

# Altium Workshop

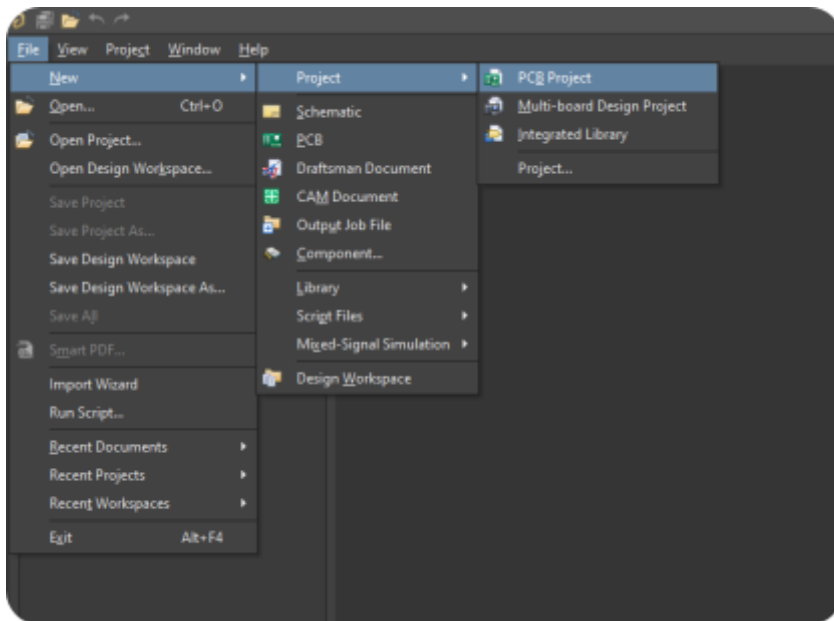
ELSOC 2024 - Joe Li

## Overview

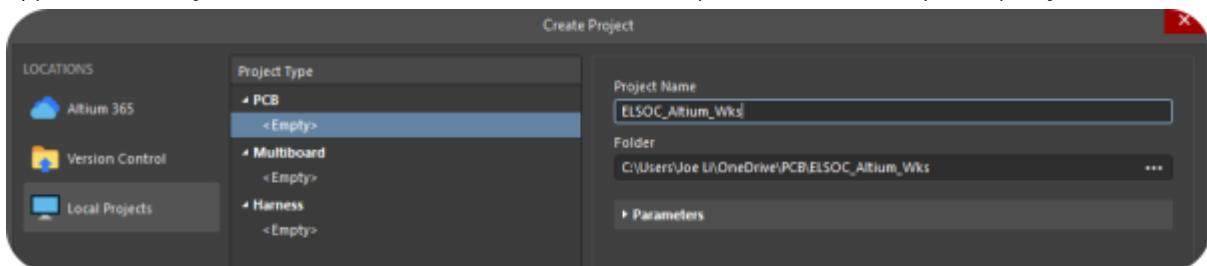
This is a workshop to get started on PCB design using Altium. We will be designing a PCB of a LED lighting sequence (the ELSOC PCB 2024) using the elec1111 lab kit. Please keep in mind this workshop is for beginners in PCB design and not for advanced features in Altium.

## Getting Started

Create a new project by going into File -> New -> Project...

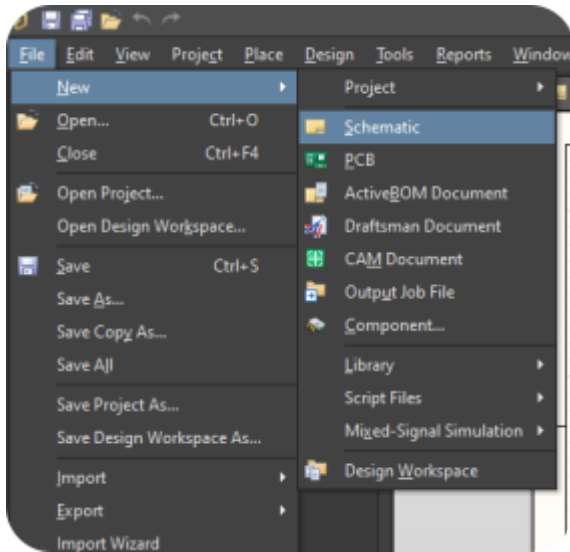


Type in a Project Name and choose a desired path to locate your project.

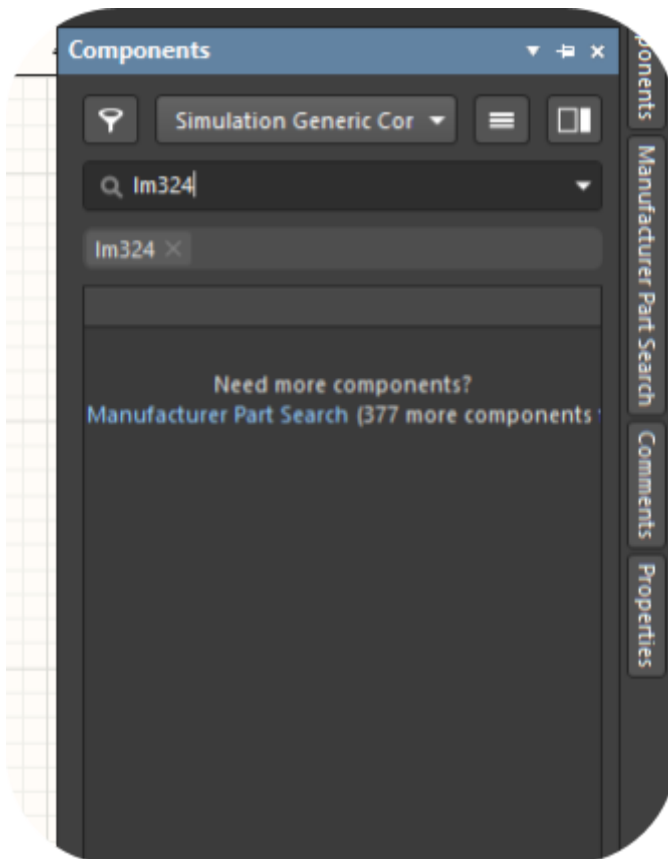


# Schematic

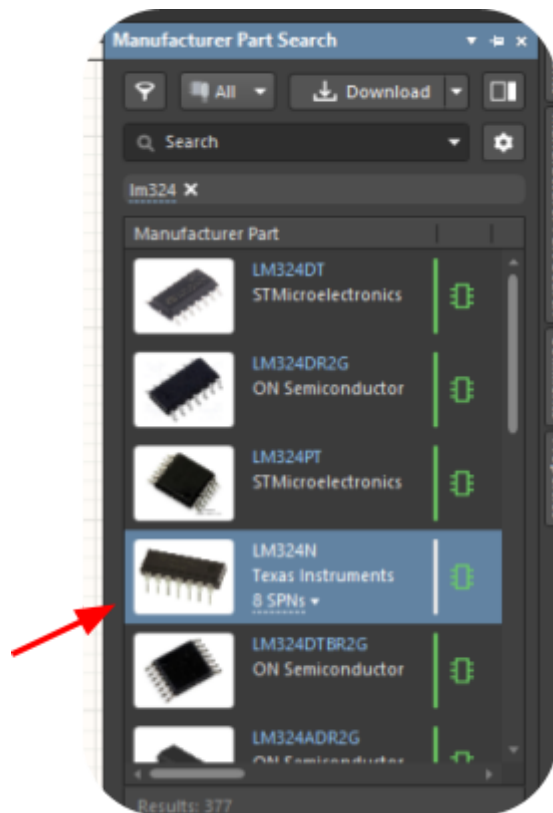
Create a new Schematic by going through File -> New -> Schematic



