

EAGLE PCB Introduction

Based on Eagle 9.4.2

Tim Steffes
Matt Smith
Mark Brehob
Revised 8/12/19 MS

Introduction

This tutorial will provide you with an introduction to board layout CAD tools. These tools will allow you to start with schematic, place your components, create all the component connections based on your schematic and finally create the files needed to have a PCB (printed circuit board) house fabricate your PCB. There tools range from free hobbyist level capability to expensive pro-tools. We will use Eagle for this lab. It is a fairly representative tool used by hobbyist and pros alike. A free version is available for students.

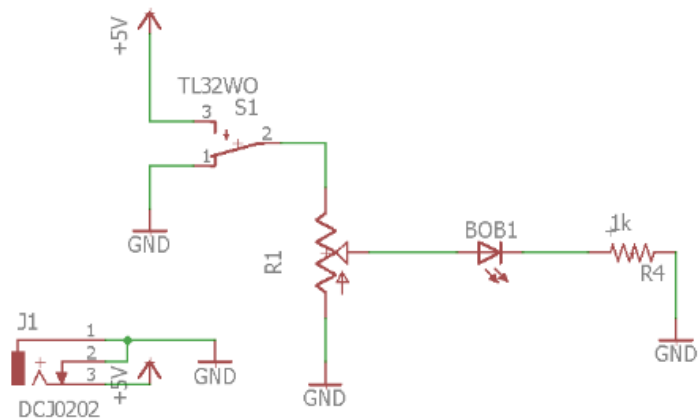
Start Eagle

Eagle is installed on the lab workstations and CAEN loads or you can download it to your personal computer. Open Eagle and you will be prompted to login or create an account. A student email address will qualify you for free use.

Altium is another CAD tool that is installed in the lab and CAEN machines with license support. It is considered a more advanced tool than Eagle and does not offer a student license. If you have used Altium in the past and prefer to use it, please feel free to do this assignment with it instead of Eagle, but you will have to use the lab or CAEN machines.

The Circuit

We will start with something simple to design. The following schematic consists of switch, LED a few passive components and a jack to connect to a power source.



Enter the Schematic

Let start by entering the schematic. Open the schematic editor with File→New→Schematic. A window will open showing something like that shown in Figure 1.

