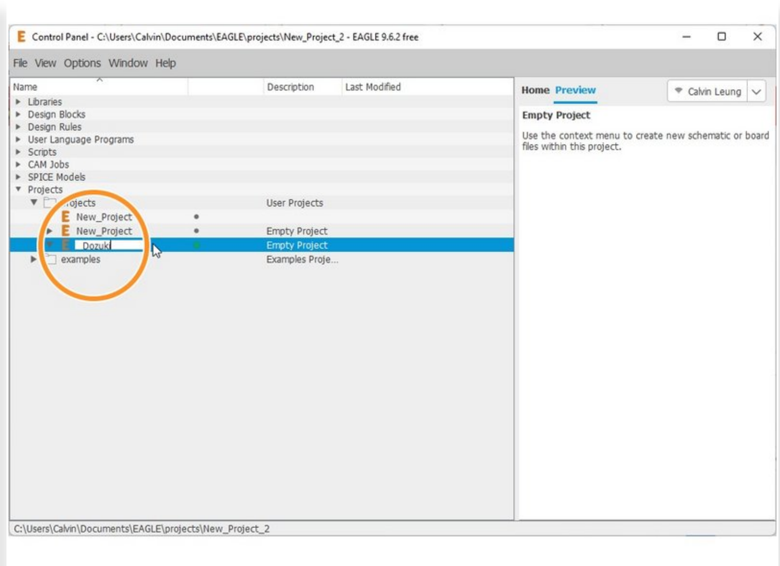


- We mentioned how parts and their symbols/footprints are used for schematics and layouts.
- We don't have to start from scratch when designing PCBs.
- A lot of common parts are included in free libraries and using them would save you a lot of time.
- Make sure when using a library, the dot next to the library is green.

This document was generated on 2022-07-21 06:04:29 AM (MST).