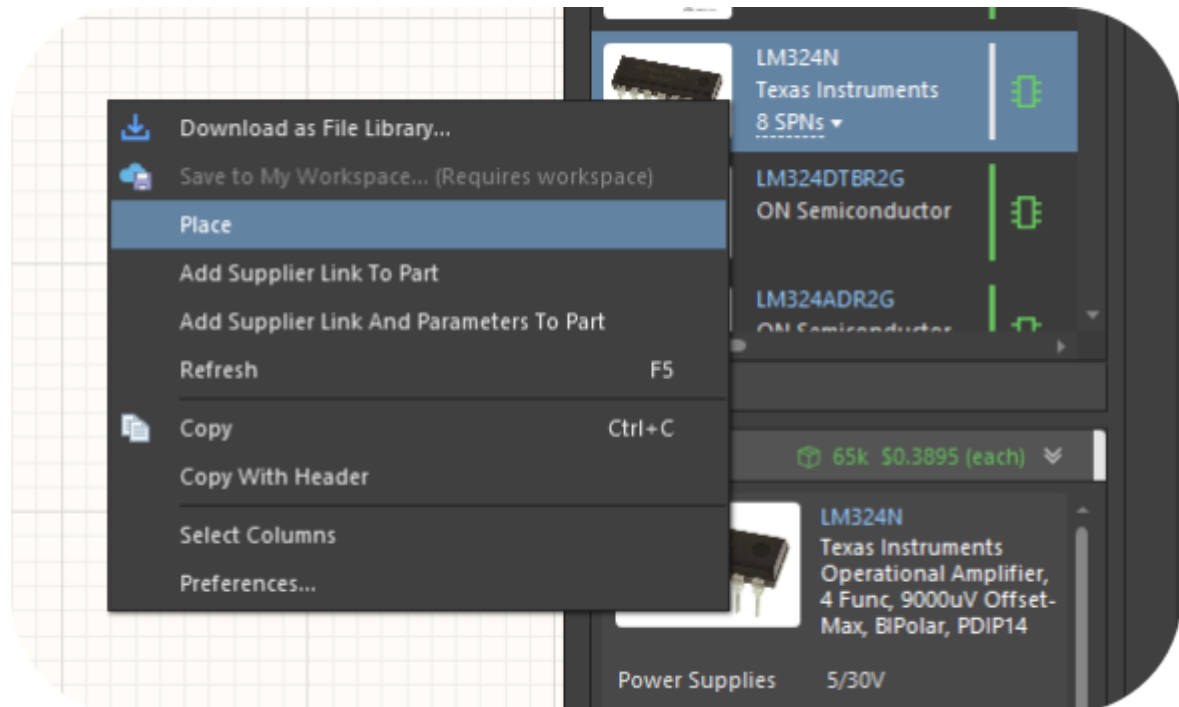
Go to the components panel on the right and search for the component you need, in this case, choose lm324 for the quad op amp chip package



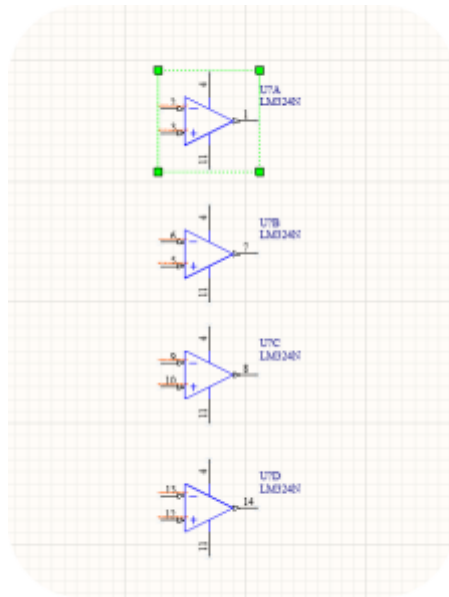
Find this through-hole package and download it, save it to your desired location.



After successfully downloading this package, right click and click “place”.



Place the four op amps into the schematics

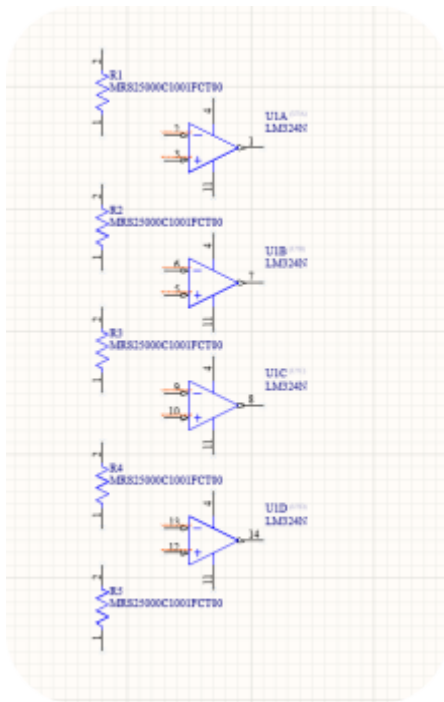


Upon placing the op amps down, click on the designators and name them "U1"

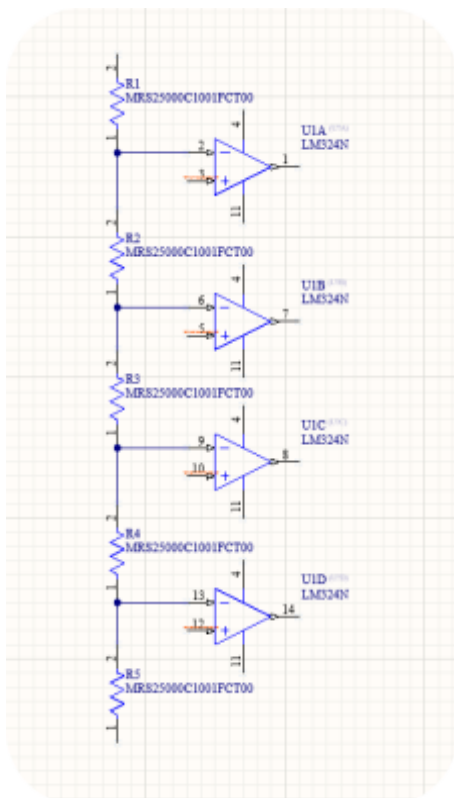
Continue to go into manufacturer part search and search for resistors. However upon searching for a resistor, it will give you a list of smd components. One might wish to solder through-hole components. To get a through-hole resistor, click on the filter function and choose "AxialMeter" under package type.



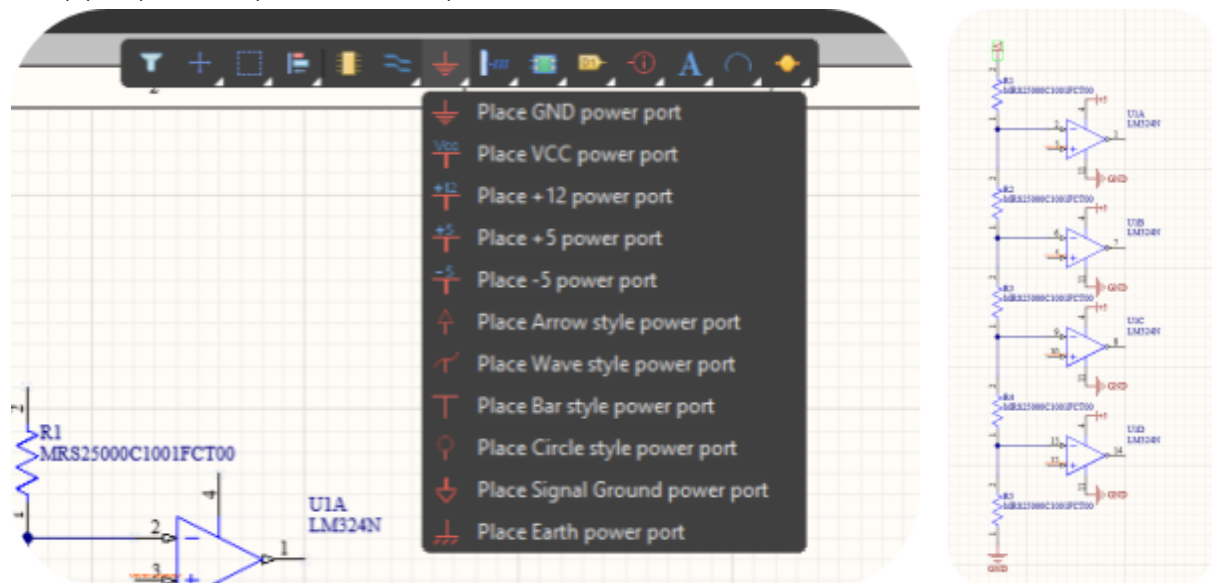
Choose a resistor that looks right (I recommend MRS25000C1001FCT00).  
 Download it and locate it in the schematic where desired.  
 Upon placing the first resistor, click on the resistor and name the Designator "R1" to "R5".. Then proceed to copy and paste this resistor.



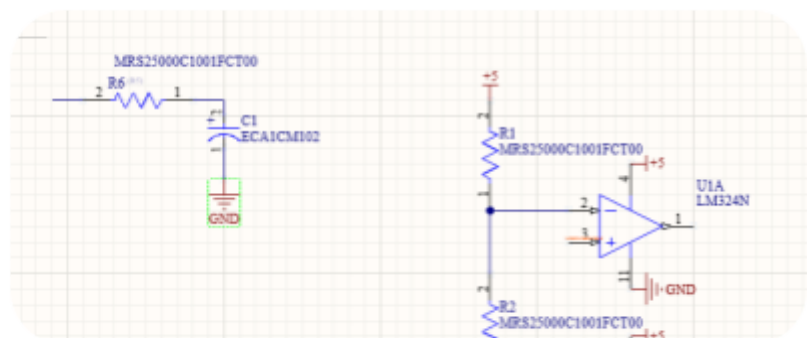
Press Control W to wire them appropriately.



