



Autodesk Eagle

Software Instructions

Software used	Autodesk Eagle		
Version #	1		
Created by	Logan F	Date	04/02/21
Reviewed by	James T	Date	04/02/21

Software Instructions: Autodesk Eagle

Acknowledgments

We would like to acknowledge the following references used to compile these instructions for students:

<https://www.youtube.com/watch?v=v979MUCKVoo>

<https://www.youtube.com/watch?v=bDwTjWpU584>

<https://www.youtube.com/watch?v=SgT2aUhJQHA&t=2821s>

<https://www.youtube.com/watch?v=JtytRPefUxk>

COMMONWEALTH OF AUSTRALIA

Copyright Regulations 1969

WARNING

This material has been reproduced and communicated to you by or on behalf of The Engineering Institute of Technology pursuant to Part VB of the *Copyright Act 1968* (the Act).

The material in this communication may be subject to copyright under the Act. Any further reproduction or communication of this material by you may be the subject of copyright protection under the Act.

Do not remove this notice.

Table of Contents

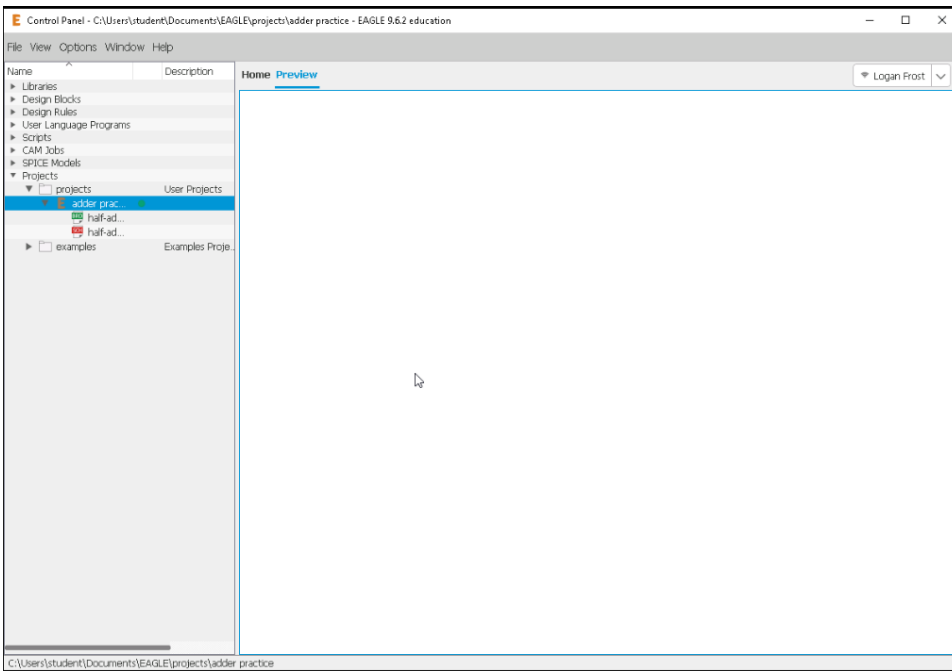
1. Intro
2. Creating a Schematic
3. Creating the PCB
4. SPICE Simulation

Software Instructions for Autodesk Eagle

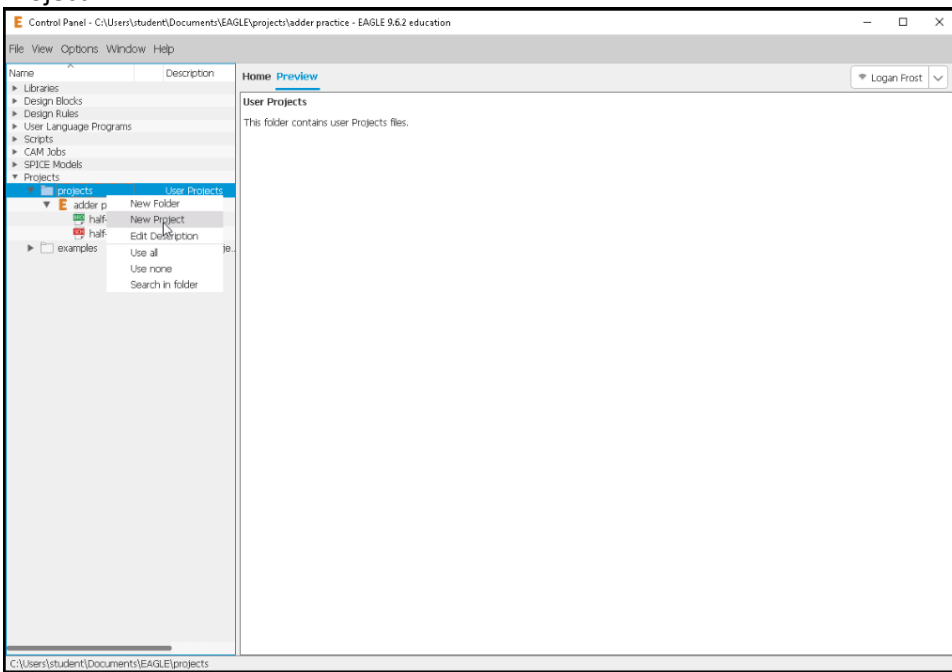
1. Intro:

Starting your project

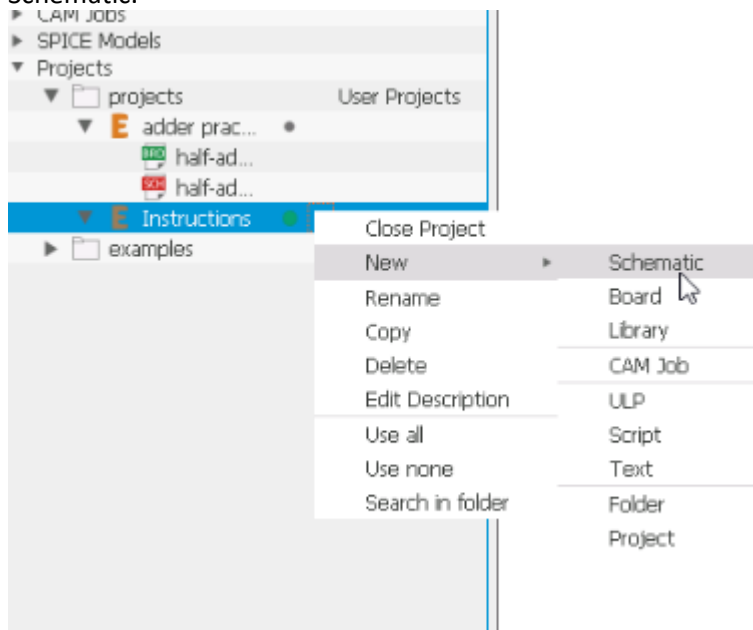
When you open your new project, you will be presented with the following screen.



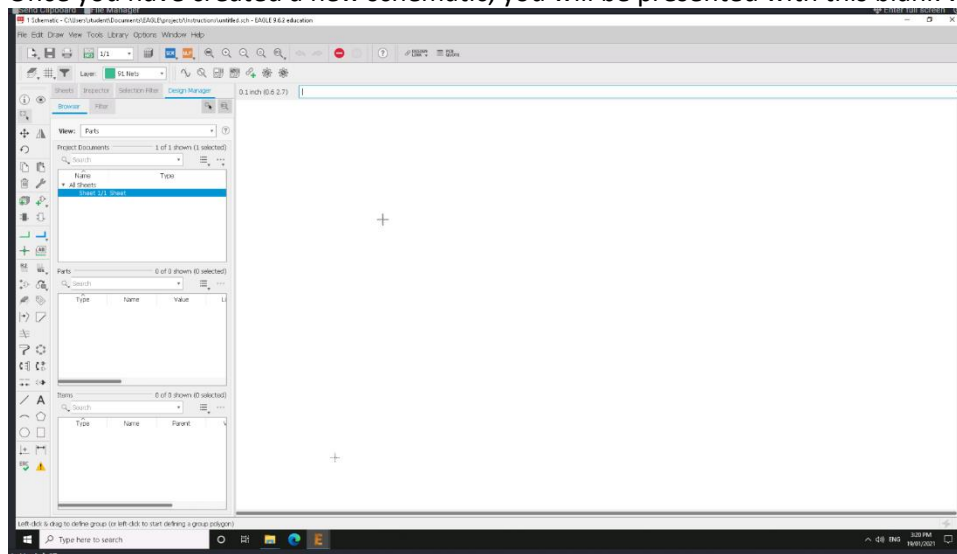
To create a new project, drop down the projects tab, right click on the new projects folder, and select New Project.



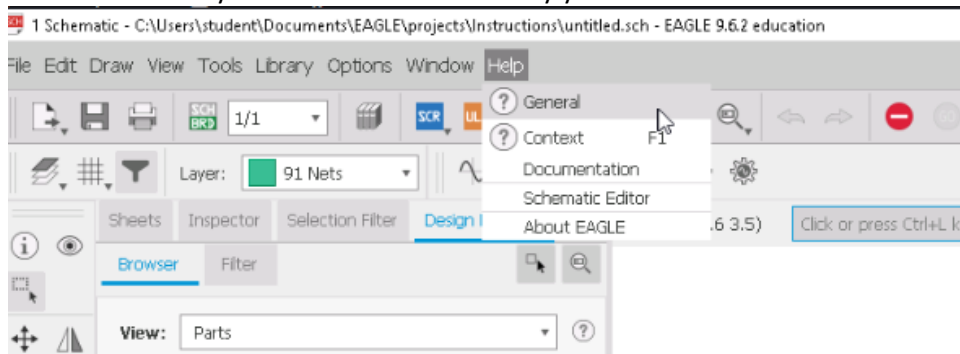
To begin working on your new project, right click on the project you just created and select new -> Schematic.