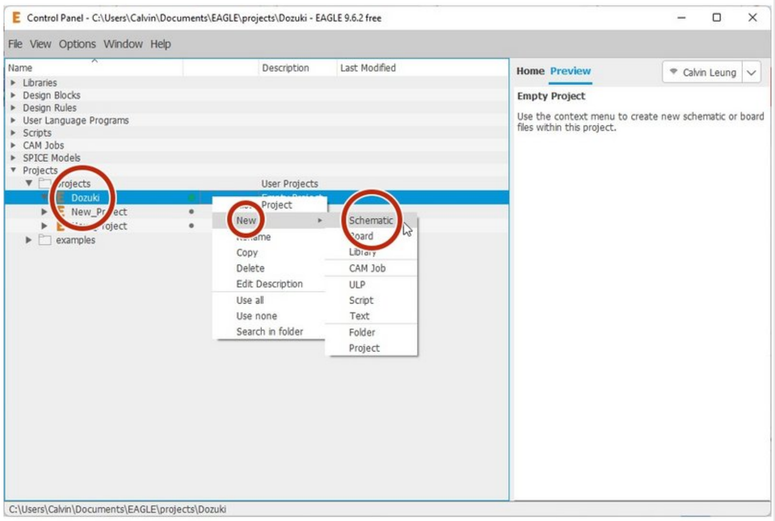
Step 3 — Create a new project

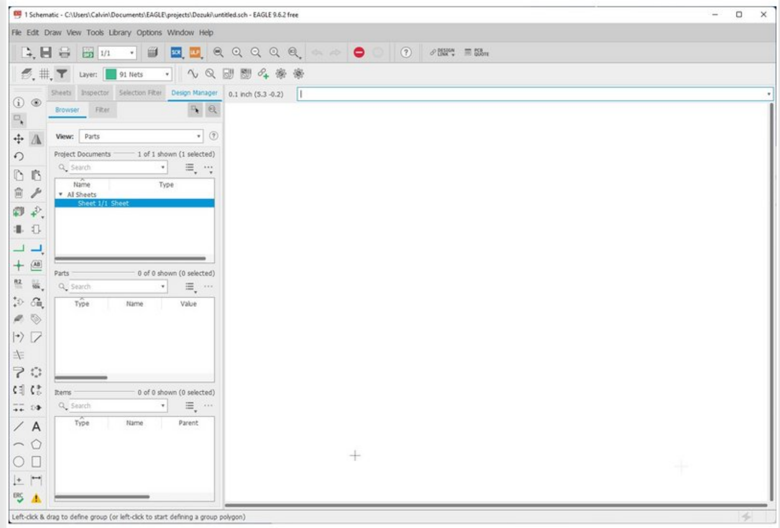




- From the control panel: File > New > Project
- Enter a name for the Project

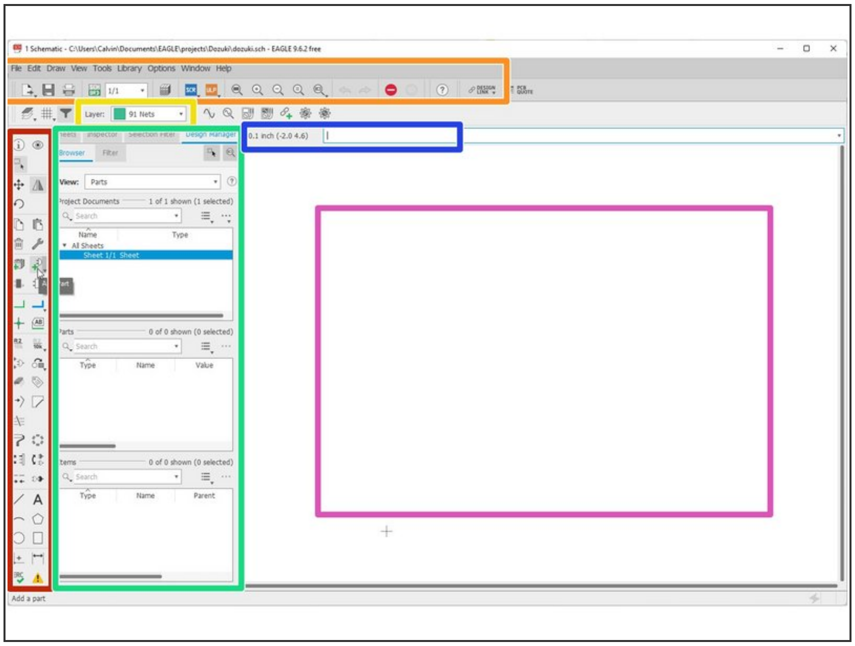
Step 4 — Create a new schematic





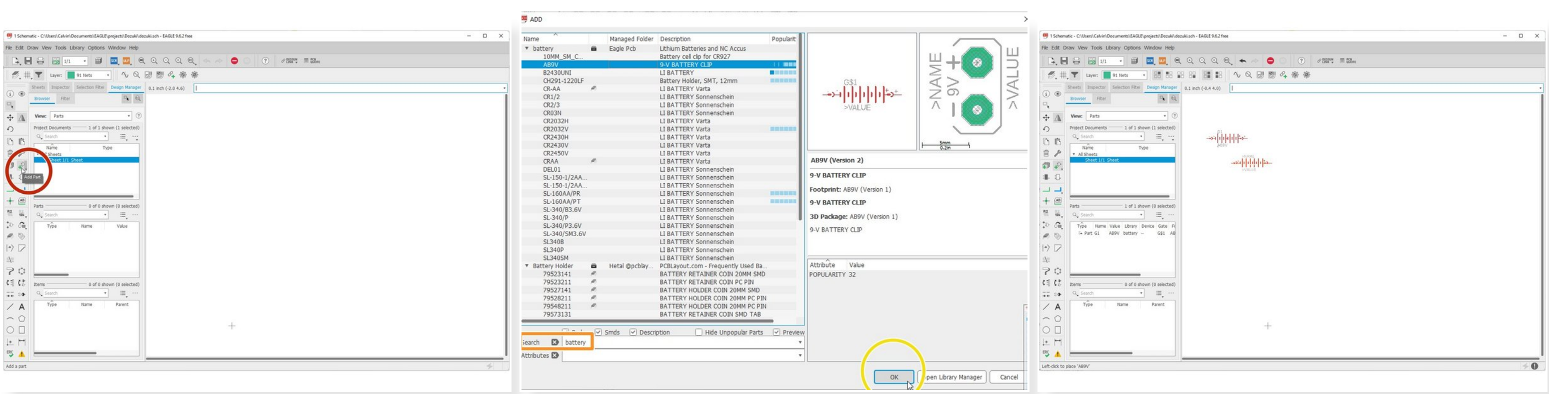
- Right-click your project > New > Schematic
- The schematic editor will pop up.
- Ctrl + s to save and name your schematic

Step 5 — Overview of Schematic Editor



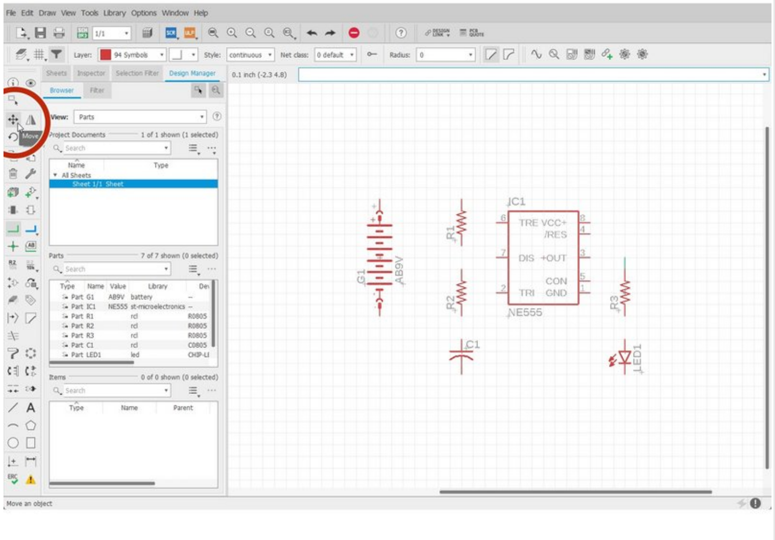
- Tool Bar
- General file control tools
- Layer
- Part Properties Editor
- Command Line
- Schematic Canvas