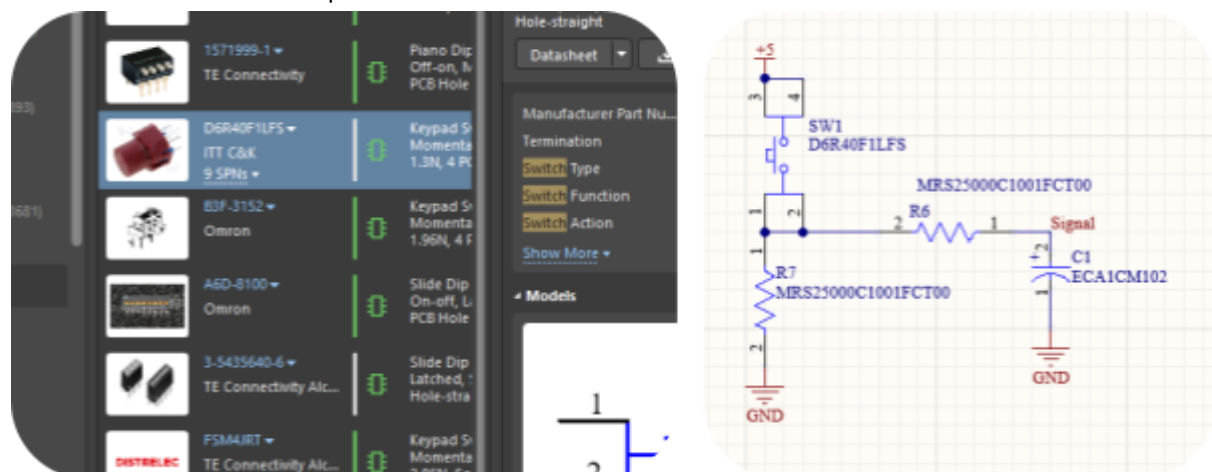
Put Power Net and GND Net by right clicking from the toolbar and place them in appropriate spots. Press Space to rotate



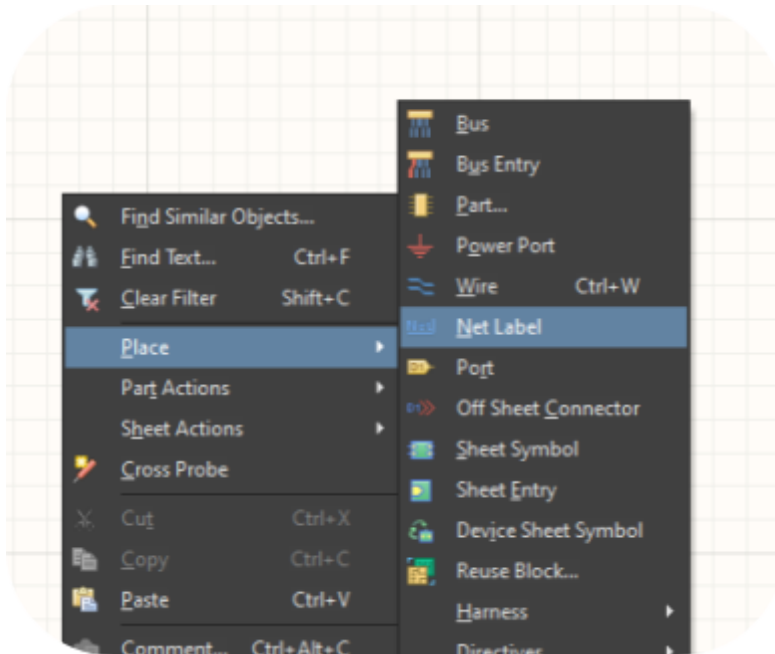
Find a 1000uF capacitor by using the filter function, download it and locate it in your schematic. Create a RC circuit with a capacitor and a resistor.



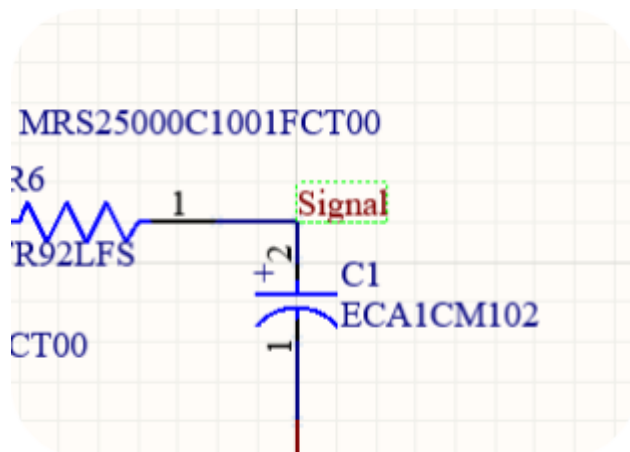
Place a switch with a pull down resistor in the RC circuit.



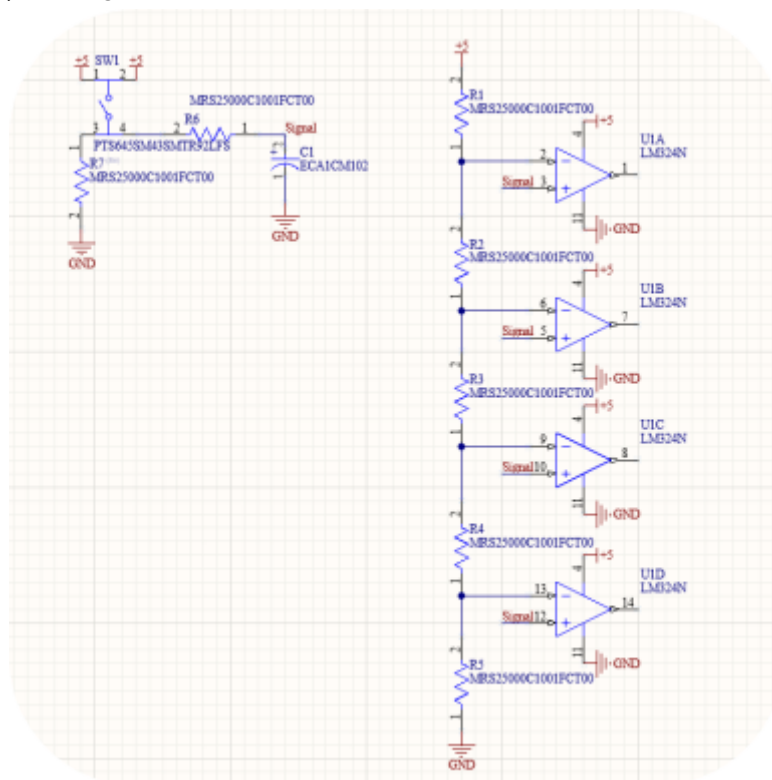
We are meant to connect this output of the RC circuit into the positive terminal of all the op-amp. However, that would create a mess. Instead what we would be doing is create a net label. First right click on an empty space in schematic, choose Place and select "Net Label"



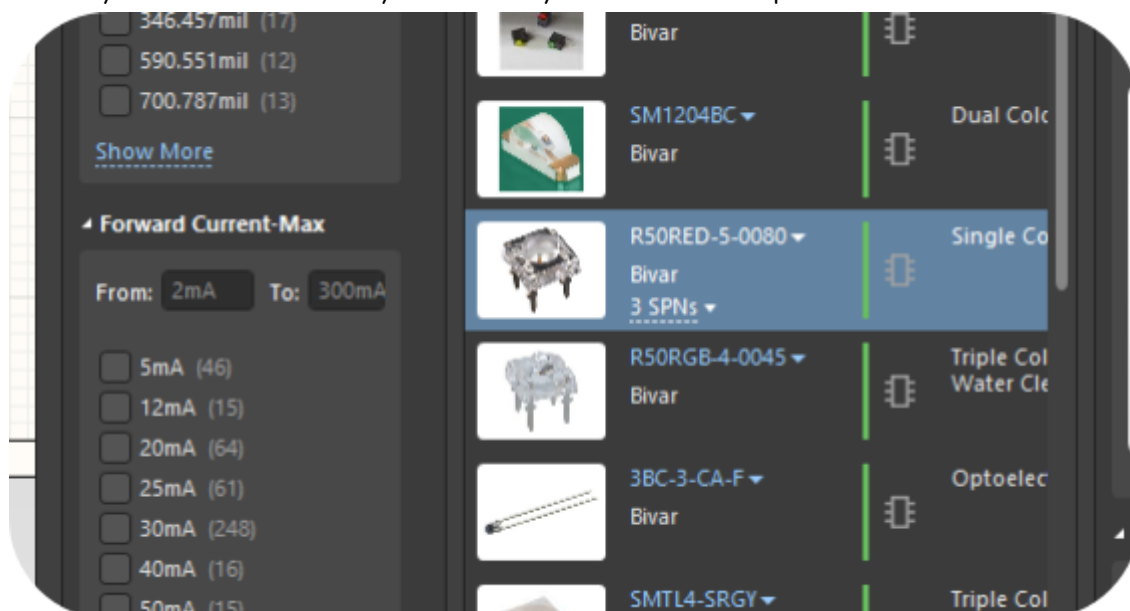
Place it on the output of the RC circuit and rename it "Signal" by clicking onto the label.