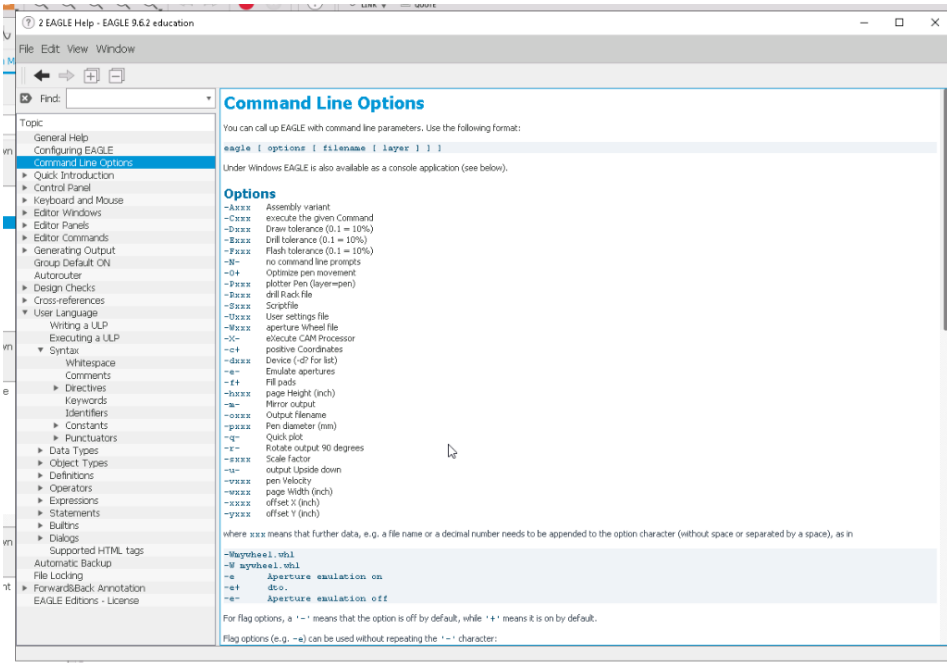
Once you have created a new schematic, you will be presented with this blank workspace.



The Help menu is very useful in this software. Following the general or documentation help menus, you can find out about any commands or functionality you need.



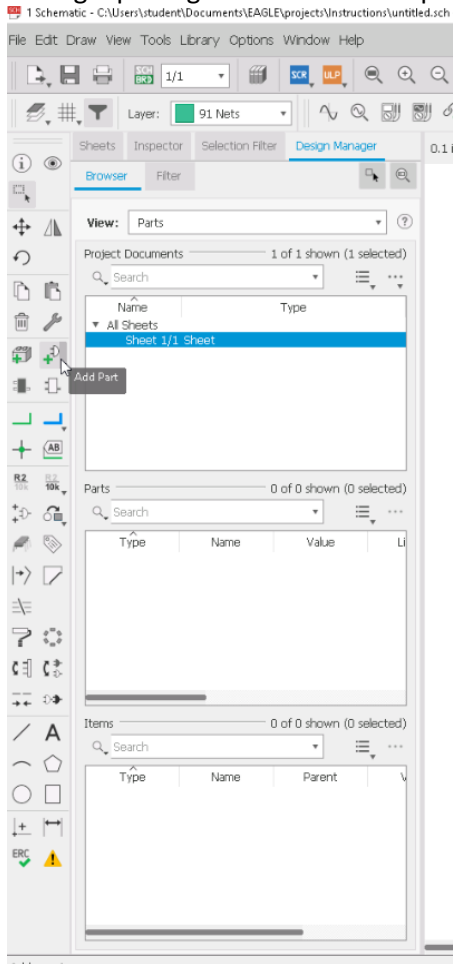
Here is a preview of the option and tabs you have available to you in the help menu.



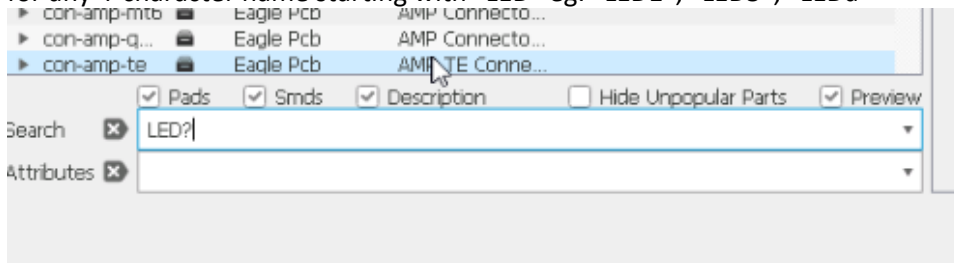
2. Creating a Schematic:

Placing Components

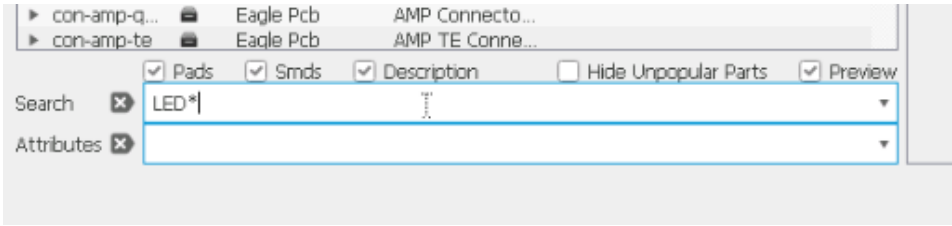
To begin placing the electrical components for your schematic, select the Add Part button in the toolbar.



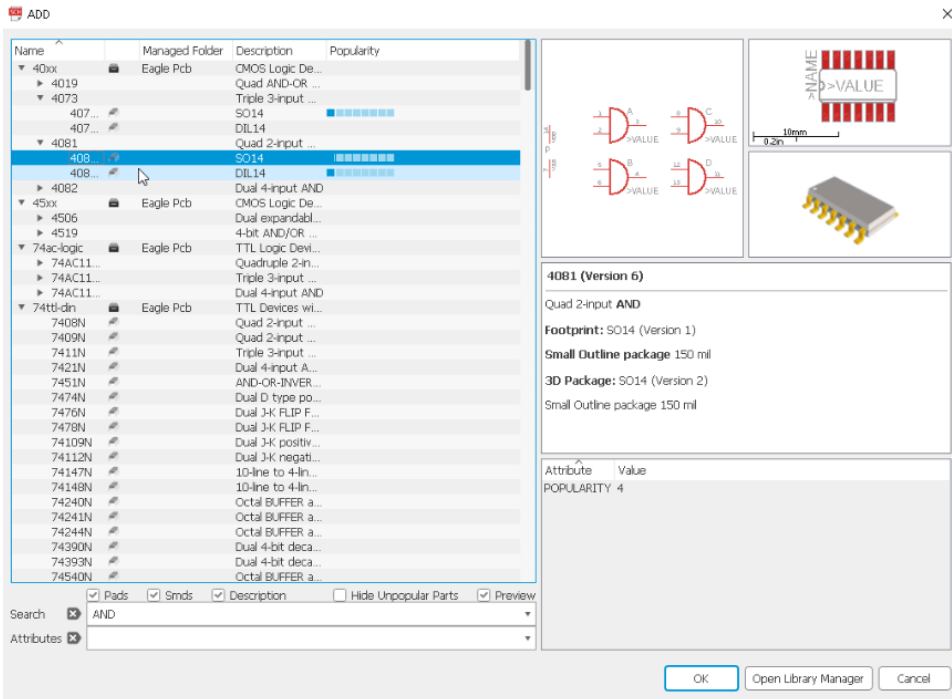
In the parts menu, your search term will be searched for exact matches in either the name or the description of the parts. To get around this, you can use wildcards to broaden your search. Using a “?” acts as a wildcard and will search for results with any character in that position. In this example, the search will return results for any 4-character name starting with “LED” eg. “LED1”, “LED5”, “LEDa”