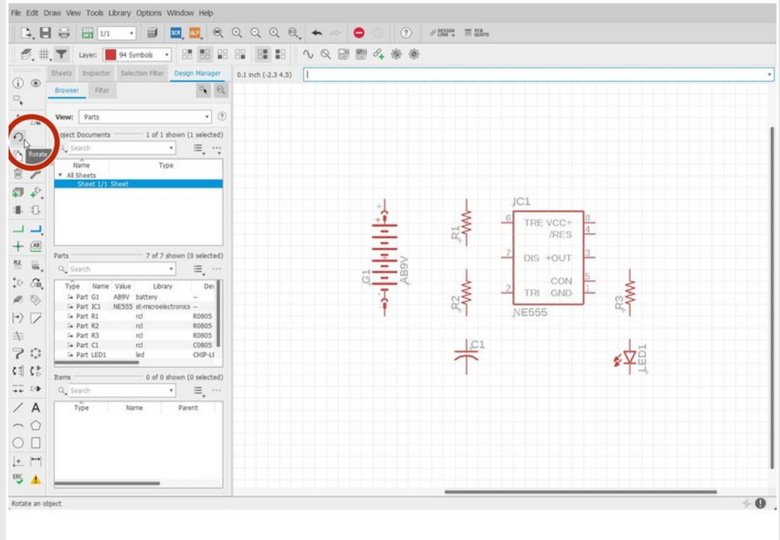
Step 6 — Schematic: Adding Parts



- Click "Add Part"
- Use the Search Bar to look for existing parts in free libraries
- Click "OK"
- Use left-click to place parts on the schematic
- Use ESC to stop placing parts

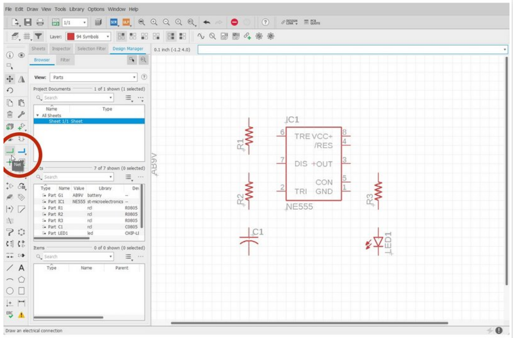
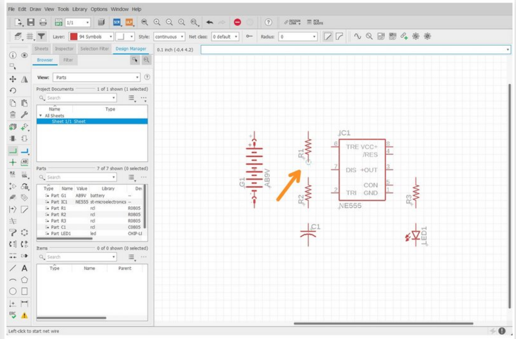
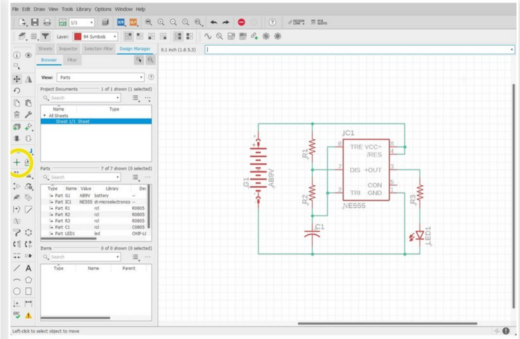
Step 7 — Schematic: Orienting Parts





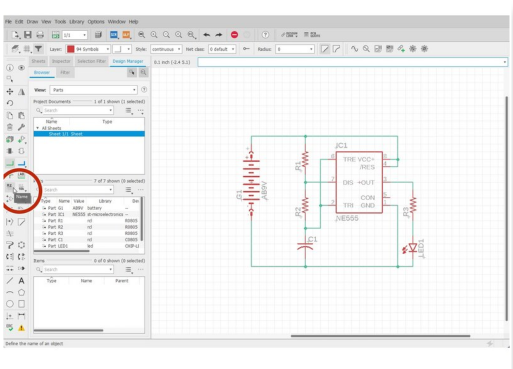
- You can oreint your parts with the:
  - Move Tool
  - Rotate Tool