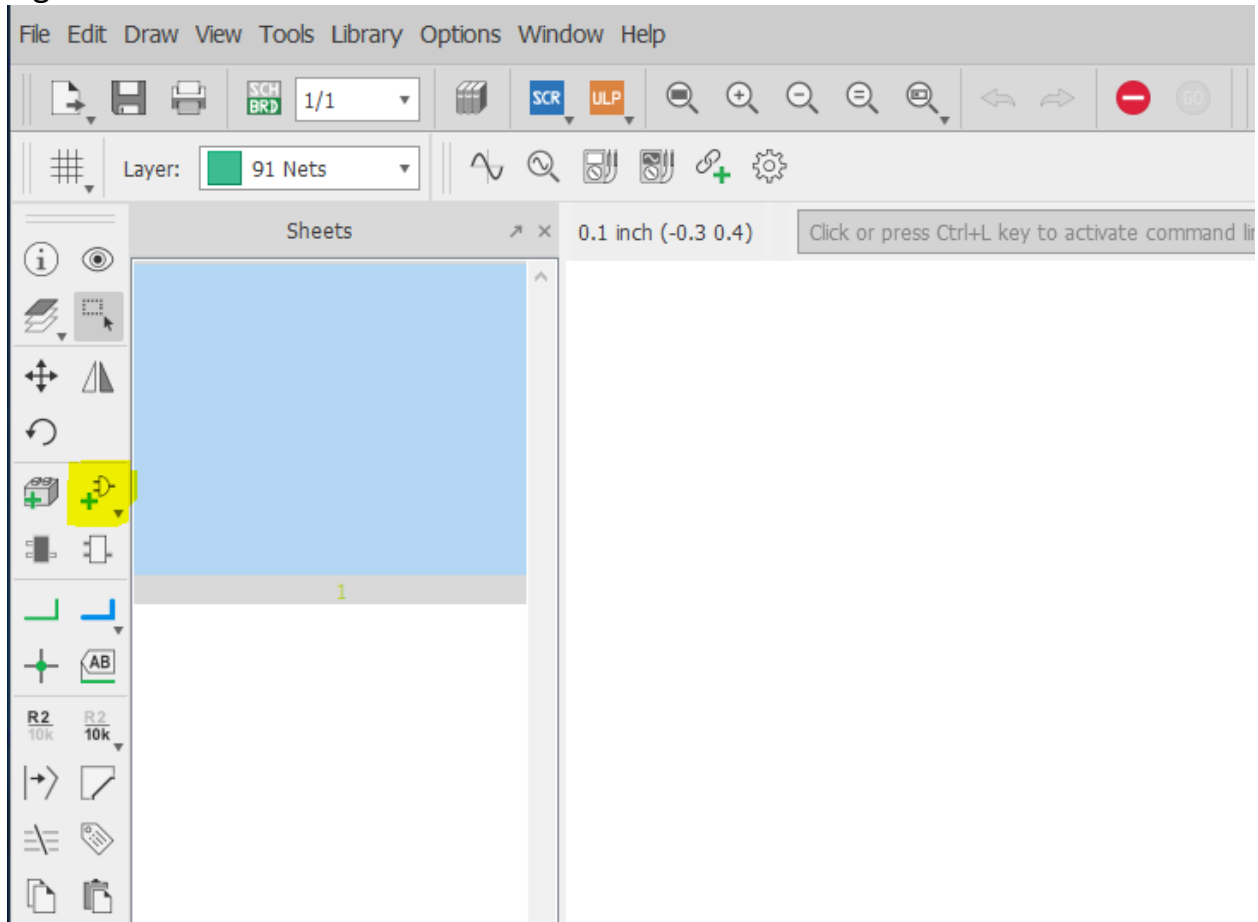


Figure 1

Adding Components

Let's start with the LED. It is not enough to choose a LED symbol. We will have to select a "real" part from the Eagle library. We are going to use a 5mm, thru hole, red LED. Eagle has libraries consisting of many "real" components that include mechanical descriptions of the component that will be used when we layout the PCB.

To open the library, click on the **add components icon** yellow highlighted in figure. A new window will appear with a listing by manufactures name and generic type.



Figure 2

Using the search bar, search for "led". Under led you will see various types can categories. Look for sub category called LED. Then under this category look for LED5MM. You should see this device.

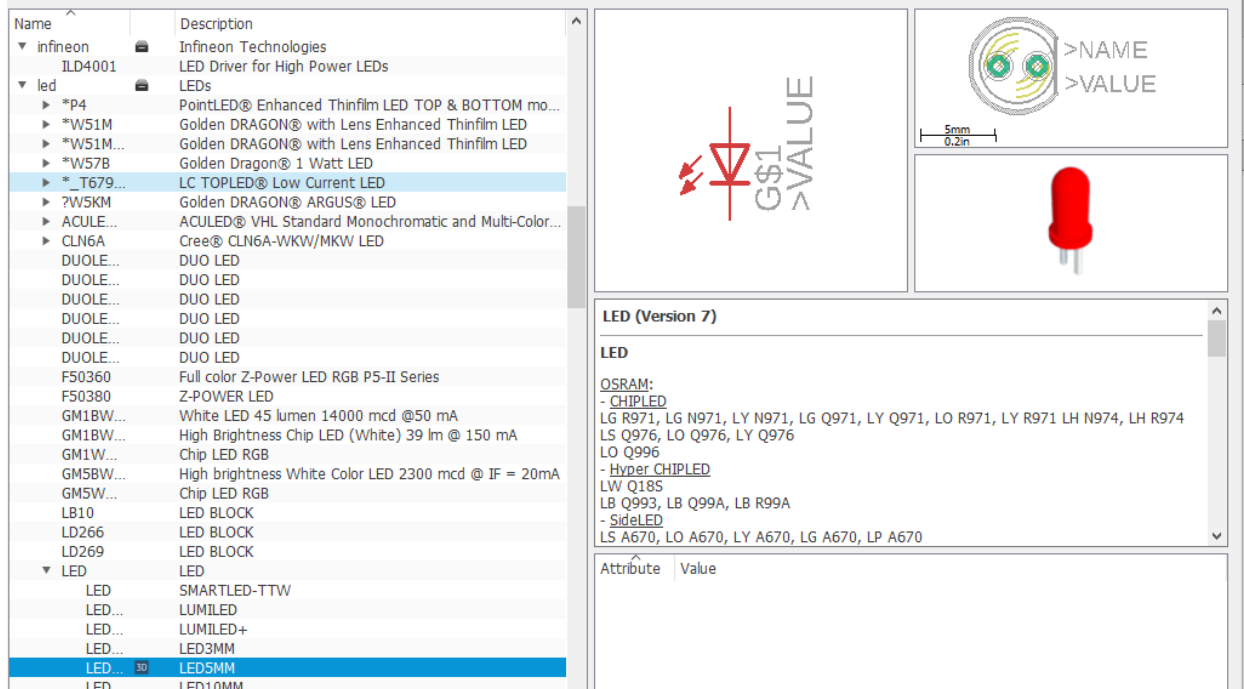


Figure 3

Click OK and add the symbol to the schematic canvas. Notice you can add more LEDs by simply clicking. To exit the add component mode hit ESC a few times.

Notice that Eagle named your LED LED1. If you wanted to name it something else, you can use the name tool labeled R2/10k. The message bar at the bottom of the Eagle window will guide you thru the process. It does this for other commands too.



Add the rest of the components.

- A current limit resistor. Search for: Resistor→US-R→R-US_0309/10. Set the resistor value to 1k. To do this edit attributes with the R2/10k tool.
- A potentiometer to vary the LED brightness. Search for: TRIM_US-RJ9W. Use the attribute tool to make the value 10k.
- An on/off switch. Search for: switch→TL32WO.
- A DC jack for power input. Search for con-jack→DCJ0202.