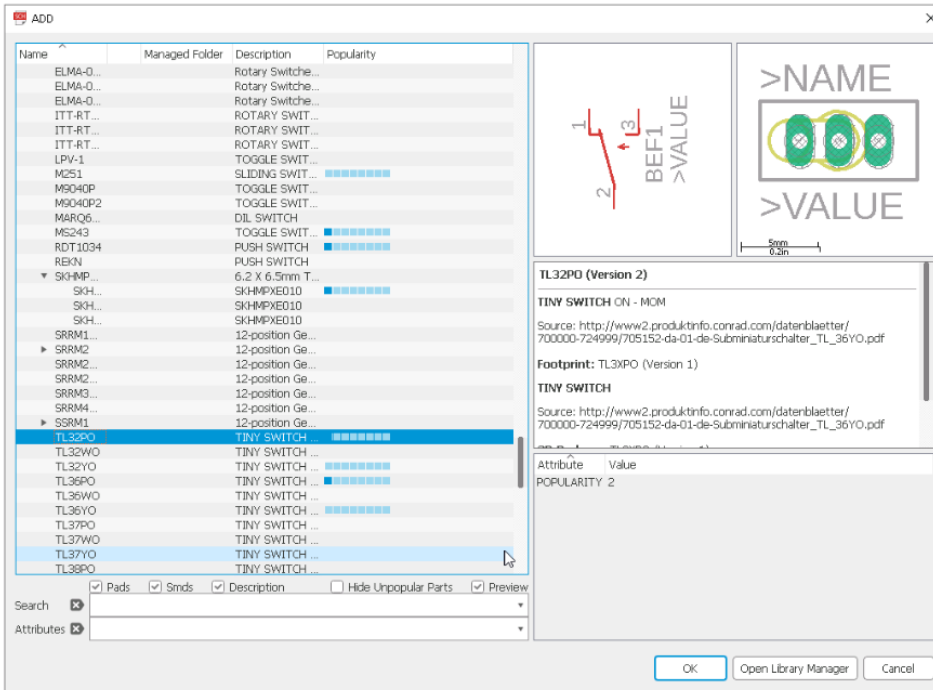
Using an “*” acts as a wildcard as well, representing any number of characters. The search in this example will return any string starting with “LED” eg, “LED”, “LED1”, “LED55”, “LED12345678987654321”



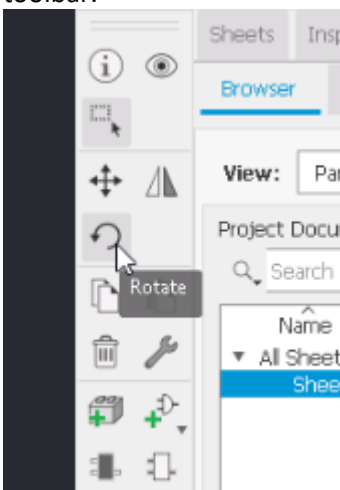
Once you have found the part you are looking for, click the name to select it. On the right of the window is a preview of the parts and their description. With the part selected, click the OK button to begin placing it. Click anywhere on the schematic to place the parts. You can place as many as you'd like without needing to reselect the part.



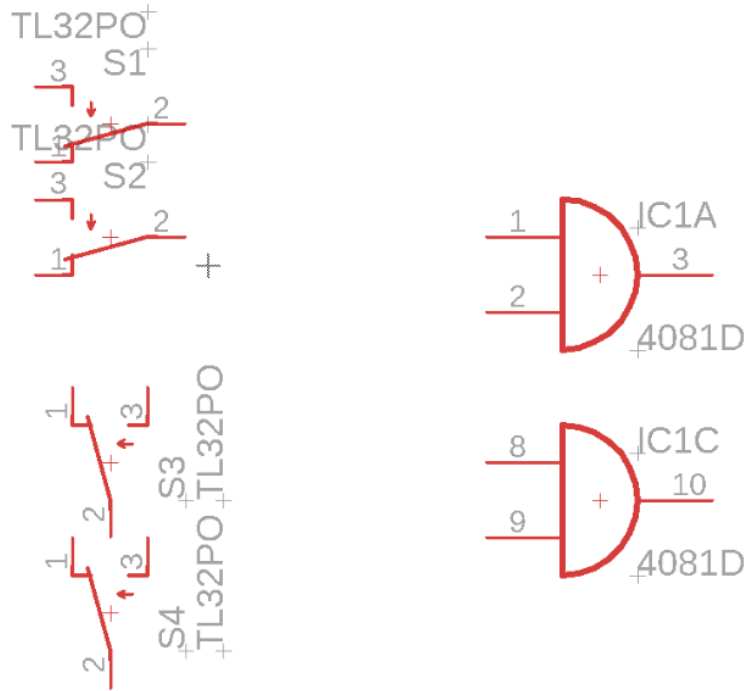
Once you have finished placing your first component, you can press the ESC key on your keyboard to open up the parts menu again to choose the next part you'll place.



When placed, the parts will all be the same orientation. You can rotate the parts using the rotate tool in the toolbar.



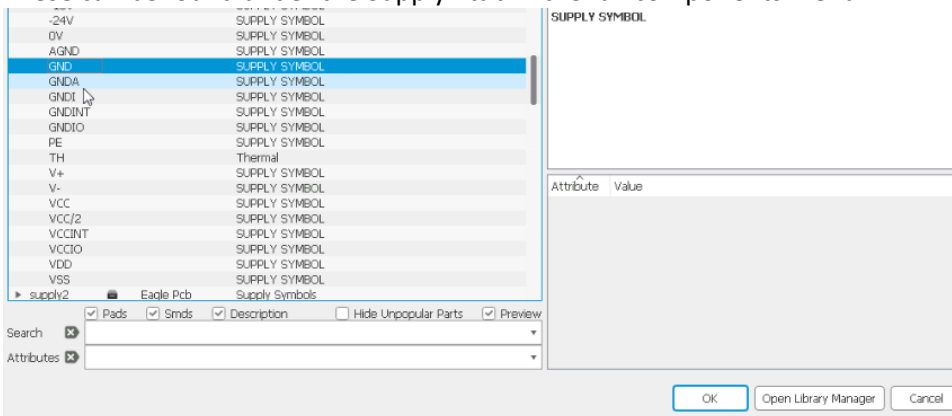
Click the central cross of the part you want to rotate. This will rotate the part by 90° in a direction. Repeat this until the parts are rotated to your liking.



Another part you can add is a supply part. This includes power and ground components. These supply components can be placed any number of times in the schematic and will be treated as though they are linked.

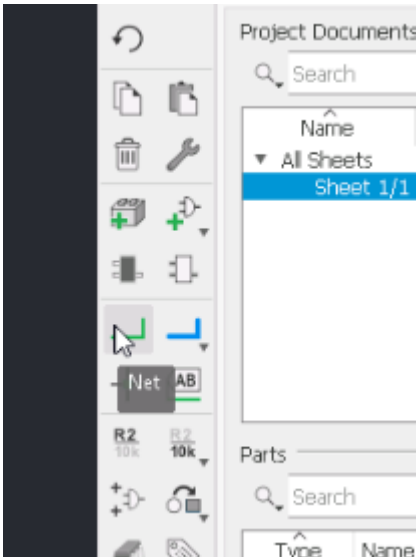


These can be found under the Supply1 tab in the full components menu.

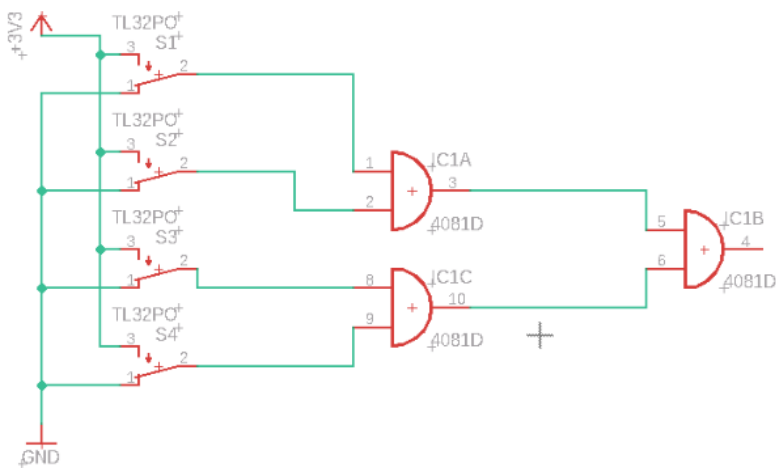


Connecting Your Parts

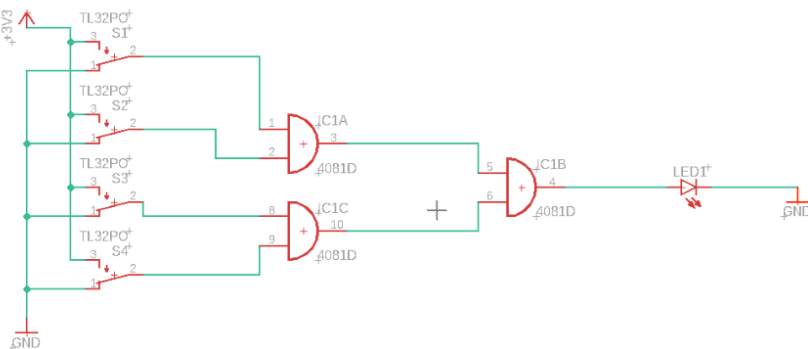
Using the Net tool in the toolbar, you can begin connecting your parts.



Click the node on your components you want to start connecting from and then click on another node to complete the connection. You can also begin drawing a net anywhere but you will need to end to net on a node.