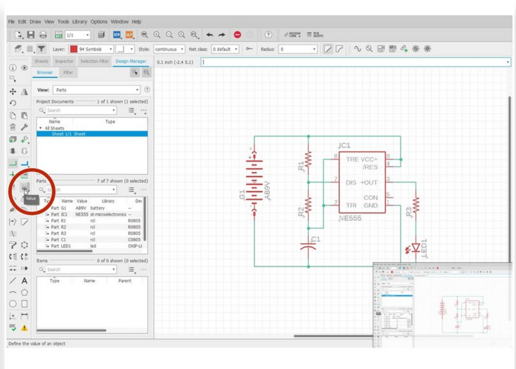
Step 8 — Schematic: Adding Nets (wires)

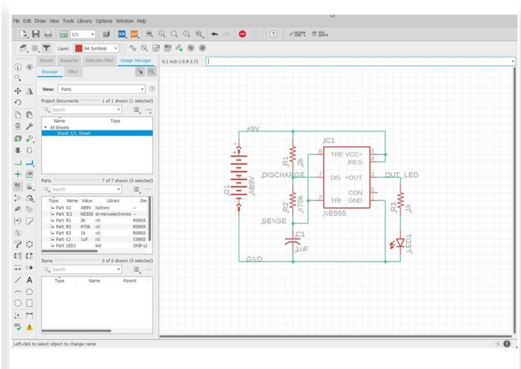




- Connect parts with nets.
  - Nets are like wires to connect components together.
- Select the Nets Tool
- Make sure you see a circle when initiating a net from one pin to another.
- Make use of the junction tool when connecting multiple nets.

Step 9 — Schematic: Label Parts and Nets

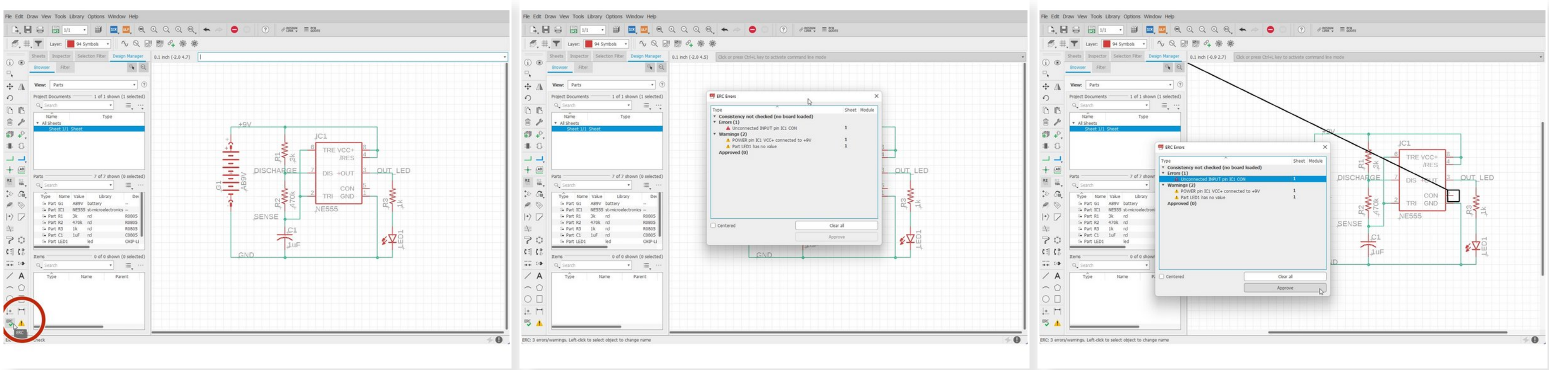






- It is important to label parts and nets so that you have the necessary information to make decisions when routing parts.
- Label Tools such as:
  - Name
  - Value

Step 10 — Schematic: Electrical Rule Check



- Perform ERC to make sure your schematic is bug-free.
- Click "ERC". A list of errors and warnings would pop up. Click individual errors to see where the issue occurs.

⚠ Errors (1) - These warrant you taking a careful look at. Anything here could very well cause your circuit to fail.