Copy the label and paste it on the positive terminal of all the op-amps in the package.



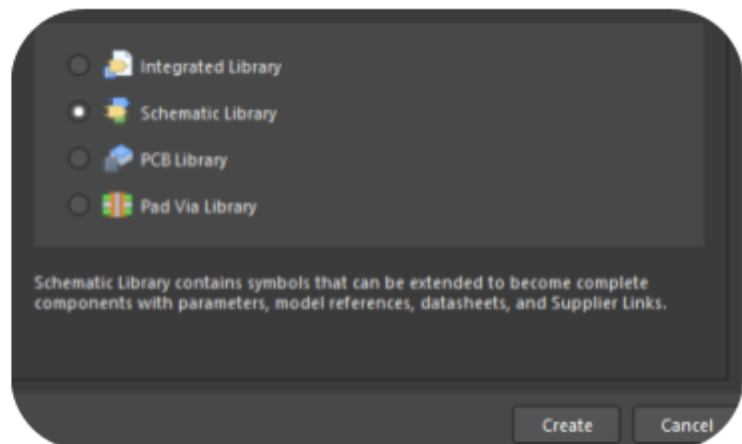
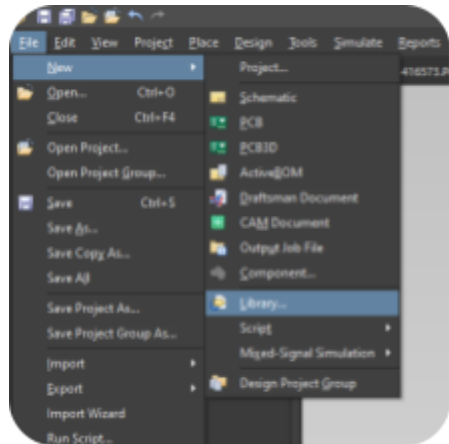
You might want to look for the LEDs you have in the ELEC1111 lab, but unfortunately, the LEDs do not have a model. We will use this opportunity to teach you how to create your own symbol and footprint.



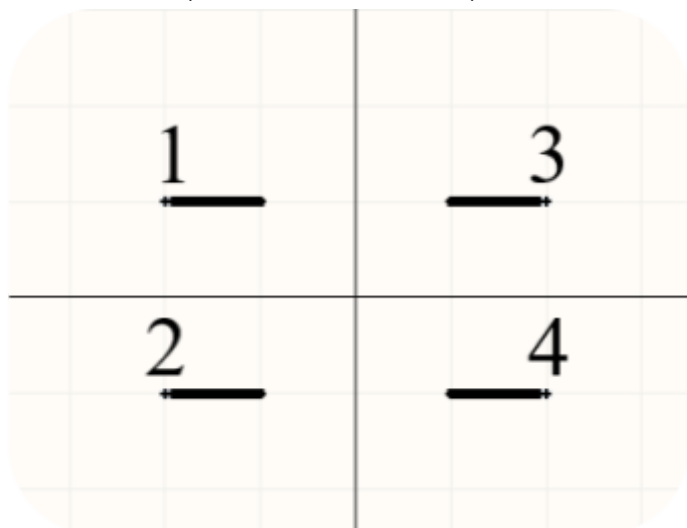
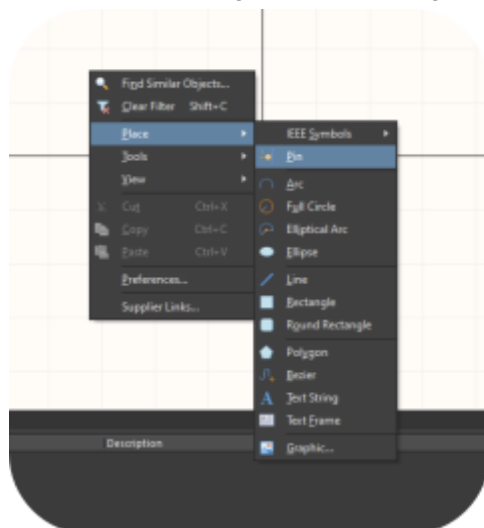


# Creating Custom Symbol & Footprint

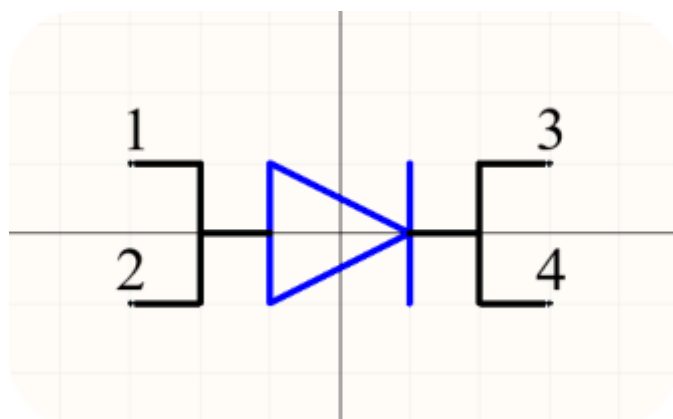
Move into File -> New -> Library, and select Schematic Library



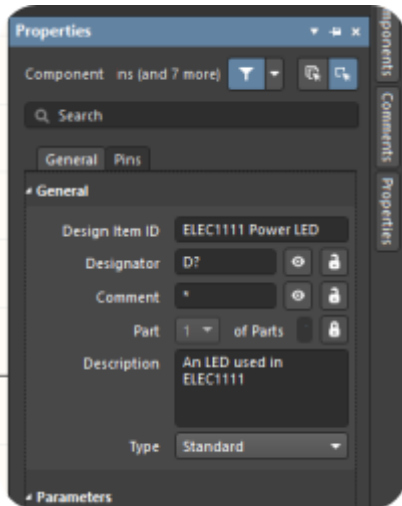
Right click into an empty spot, select Place -> Pin, place down four pins and have their designators being 1, 2, 3 and 4. Keep the names of all pins blank.



Use your artistic skills to draw a lovely diode that looks like below using the line function in the toolbar.

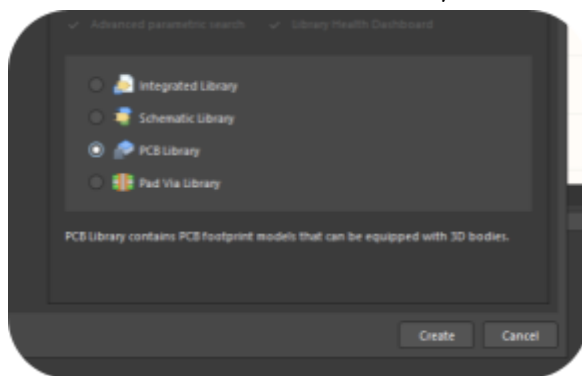


Fill in the properties tab on the right as follows.



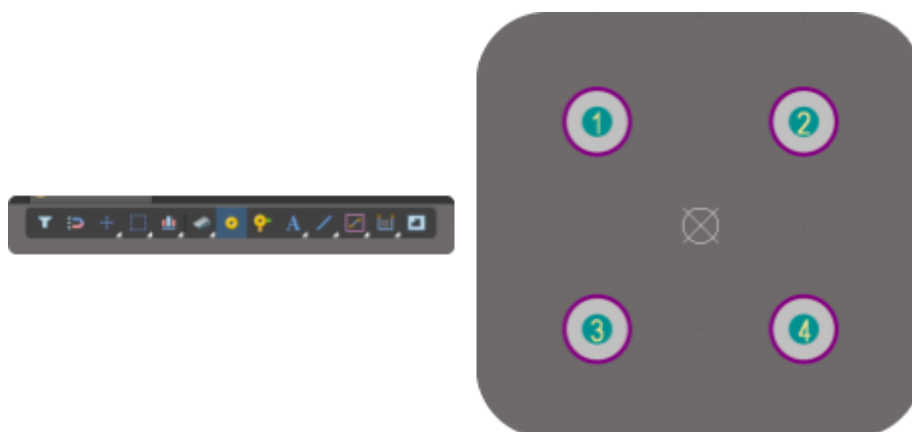
Save this schematic library somewhere you desire.