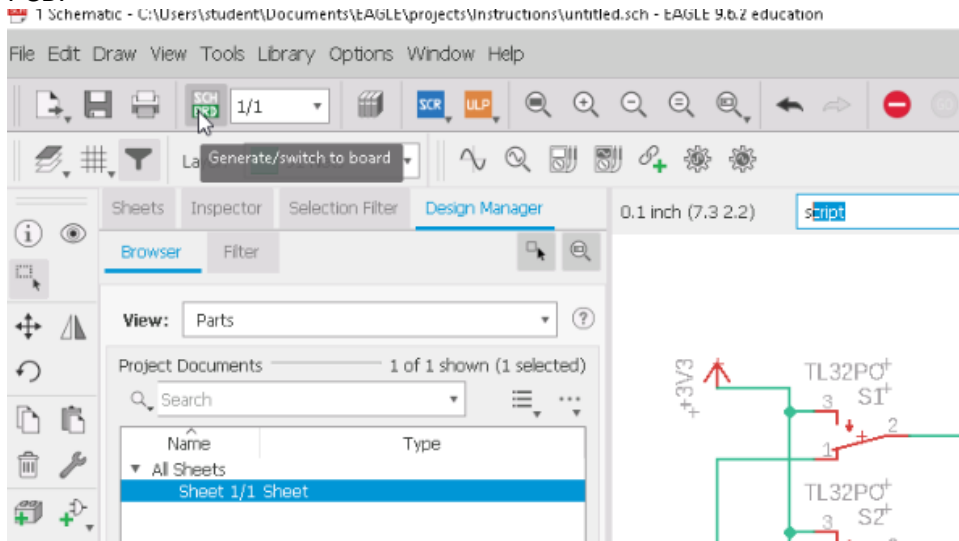


You can place source and ground components anywhere and as many times as you like. All ground connection will be connected when brought into the PCB builder.

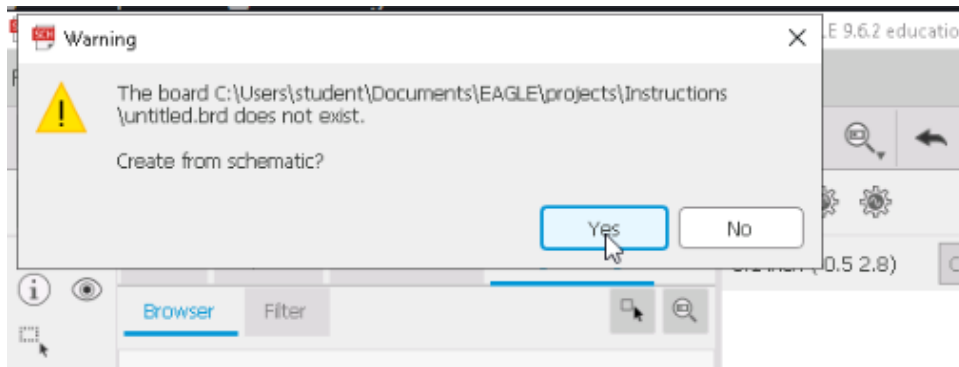


3. Creating the PCB:

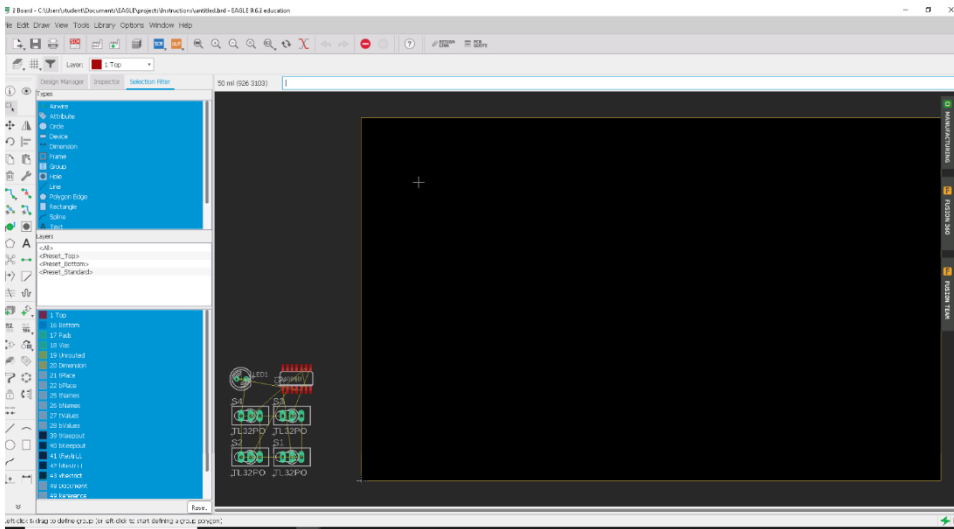
Once you have your circuit prepared, click the “Generate/switch to board” to transfer the circuit to a PCB.



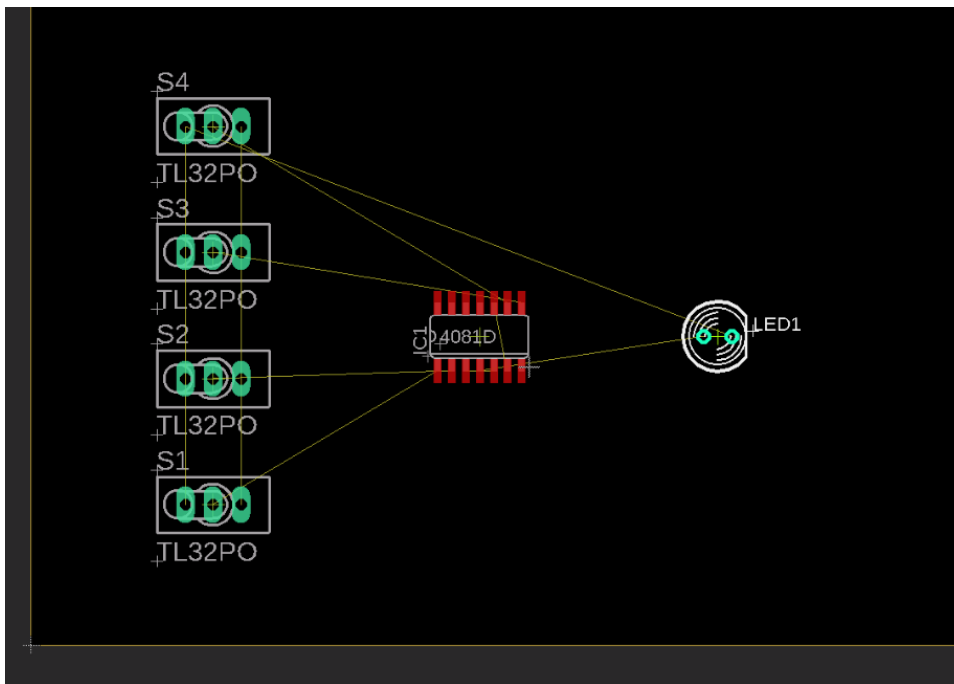
When you do this the first time in your project, you will be asked if you want to create from schematic. Click yes.



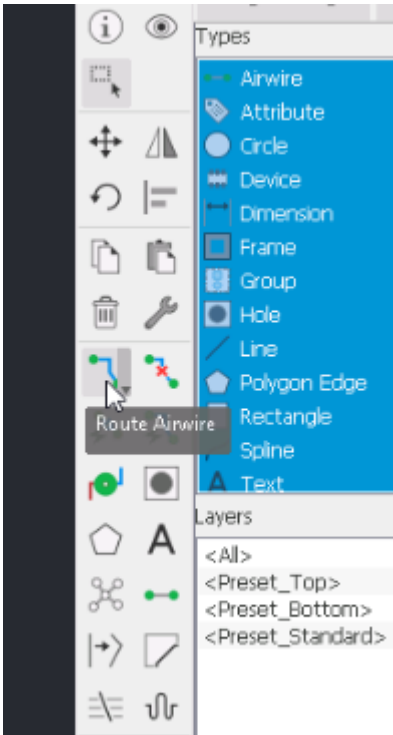
You will be presented with a screen of one colour, and a rectangle of a darker colour. The darker rectangle is the PCB.



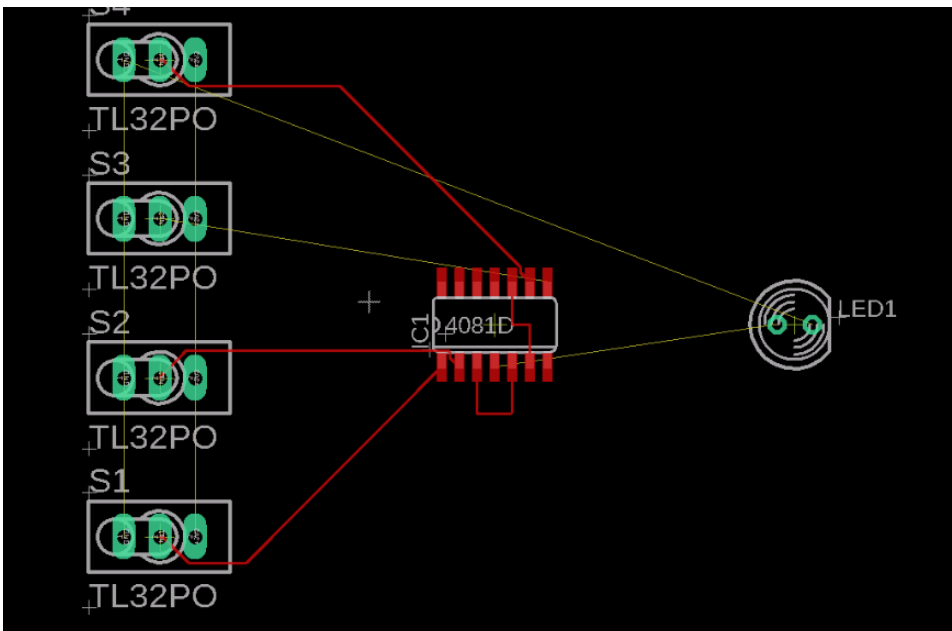
You can move the components from the left into the board. Adjust their positions to how you'd see them on the PCB. Try to avoid the lines crossing too much. The connections can be made later to ensure nothing gets tangled.