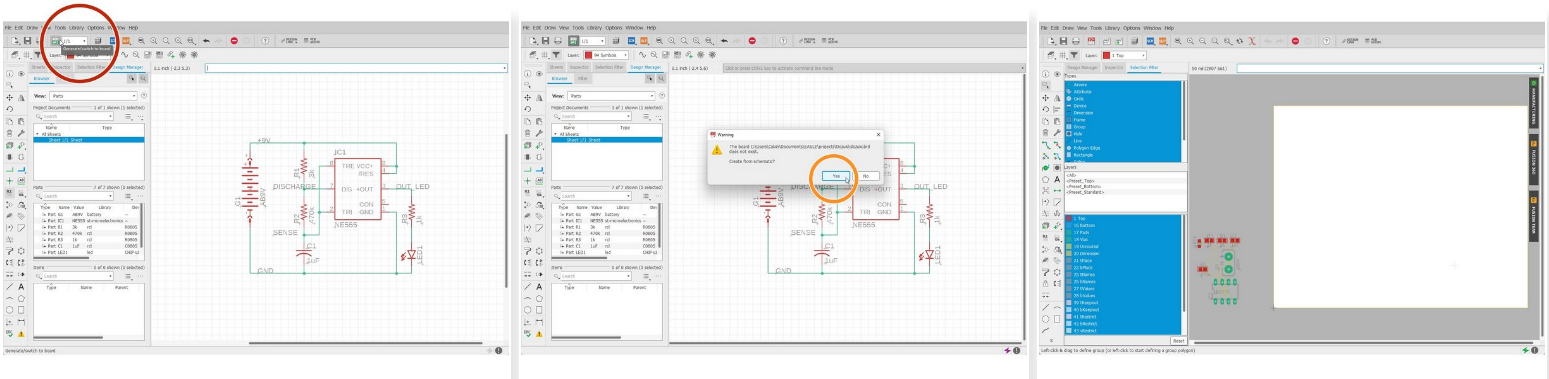
- Unconnected INPUT pin IC1 CON - In general, unconnected input pins are bad. In this case, the CON pin is a reference voltage that you can manually set, but nothing bad happens if you leave it unconnected (floating).

⚠ Warnings (2) - These are not as urgent as errors, but still require a cursory glance. One warning to look for is the one about a net only having one node/pin. That means that you didn't connect that net on both ends.

- POWER pin IC1 VCC+ connected to +9V- Eagle warns you whenever you connect different voltages of power together (if you connect a 12V power supply line and a 5V power supply line, bad things happen). In this case, it's just a nomenclature difference, so it's OK to approve.
- Part LED1 has no value - If I wasn't as lazy as I am, I'd have given the LED1 part a value, but until then, this warning will exist.

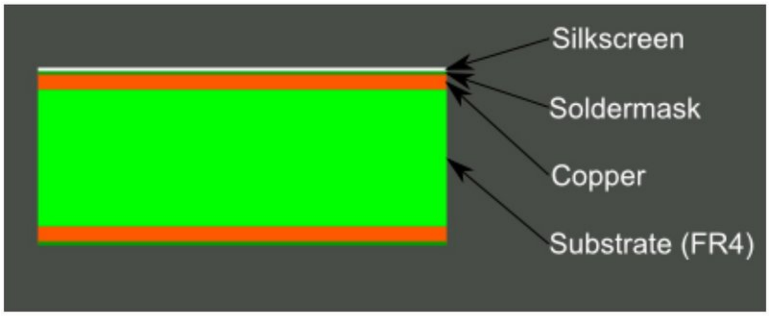
- New line.Approved (0) - After you click the 'Approve' button on a warning/error, it goes in here.

Step 11 — Schematic to Layout



- After the schematic is completed, Eagle can use your schematic information to prepare the layout procedure.
- Click "Generate/Switch to board"
- Click "Yes"
- The layout editor will pop out.

Step 12 — Overview of Eagle's Layer Hierarchy

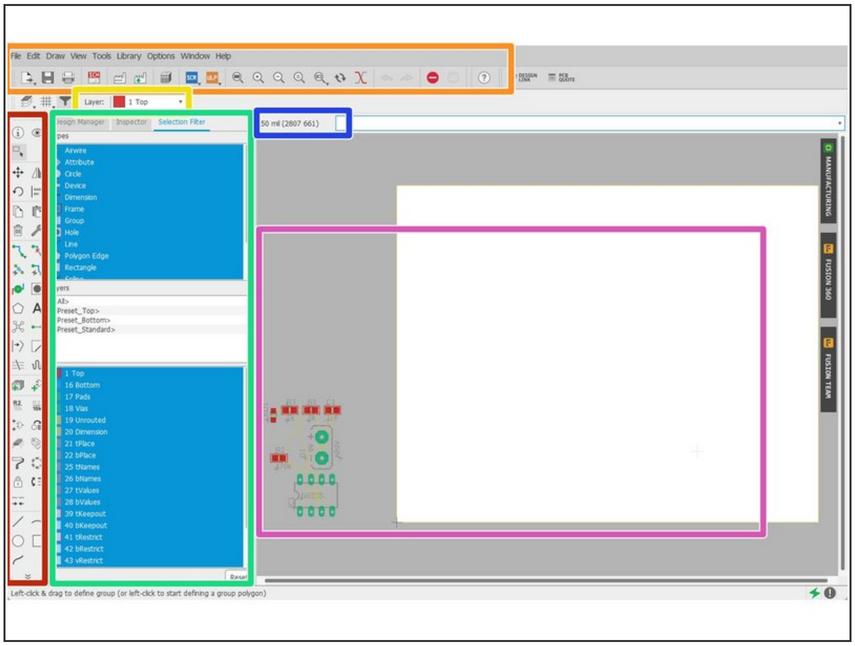


The layers of a double-sided PCB (image from the PCB Basics tutorial).