Now to create a footprint for the schematic (so your schematic links to the PCB itself. Go File -> New -> Library and select PCB Library.

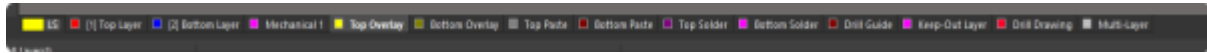


Upon entering the footprint editor, press G and select 100 Mil as the snap grid size. This is because a breadboard pin is 100 Mil apart and helps us easily put every pin hole in a safe distance on a PCB.

Choose the pin hole. Place them as follows.



Choose "Top Overlay" in the bottom panel to choose the silk screen layer

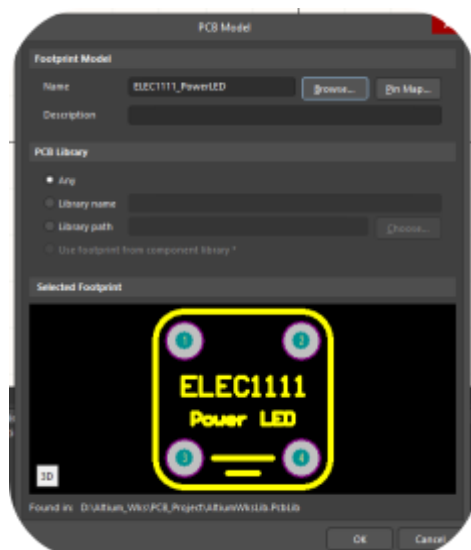


Decorate the footprint with your artistic skills, this is going to be what is displayed on the physical PCB with each LED you place. Once again, use the line and text function on the taskbar and your artistic skill to create a beautiful footprint. I recommend having an indication of GND at pin 3 and 4 such that soldering does not get confusing.

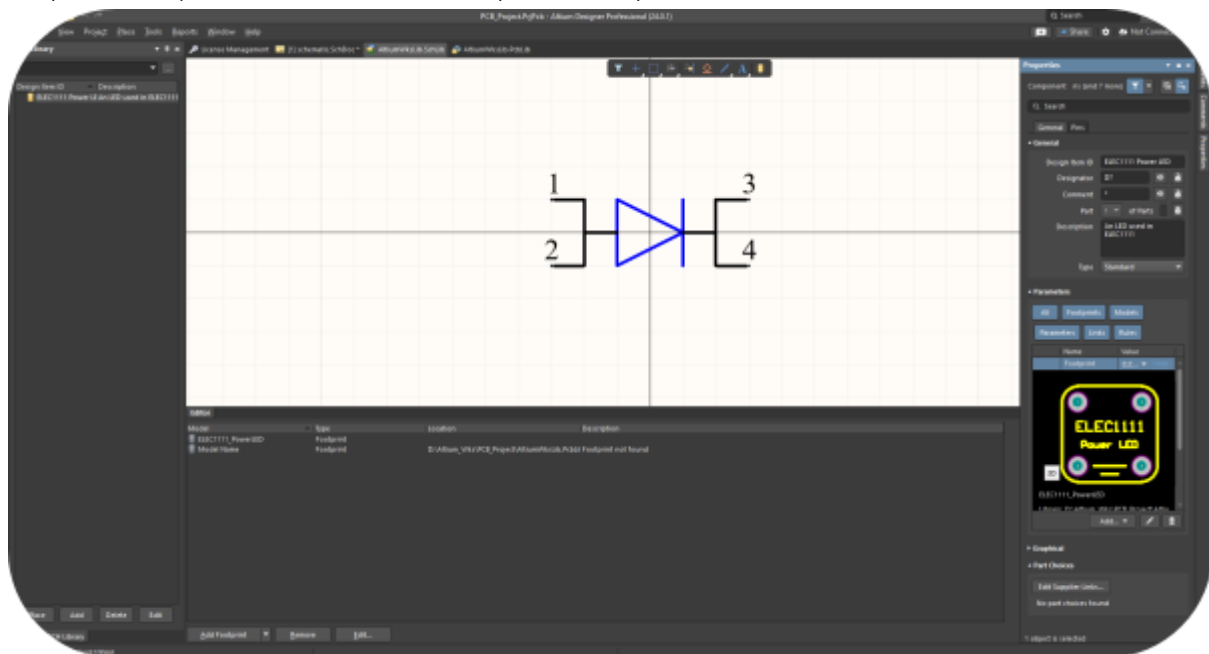


Press 3 to enter the 3D view and 2 to return to 2D. Rename this to "ELEC1111\_PowerLED" under properties. Save this file at a desired spot.

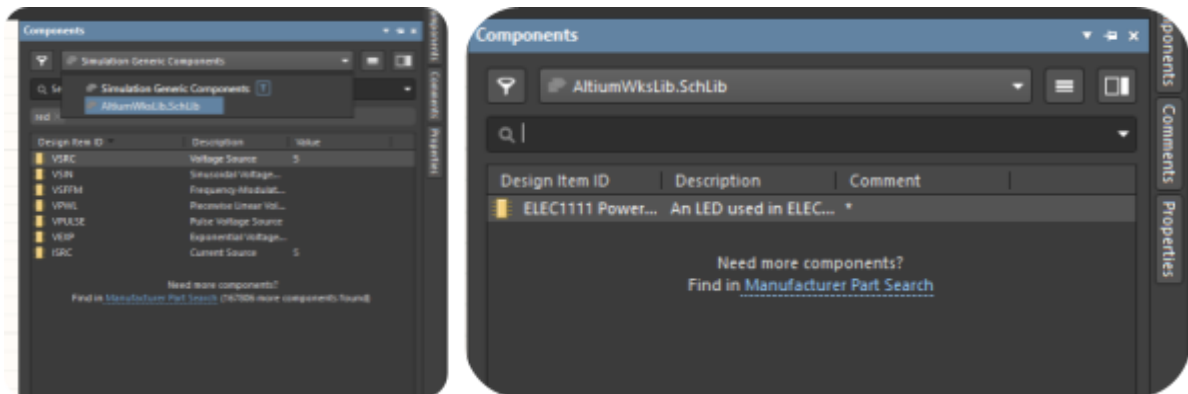
Return to the symbol library and click "Properties", at Parameters click Add...-> Footprint. Click "Browse" and select the footprint that you have created. Press Ok to create.



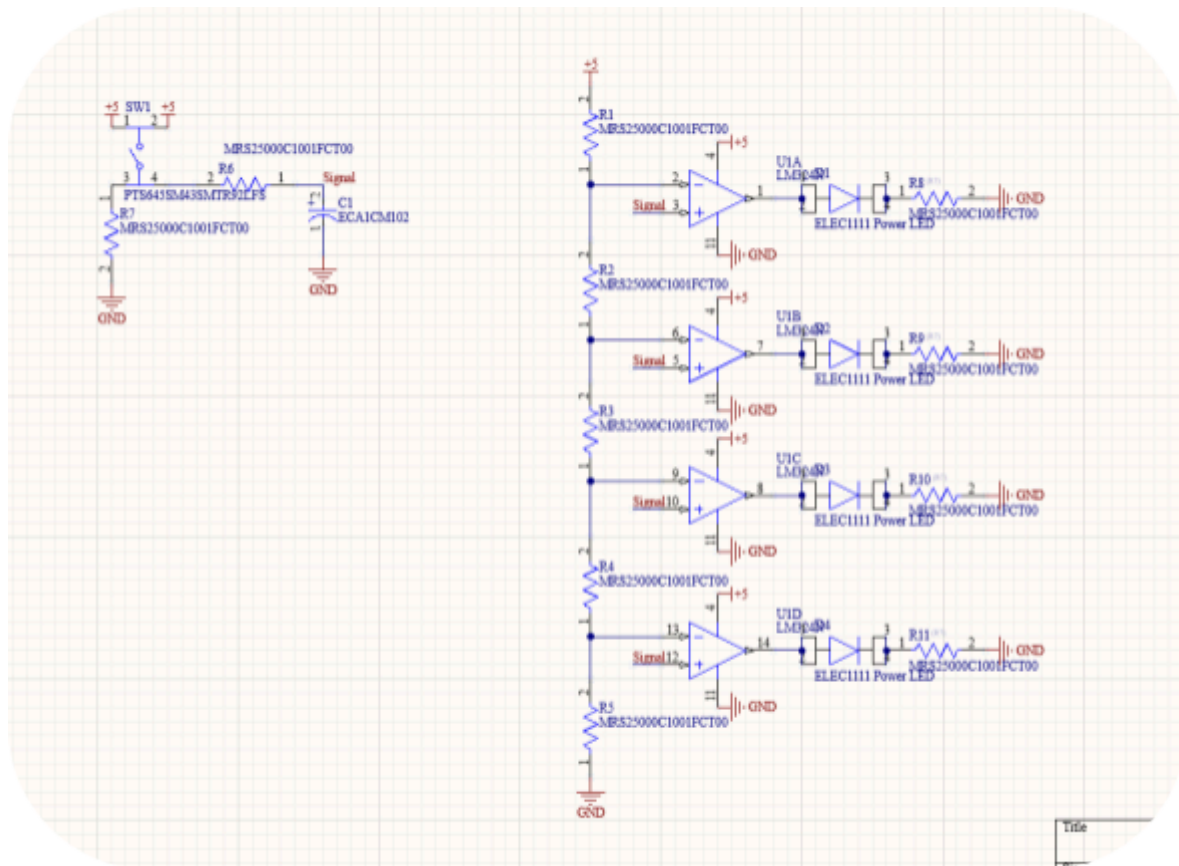
You have successfully linked your footprint to your symbol, save the schematic so you can proceed to use this symbol in your schematic.