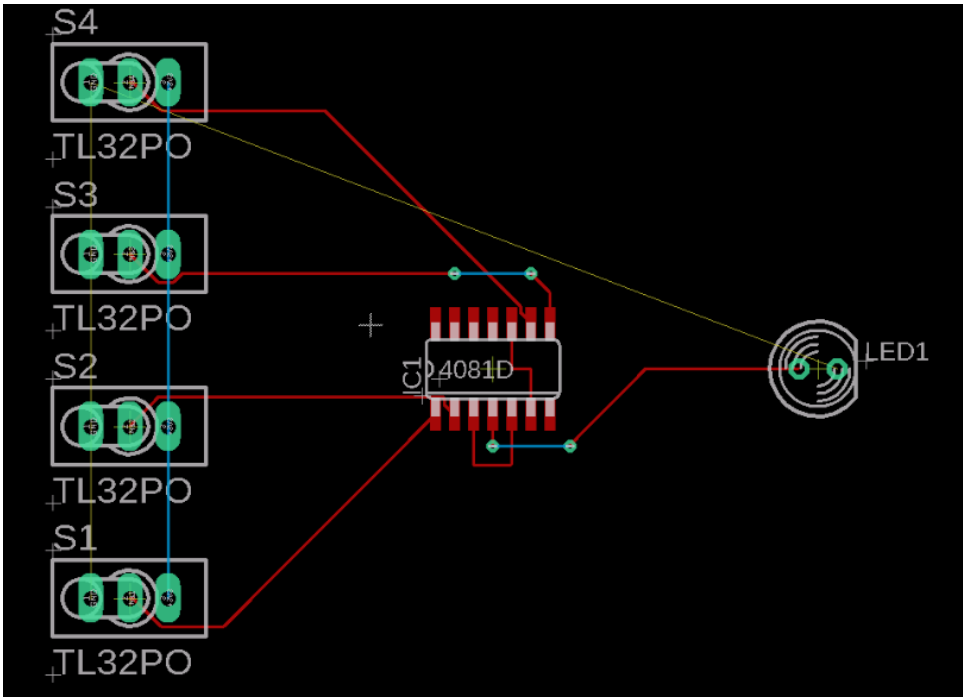
To begin connecting the components on the PCB, select "Route Airwire".



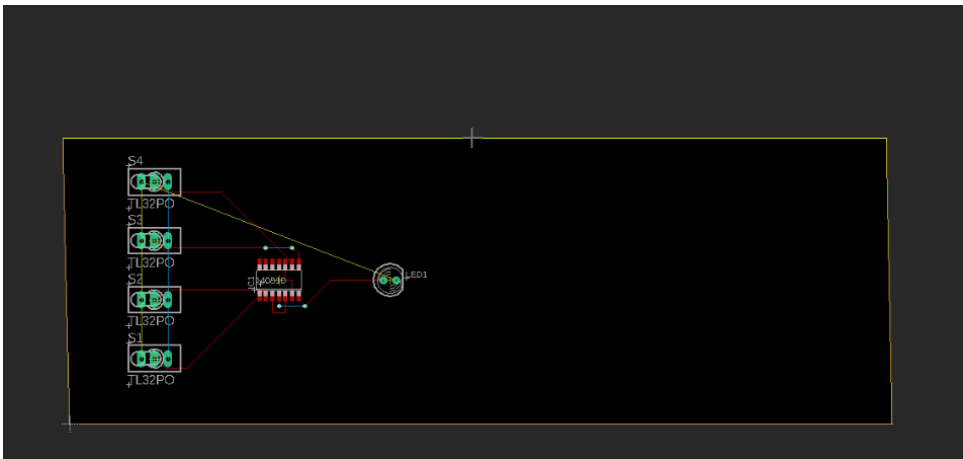
Click the node you want to start drawing from and draw towards the correct connection. The lighter lines will update as you draw to show where the connection should reach. There are two options in the upper toolbar for drawing these traces. One pushes aside already placed traces, and the other lets the trace you are drawing walk around the other components.



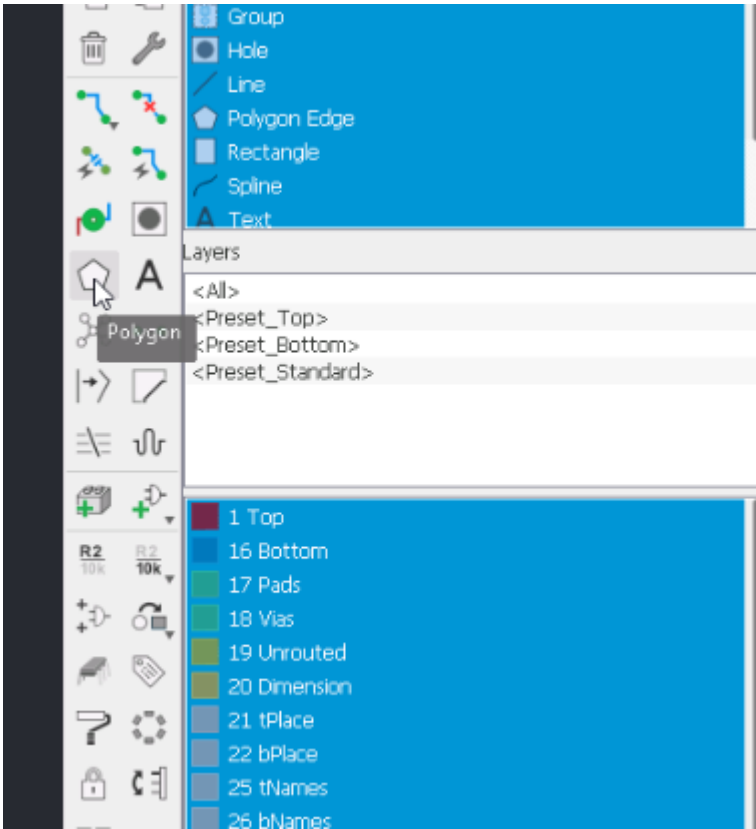
When drawing the traces, you can press the spacebar to switch the trace to the opposite side. This is useful for letting lines cross over each other.



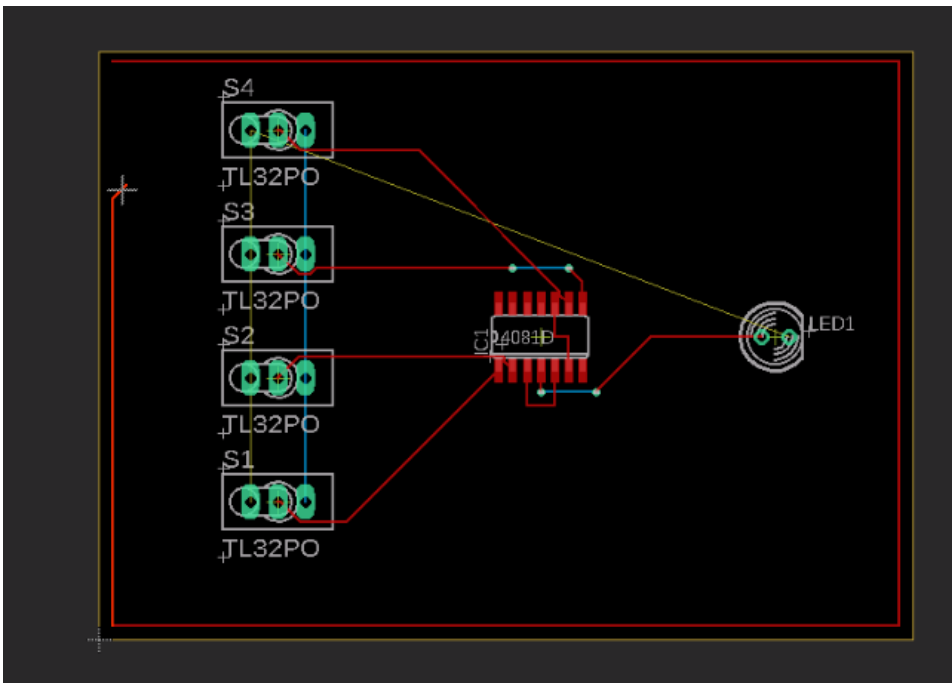
You can click and drag the outlines of the PCB board to adjust the size of the PCB itself.