Color	Layer Name	Layer Number	Layer Purpose
	Top	1	Top layer of **copper**
	Bottom	16	Bottom layer of **copper**
	Pads	17	Through-hole pads. Any part of the green circle is exposed **copper** on *both* top and bottom sides of the board.
	Vias	18	Vias. Smaller copper-filled drill holes used to route a signal from top to bottom side. These are usually covered over by soldermask. Also indicates **copper** on both layers.
	Unrouted	19	Airwires. Rubber-band-like lines that show which pads need to be connected.
	Dimension	20	Outline of the board.
	tPlace	21	**Silkscreen** printed on the top side of the board.
	bPlace	22	**Silkscreen** printed on the bottom side of the board.
	tOrigins	23	Top origins, which you click to move and manipulate an individual part.
	bOrigins	24	Origins for parts on the bottom side of the board.
// Hatch	tStop	29	Top stopmask. These define where **soldermask** should *not* be applied.
\\ Hatch	bStop	30	Absent **soldermask** on the bottom side of the board.
	Holes	45	Non-conducting (not a via or pad) holes. These are usually drill holes for stand-offs or for special part requirements.
	tDocu	51	Top documentation layer. Just for reference. This might show the outline of a part, or other useful information.

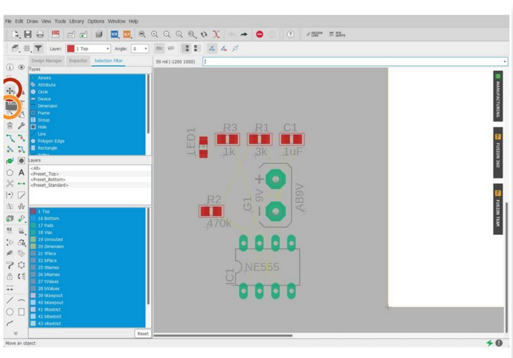
- Silkscreen: This layer refers to the markings on the board that usually provides information such as component name, outlines, etc.
- Soldermask: This is a layer of polymer that is put on a circuit board to protect the copper from oxidation and shorts during operation.
- Copper: This is the layer where all electrical routes and vias (through holes) exist.
- Substrate: This is a nonconductive layer that separates the top copper and bottom copper. This allows two layers of non on-overlapping routing.
- The PCB is structured so that the above layers are sandwiched together with the substrate in the center.

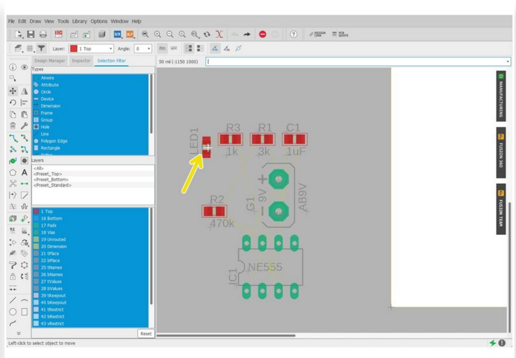
Step 13 — Overview of Layout Editor

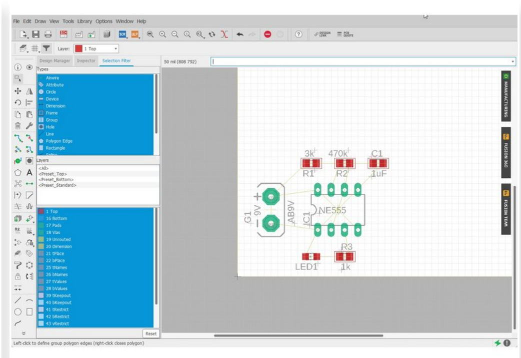


- Tool Bar
- General file control tools
- Layer
- Part Properties
- Command Line
- Layout Canvas