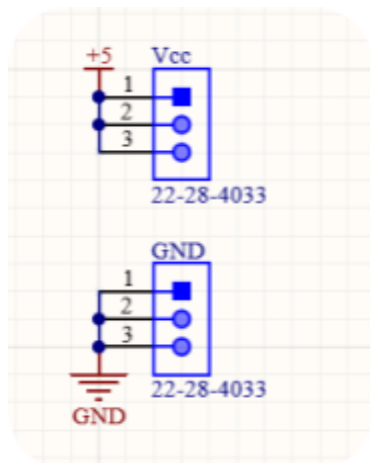
In the schematic, click on “Components” and select the symbol library that you have created under the drop bar



Place the component as shown below wire them up along with current limiting resistors.



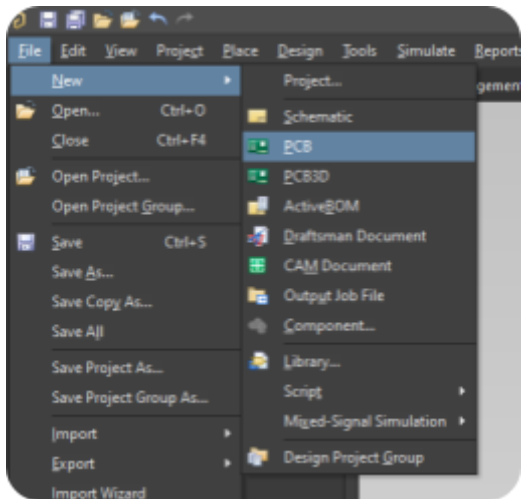
Place two connector pins for power and GND



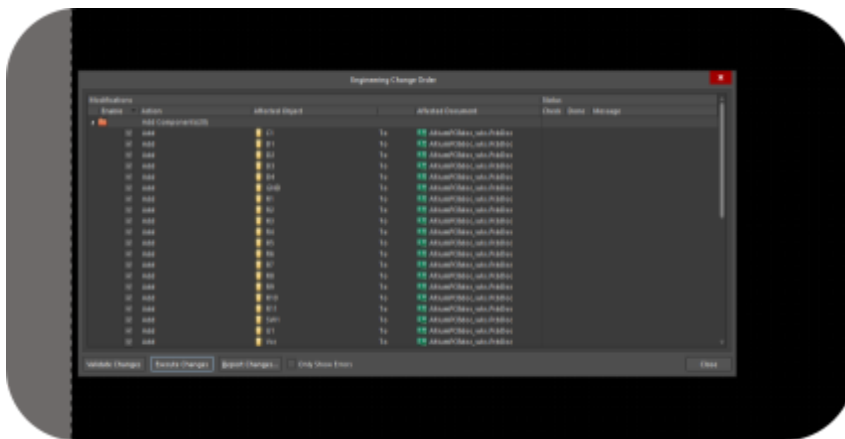
Save this before you lose your progress and cry.

PCB design

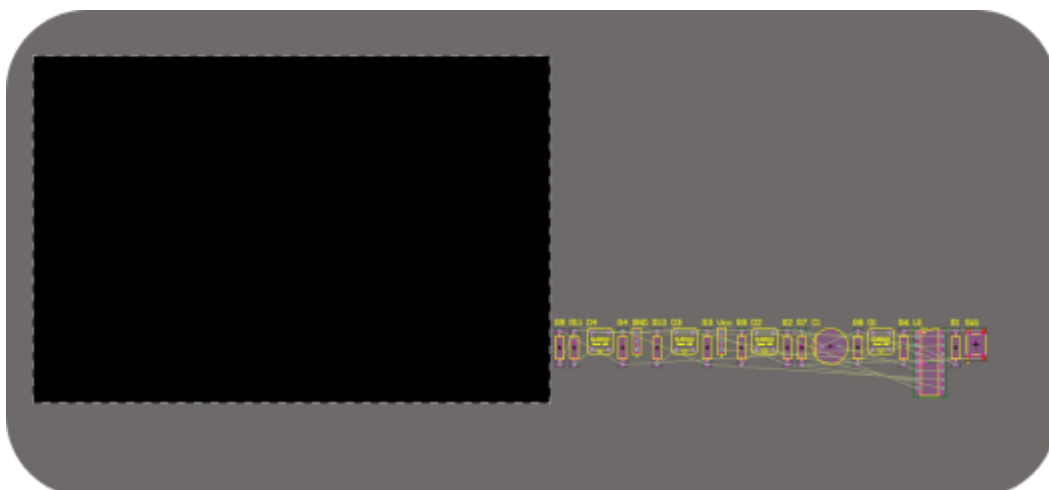
Create a new PCB by going through File -> New -> PCB



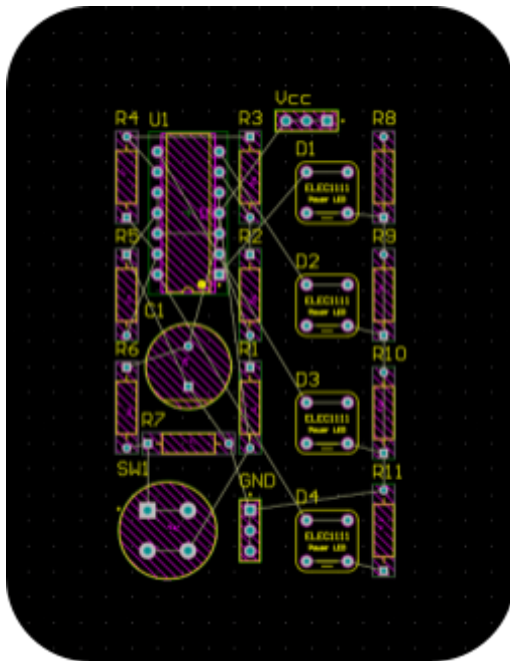
You should see a dark panel. Go Design -> Import Changes From ... and Execute Changes.



You should find your components placed down somewhere in your PCB document.

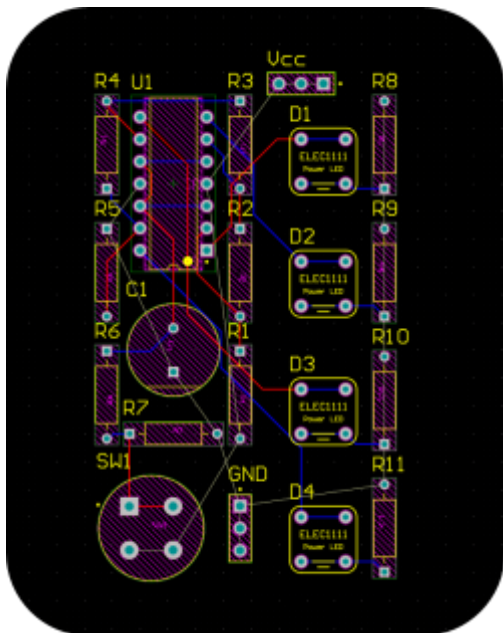


Arrange the component depending on how you want it to look like.  
Recommend a location where track length is minimal, i.e. lines between components are not long.