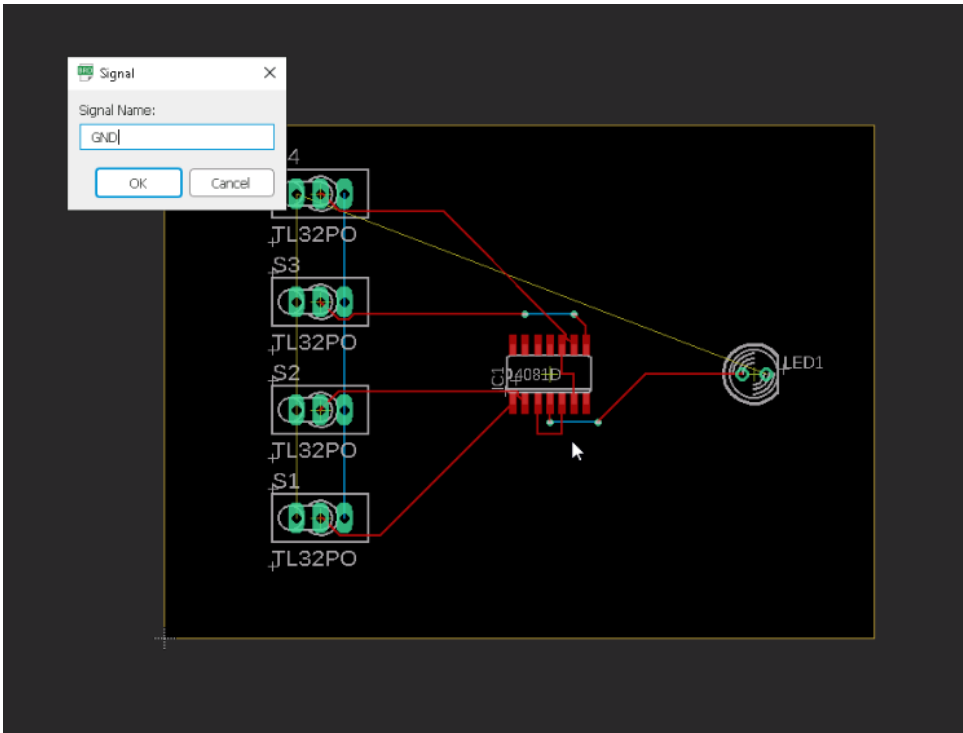
Another connection that needs to be made is with the power source. Select the polygon tool.



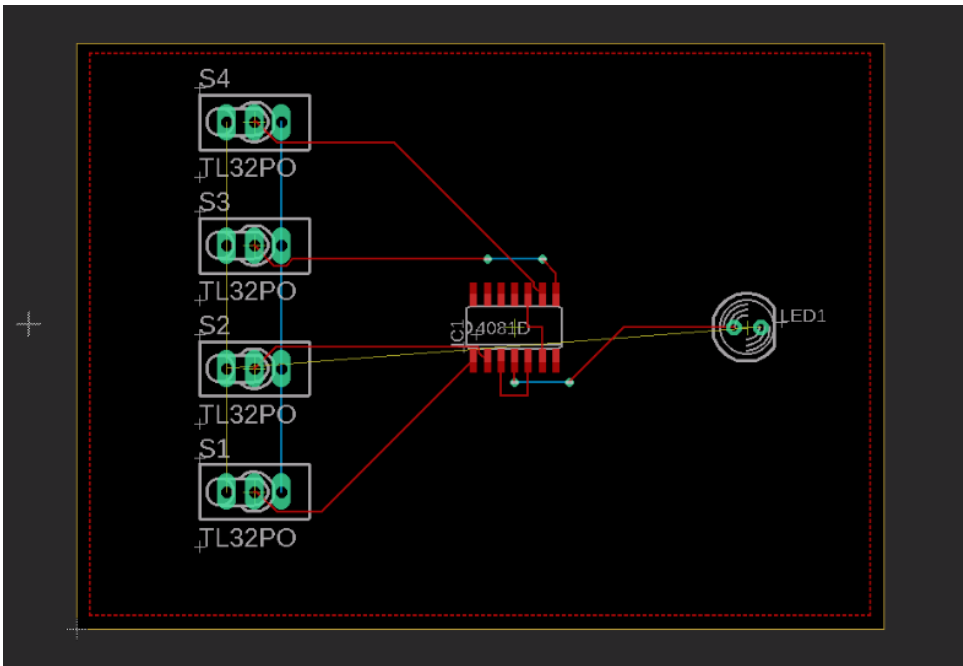
Draw an outline using the tool which covers all of the necessary components.



Once the drawing is complete, select either the ground or power connections to set the polygon to.



Here is what the complete polygon looks like.

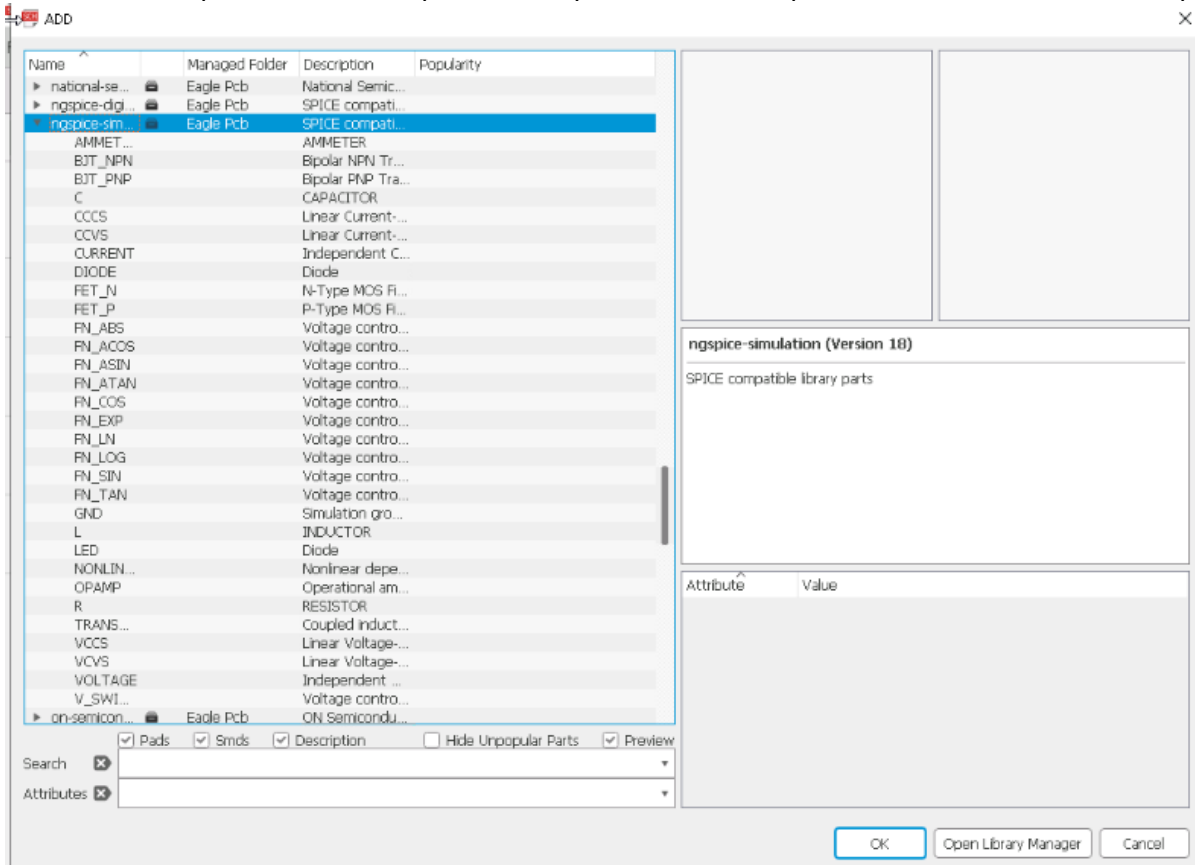


Click the ratsnest tool to complete the polygon. This will fill the polygon with contacts connecting all of the nodes of the connection type you chose earlier. This fill will stay separate from the other traces on that layer.

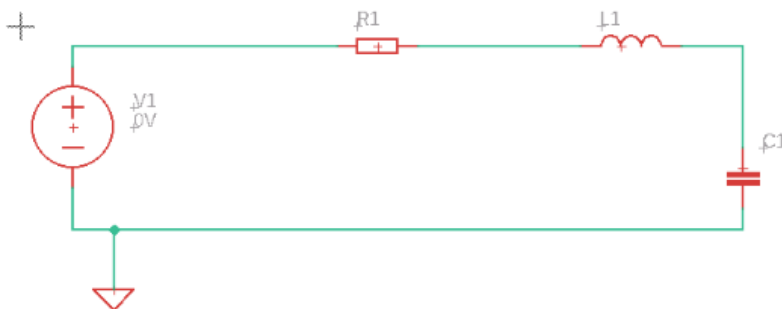


4. SPICE Simulation:

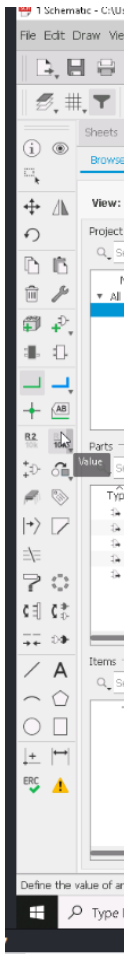
In the add components menu, open the drop down labeled “spicesim” to access simulation parts.