Step 14 — Layout: Arranging Footprints

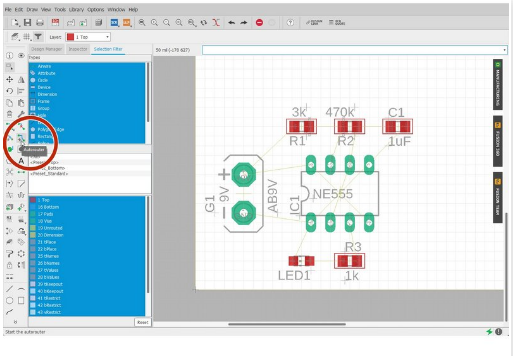


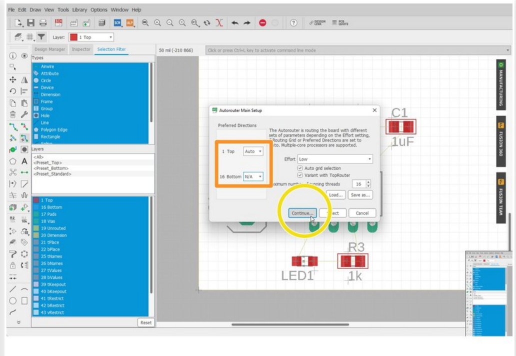


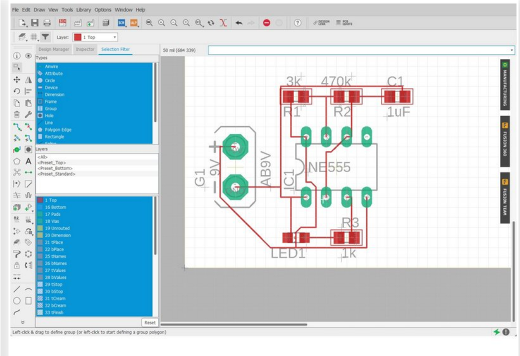


- You can arrange your footprints with the:
  - Move Tool
  - Rotate Tool
- Note that when selecting a footprint to edit, make sure you click on the cross that is in the center of the footprint.

Step 15 — Layout: Routing (Option 1: Automatic)

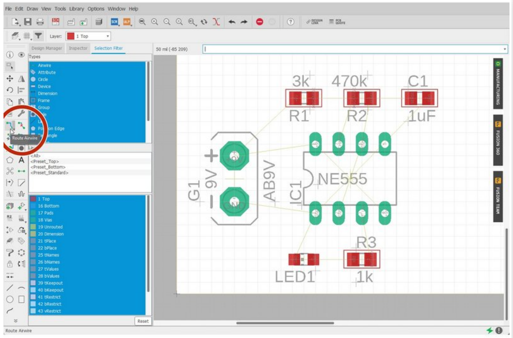


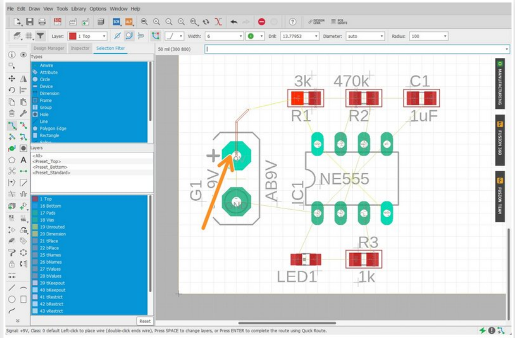


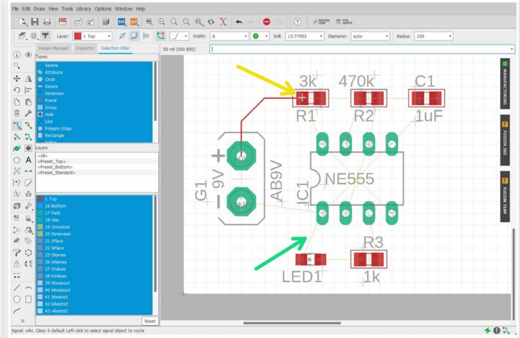


- Routing can be done manually or automatically.
  - You would want to route manually if there is a specific routing method you want to use.
- To perform automatic routing:
  - Click "Autorouter"
  - Select the board configuration for top and bottom.
    - In this example, I only want to route the top layer, so I select N/A for the bottom layer
  - Click "Continue" > "Start" > "End Job"

Step 16 — Layout: Routing (Option 2: Manual)

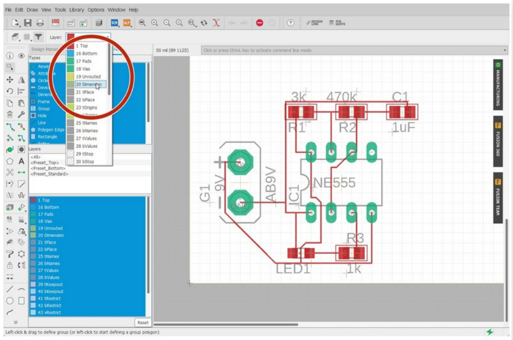


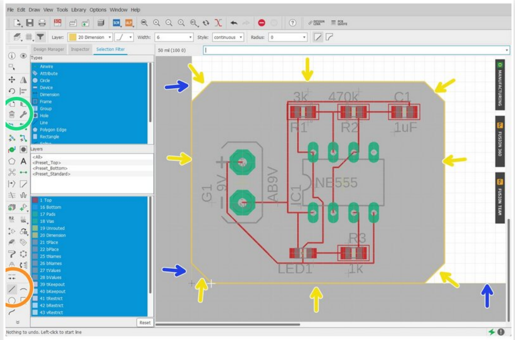


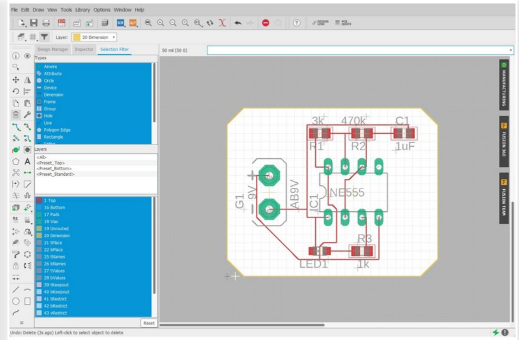


- With Manual routing, you have full control over how you want to route your traces.
- Select Route Airwire
- Click the start of an airwire
- Click the end of an airwire to complete a route
- Complete by routing all the remaining airwires