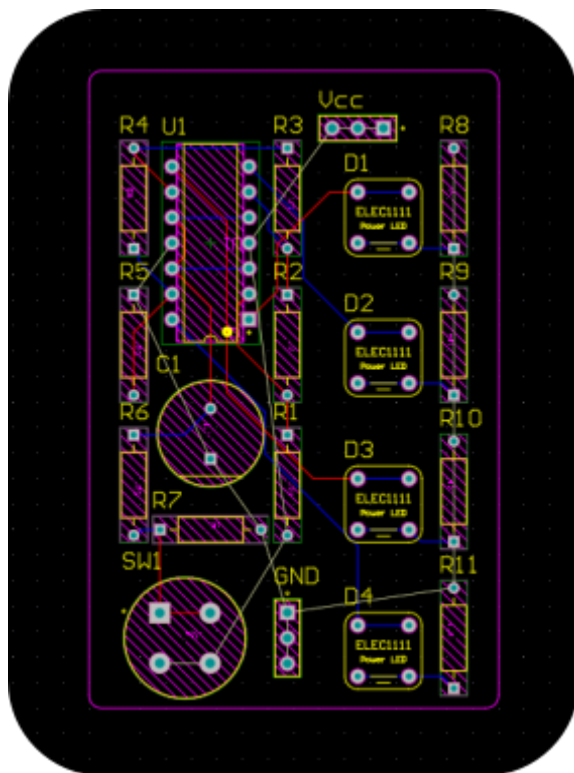


Now the fun part, Ctrl-W to start routing,

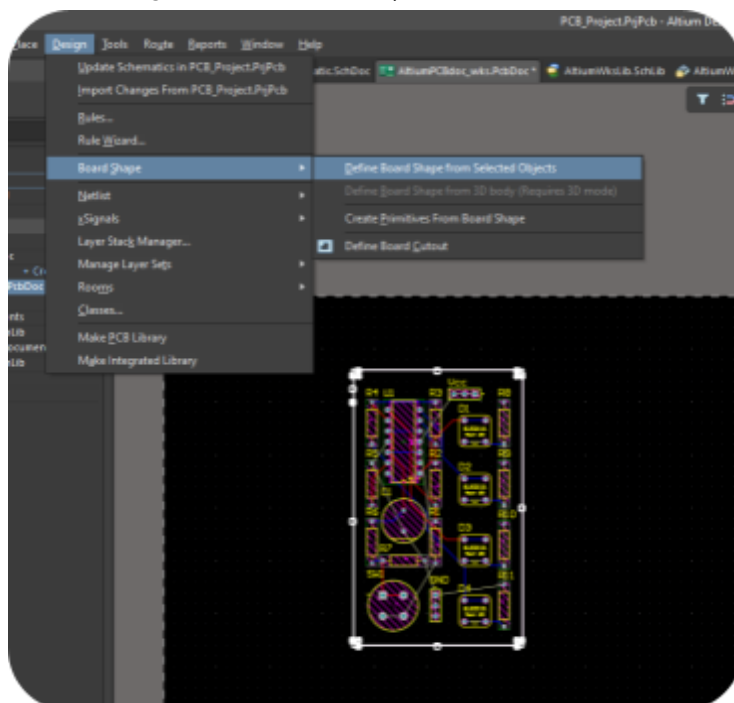
- route every pin except for GND and 5V.
- Route by clicking a pin and following these lines into the pins it instructs you to connect. These lines are based on what you have designed in the schematic.
- Ctrl Shift Scroll to swap between top layer and bottom layer.



Now Ctrl Shift scroll or click to Mechanical Layer and draw an outline for the board. Use the line function from the toolbar.



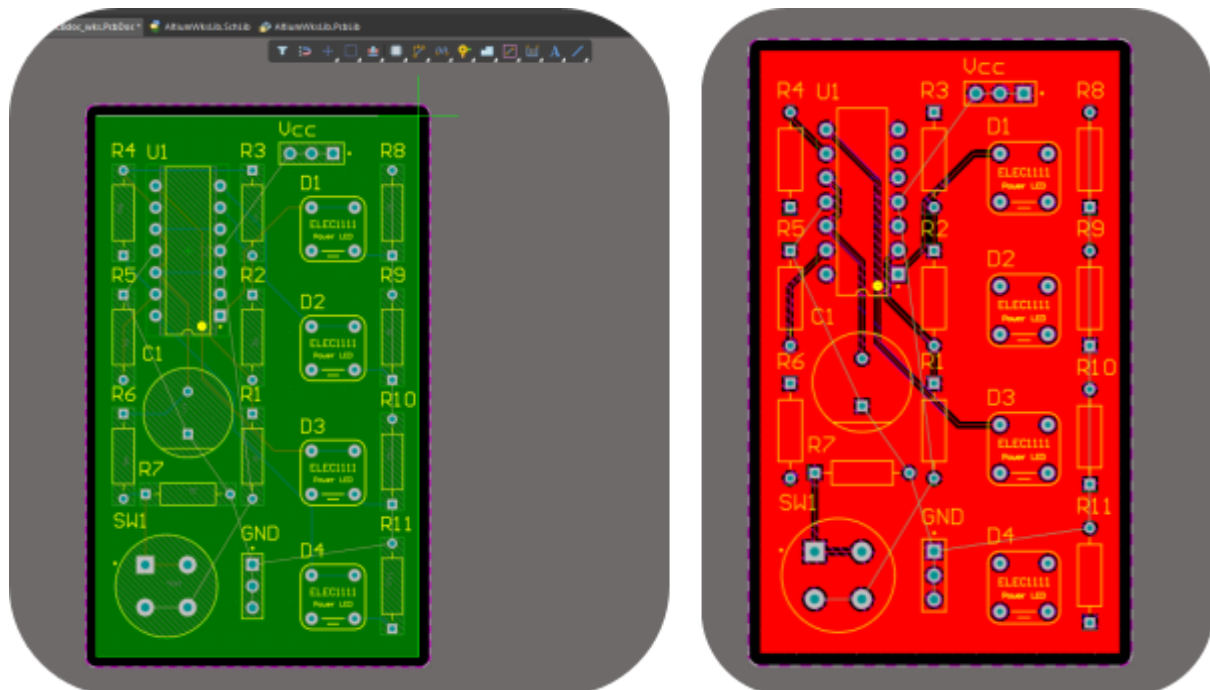
Click on one of the edge of the outline and press Tab to select all outline, then click Design -> Board Shape -> Define Board Shape to define board shape.



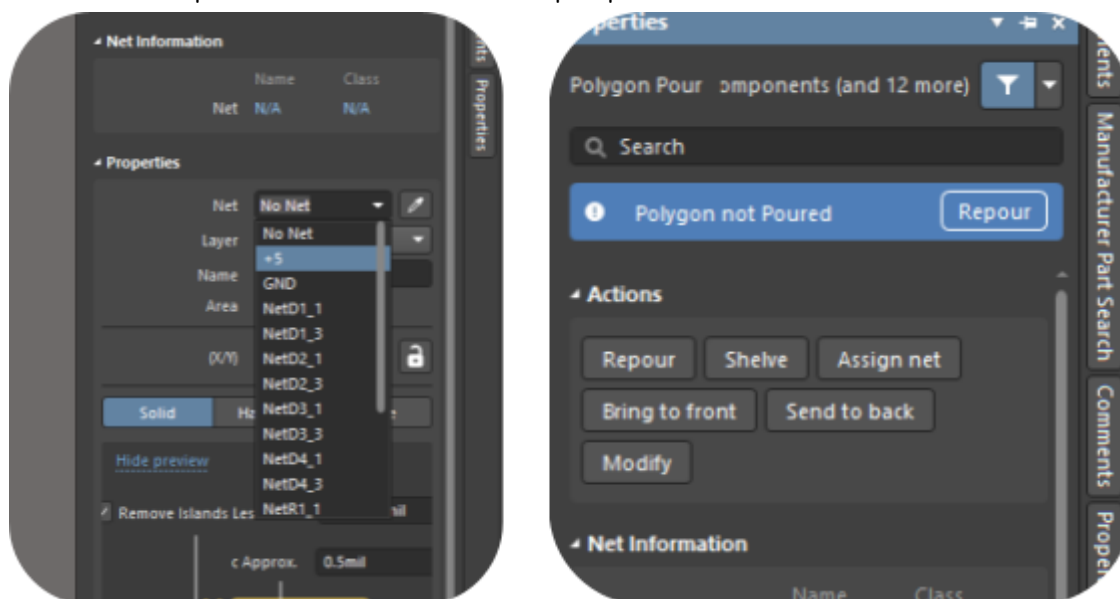
Press 3 to look at the 3D view and 2 to return.