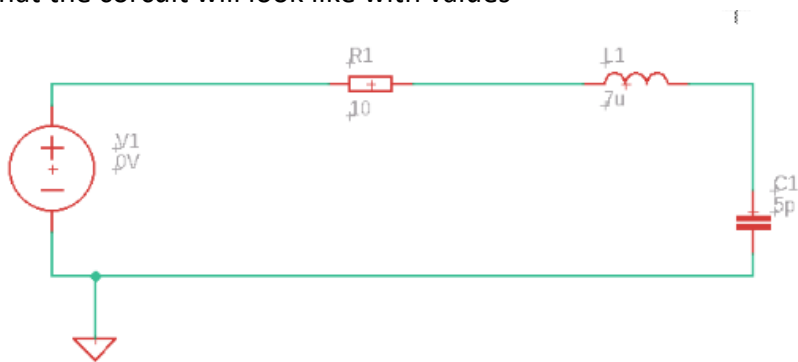
Using the components in that menu, build the circuit you’d like to simulate.



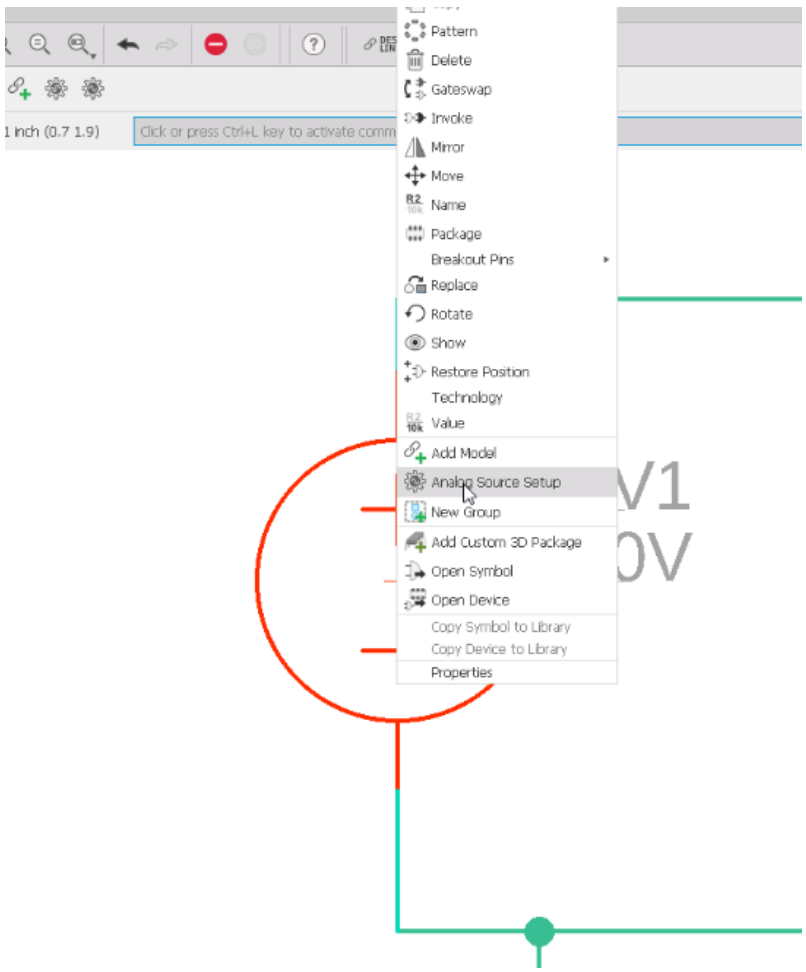
Using the Value tool, assign all of the components their values. Do not use the value tool on the voltage source.



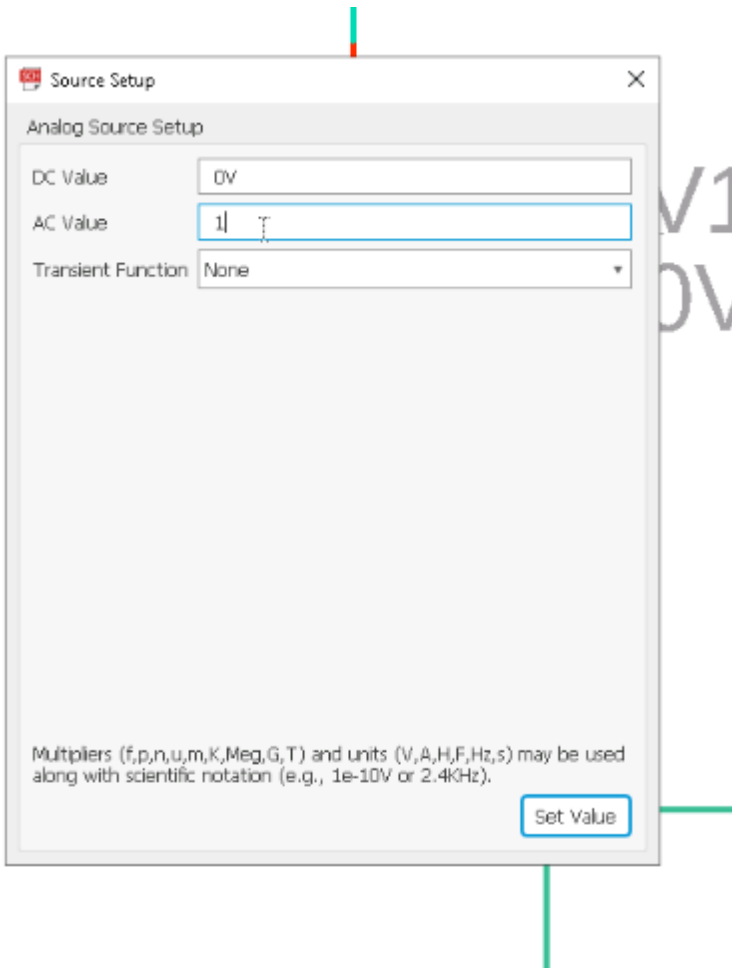
Here is what the circuit will look like with values



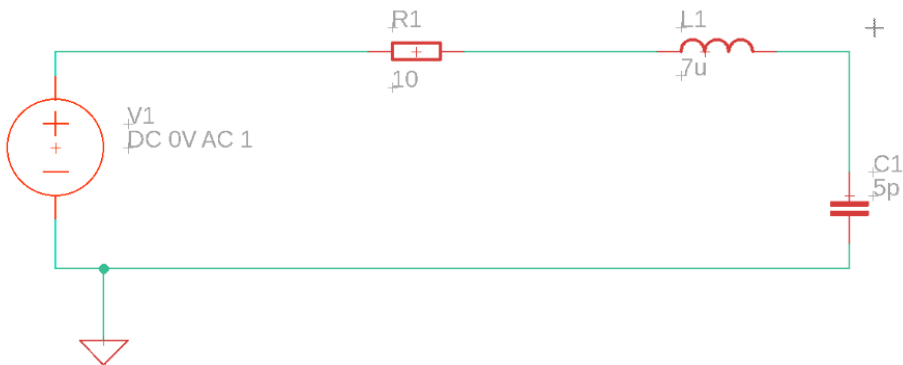
To assign a value to the voltage source, right click the voltage source and select Analog Source Setup.



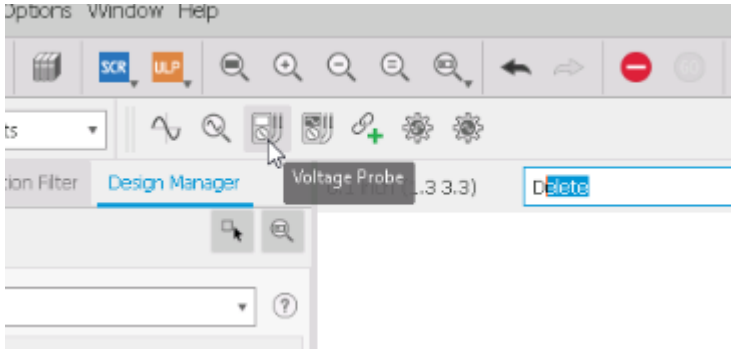
Input your values and click “Set Value”. In this case there is no DC value because we are doing an AC sweep simulation.



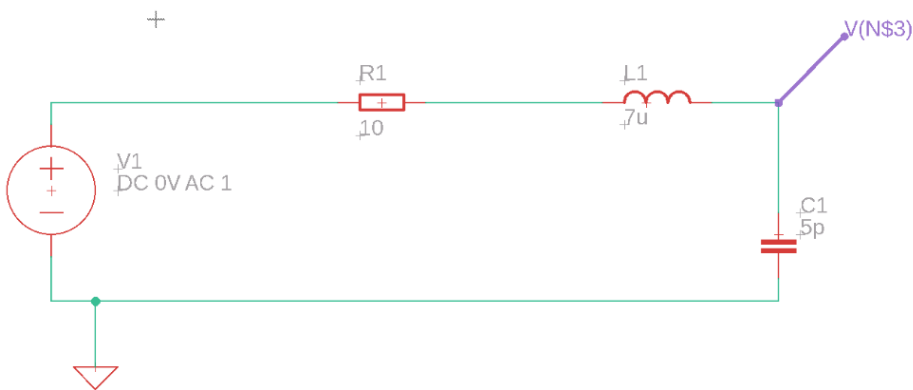
Here is the result.



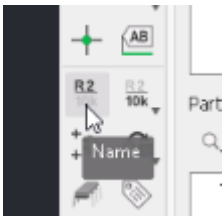
To gain data in the simulation, you will need to add probes. In the toolbar, select Voltage Probe.