Step 17 — Layout: Outline / Border

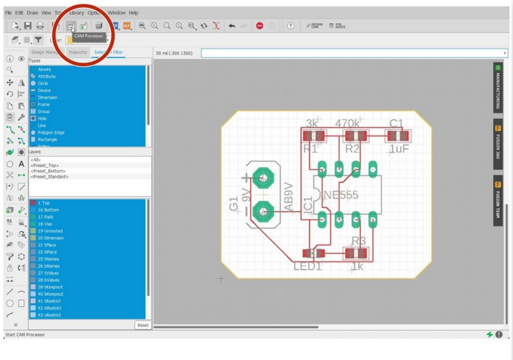
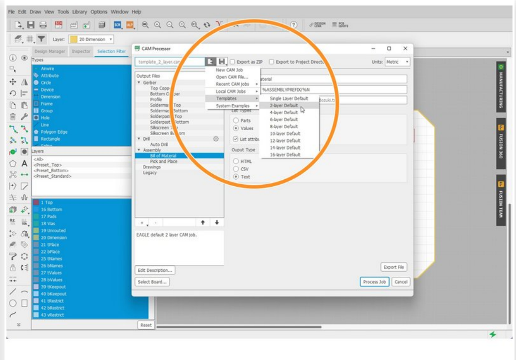
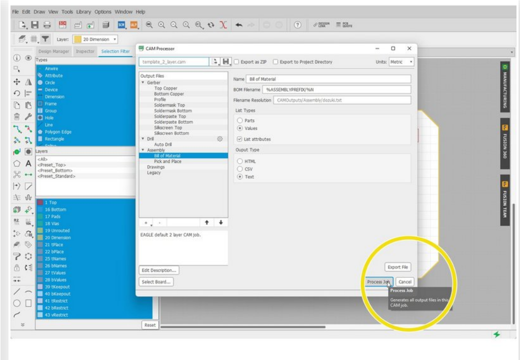






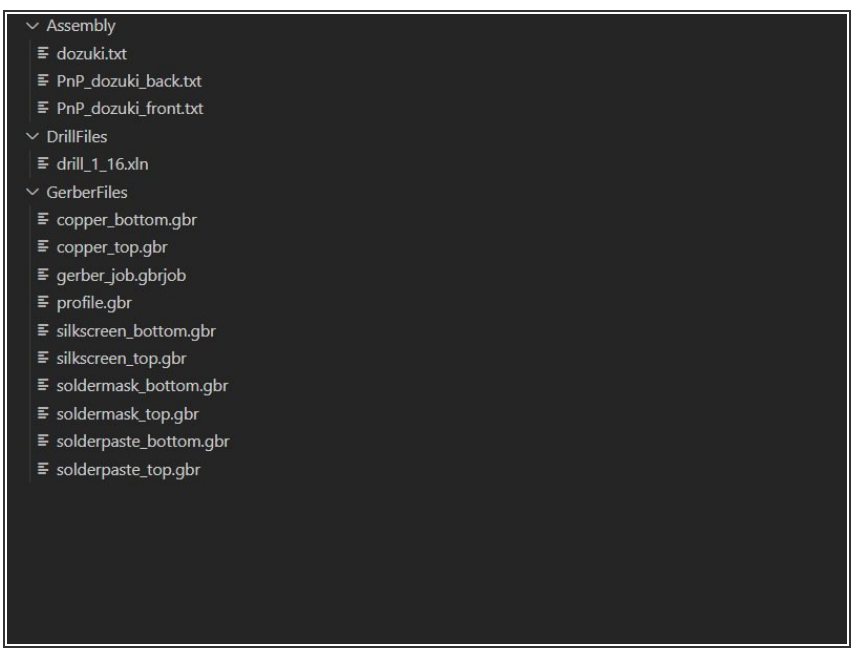
- Select layer 20
- Select Line Tool
- Trace the board outline with the line tool
- Select the Delete Tool
- Remove Unnecessary lines

Step 18 — Generate Gerber Files

- Select CAM Processor
- Select Load > Templates > 2 Layer Default
- Or choose other templates that works best for your design
- Click Process Job

Step 19 — Overview of Gerber Files



- After selecting a directory to save your files in, the directory tree looks like this.
- Congratulations! You've successfully completed a schematic design and PCB layout!

This document was generated on 2022-07-21 06:04:29 AM (MST).

References:

https://www.instructables.com/PCB-Creati...

https://learn.sparkfun.com/tutorials/usi...