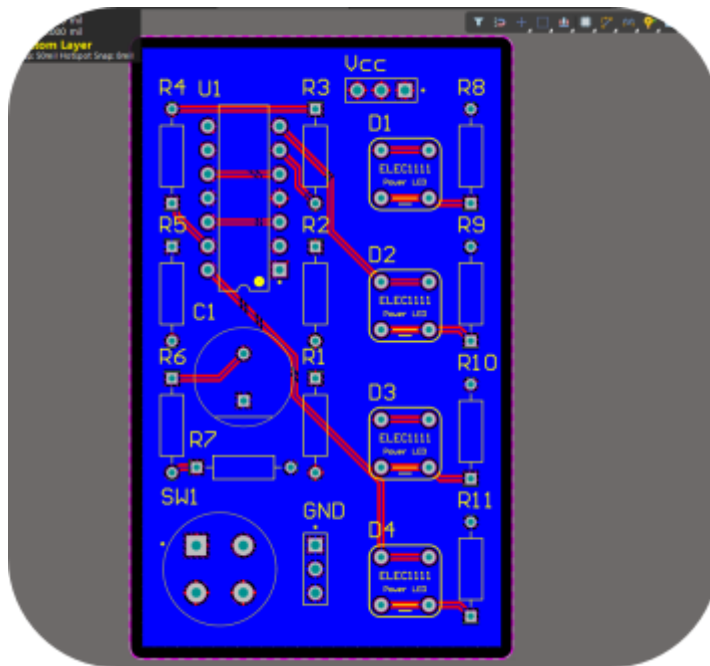
Now to finish the PCB off, start by choosing the Top layer, and to add a power plane, click "Place Polygon" on the toolbar and place a polygon over the PCB. Right click to finish the drawing.



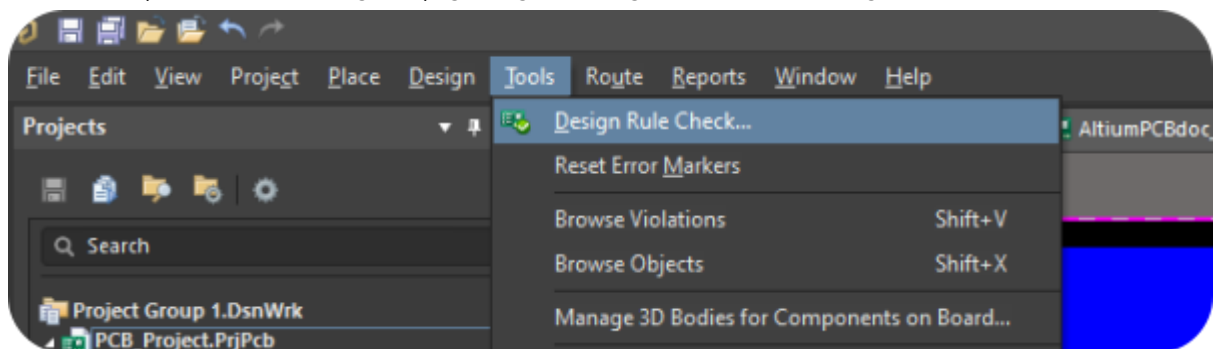
Click on the plane and select under properties "Net" to be +5. And click Repour.



Now repeat this for the bottom layer. Except, assign the net name to GND instead of +5, this should create a GND plane.



Your PCB design is complete. Run the Design Rule Check to check for potential errors in your PCB design by going through Tools -> Design Rule Check.



Click Run Design Rule Check as the window appears. This should generate a report about what you have missed out. There should be 0 Rule Violations if your PCB is good.

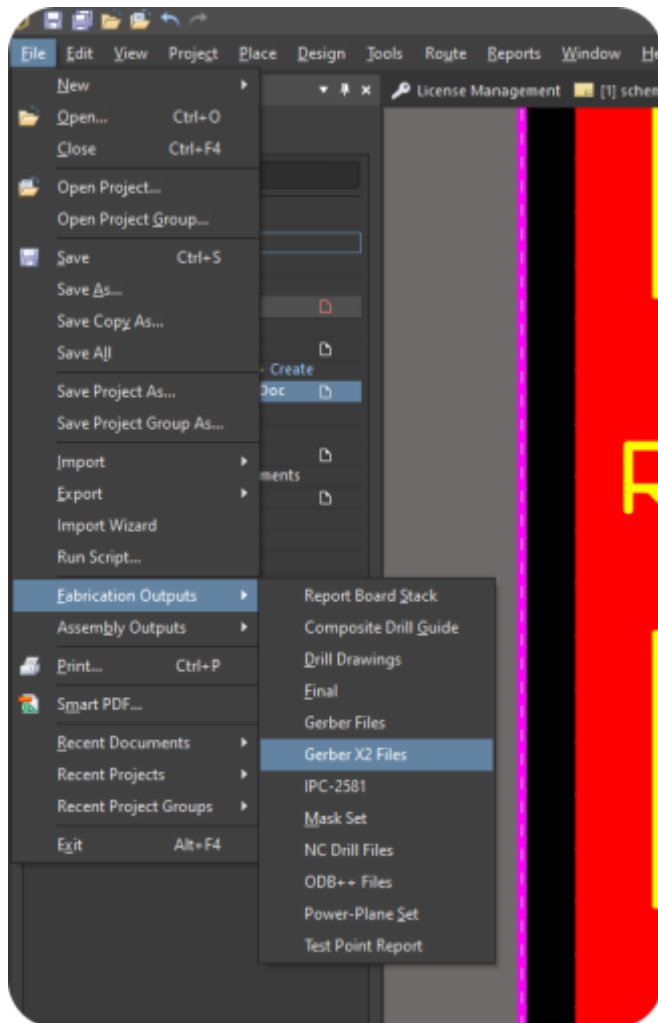
## Fabrication Output

One might wish to send this to a manufacturer to make and send it to your house for you to use and solder, two popular manufacturers are JLCPCB and PCBway.

These manufacturers generally require you to upload a Gerber File and a Drill File.

A Gerber file tells it where to trace the copper and the silk screen while the drill files tells it where to drill holes.

Start by going to File -> Fabrication Output -> GerberX2



This will open a window, check every layers that you might want the manufacturer to consider, include

- Board outline
- copper layers for both top and bottom
- Silk Screen for both, we only need top layer though.
- Solder Mask and Paste Mask
- Plated and Non plated drill files

After selecting everything you might need, click Apply

This should generate a Project Output folder inside your working directory, find it in your files. Select all the Gerber and Drill Files and compress them into a zip file, call it "CoolJoePCB".

name	Date modified	Type	Size
AltiumPCBdoc_wks	21/06/2024 2:36 AM	CAMtastic Apertu...	3 KB
AltiumPCBdoc_wks.EXTREP	21/06/2024 2:36 AM	EXTREP File	2 KB
AltiumPCBdoc_wks	21/06/2024 2:36 AM	Report File	4 KB
AltiumPCBdoc_wks_Copper_Signal_Bot	21/06/2024 2:36 AM	CAMtastic Layer G...	33 KB
AltiumPCBdoc_wks_Copper_Signal_Top	21/06/2024 2:36 AM	CAMtastic Layer G...	32 KB
AltiumPCBdoc_wks_Legend_Bot	21/06/2024 2:36 AM	CAMtastic Layer G...	1 KB
AltiumPCBdoc_wks_Legend_Top	21/06/2024 2:36 AM	CAMtastic Layer G...	19 KB
AltiumPCBdoc_wks_NPTH_Drill	21/06/2024 2:36 AM	CAMtastic Layer G...	1 KB
AltiumPCBdoc_wks_Paste_Bot	21/06/2024 2:36 AM	CAMtastic Layer Gerber Data	1 KB
AltiumPCBdoc_wks_Paste_Top	21/06/2024 2:36 AM	CAMtastic Layer G...	1 KB
AltiumPCBdoc_wks_Profile	21/06/2024 2:36 AM	CAMtastic Layer G...	1 KB
AltiumPCBdoc_wks_PTH_Drill	21/06/2024 2:36 AM	CAMtastic Layer G...	2 KB
AltiumPCBdoc_wks_Soldermask_Bot	21/06/2024 2:36 AM	CAMtastic Layer G...	3 KB
AltiumPCBdoc_wks_Soldermask_Top	21/06/2024 2:36 AM	CAMtastic Layer G...	3 KB
AltiumPCBdoc_wks-macro.APR_LIB	21/06/2024 2:36 AM	APR_LIB File	0 KB
Design Rule Check - AltiumPCBdoc_wks	21/06/2024 2:29 AM	Design Rule Chec...	2 KB
Design Rule Check - AltiumPCBdoc_wks	21/06/2024 2:29 AM	Chrome HTML Do...	11 KB
Status Report	21/06/2024 2:36 AM	Text Document	1 KB

This zip file can be uploaded to the manufacturer website and ready to order!!

The screenshot shows the JLCPCB website interface for configuring a PCB. The main configuration area displays a 2-layer board with dimensions 40.64 x 66.04 mm. The right sidebar shows pricing details, including a special offer of A\$3.00. The bottom section shows shipping estimates and a welcome message.