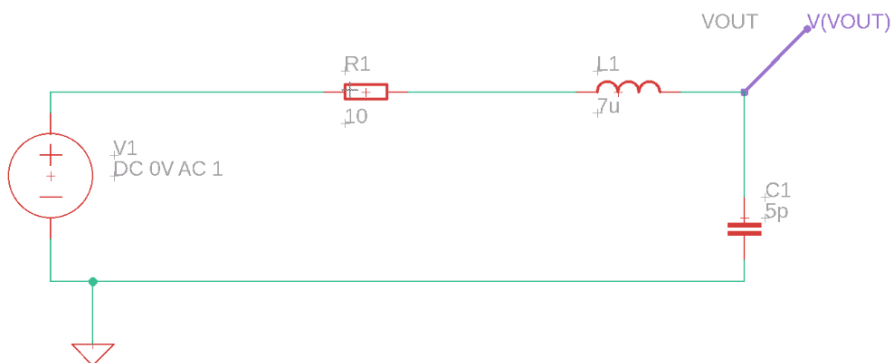
Choose where to place your probe in the circuit.



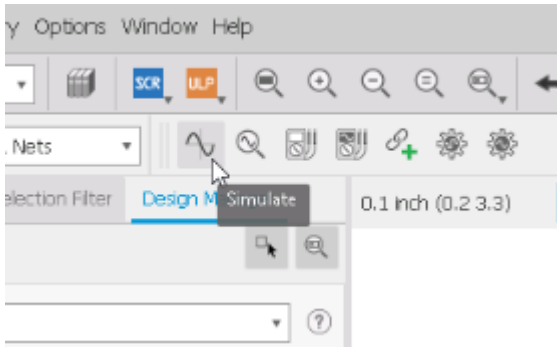
N\$3 isn't the most descriptive name, so you can use the "name" tool in the toolbar to change the name of that node in the net.



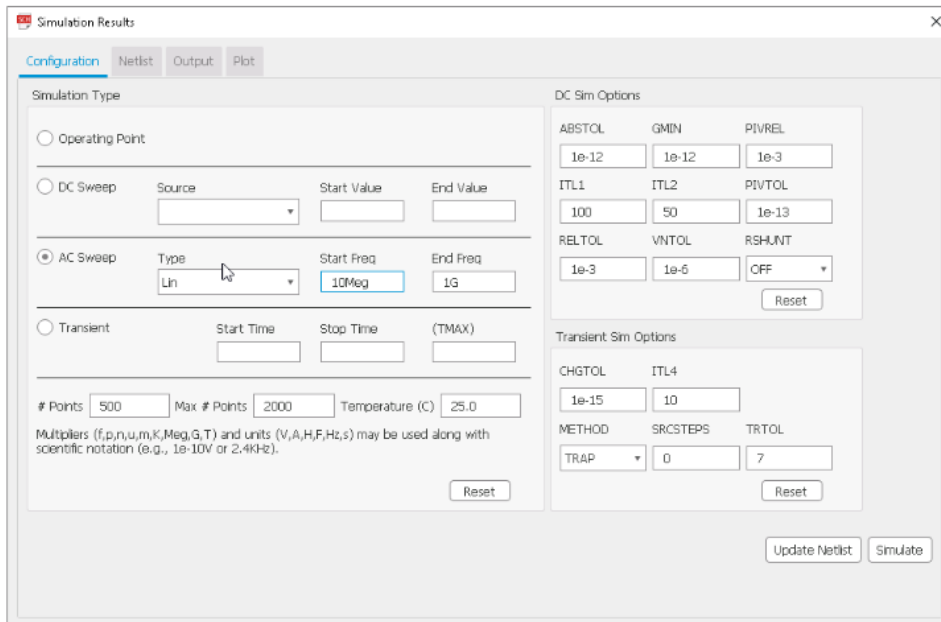
Here is my outfit named as VOUT.



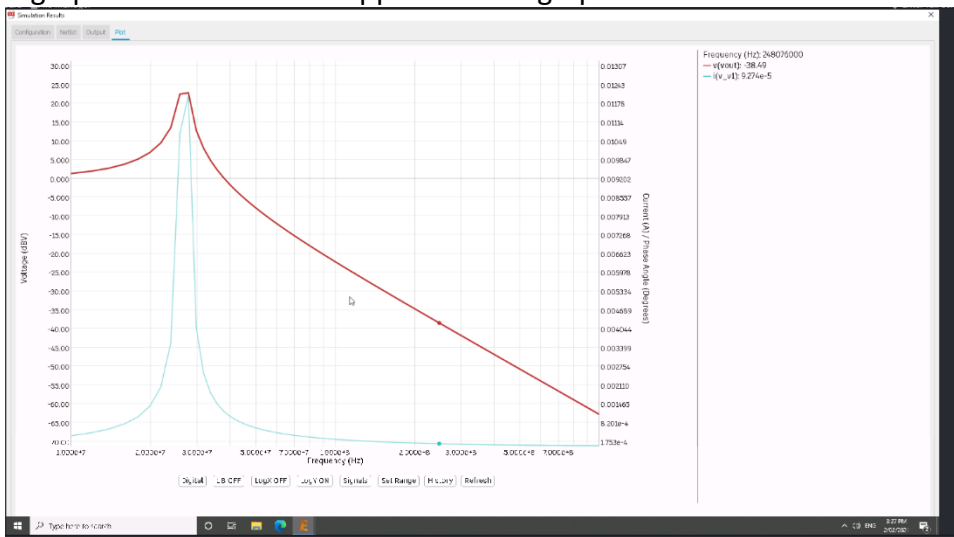
Once you've placed all of your probes, you can click the "Simulate" button.



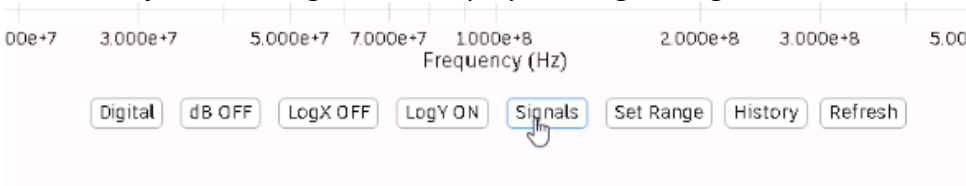
In this case we are doing an AC sweep, so fill in your simulation details and click simulate.



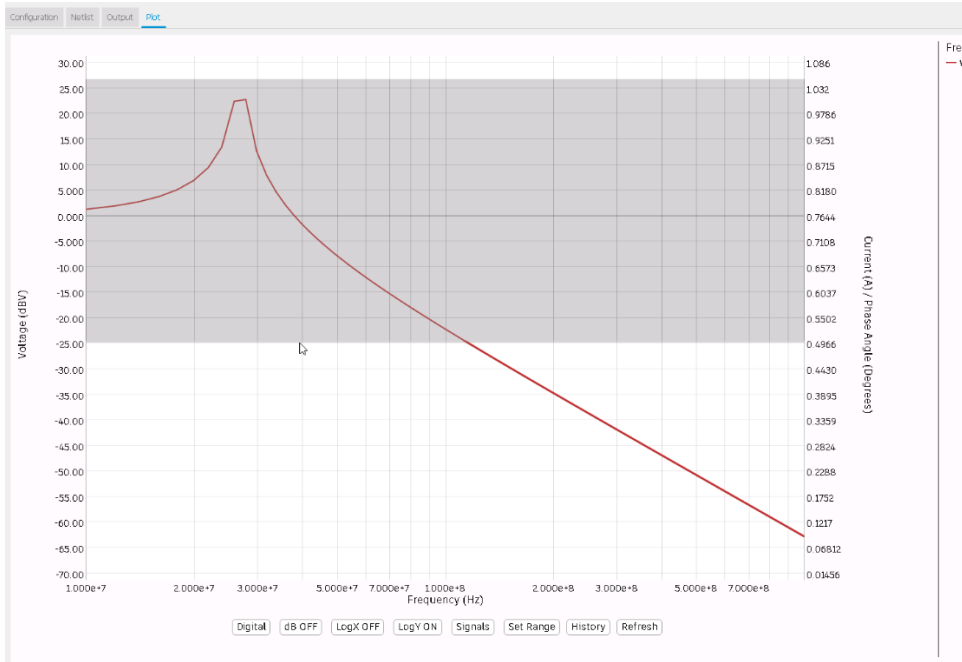
A graph of the results will appear. In this graph there is a VSource and a VOUT line.



You can adjust which signals are displayed using the Signals Button.



You can click and drag to select regions of the graph to zoom in on either vertically or horizontally. Any of these adjustments you make can be undone using the refresh button.



Under the output tab, you can see all of the data points that were used to make the plot.

Index	frequency	voltage	current
0	1.00000000000000e+007	0.00000000000000e+000	0.00000000000000e+000
1	1.160312229790391e+007	-4.229559269336404e-003	-1.328786648768787e-006
2	1.198396793587174e+007	0.00000000000000e+000	0.00000000000000e+000
3	1.396793587174349e+007	0.00000000000000e+000	0.00000000000000e+000
4	1.595190380761523e+007	0.00000000000000e+000	0.00000000000000e+000
5	1.793587174348597e+007	0.00000000000000e+000	0.00000000000000e+000