You should have the following components in your schematic canvas.

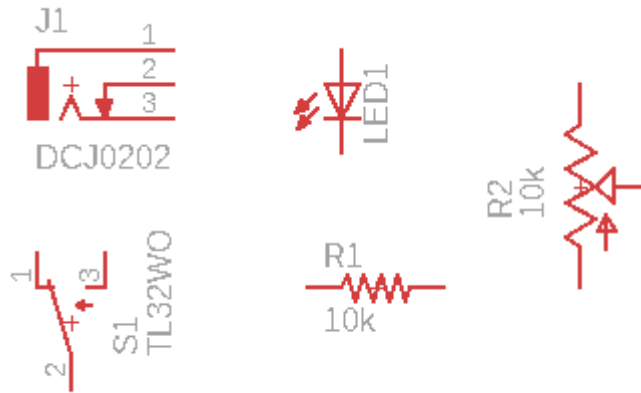


Figure 4

Connecting Components

Components are connected using the NET tool and moved or oriented with the Move tool. Start by arranging the components with the Mover tool. First select the Move tool (cross with arrows), then left click to on the component to select. You can now move the components. Right clicking rotates the component when it is selected. Left clicking again deseles the component. Select the Net tool (green right angle), left clicking on a connection point on the component will start a wire. Continue to the component end you wish to connect to. Left clicking again will attach the wire. Our circuit will be powered by from the jack component J1. Five volt power will be supplied from pin 3. We could just hook up a wire to the switch, but will instead use a special symbol called 5 volts. You can get with the component tool by typing +5V in the search bar (use supply 1). Add it two to your schematic and connect it to pin 3 of the J1 and pin 3 of S1. The 5 volts symbol will act like a connection between the jack and switch. We can now do the same thing using the GND symbol. Search for GND (supply 1). Using the supply symbols keeps the clutter down in our schematic and will be used in the PCB layout to provide some special connections.



When you are done, you should have something that looks like this.

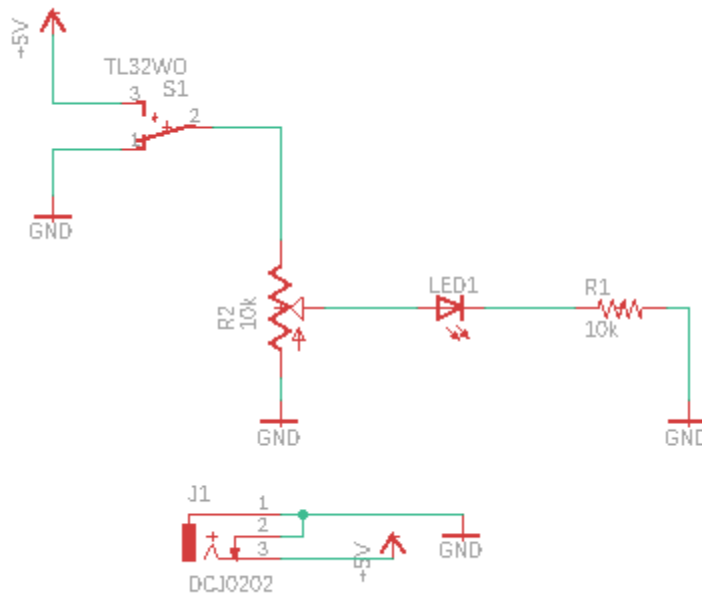


Figure 5

Using Net Names to Make Connection

The GND and +5V symbols essentially form connections labeled GND and +5V. You can do something very similar by replacing wires with labels and effectively make a connection. In this simple schematic it will not make much difference, but in larger more complex designs it can go a long way to simplify the schematic.

You can try with the schematic you just entered. Remove the connection between LED1 and R1 with the delete key. Add a small segment of NET to LED1 and R1. If you left click twice you can end the wire. Next use the Name tool and select one of the net segments. You should be prompted to name the net. Name the net to BOB and place the label. Do the same for the other segment naming it BOB. You should be asked if you want to make a connection with the other net BOB. Confirm and place the label. Now you have a connection between LED1 and R1.



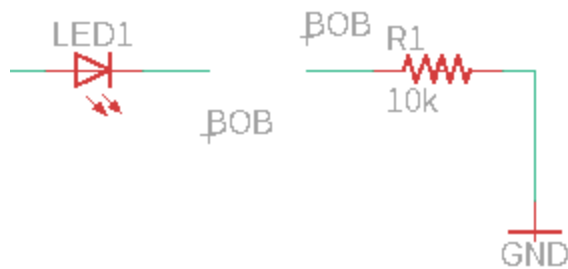


Figure 6

Adding Libraries

More components can be added to the Eagle library. Manufacturers and suppliers provide libraries for Eagle and other popular CAD tools. SparkFun and Adafruit are two such sources that provides many parts you are likely to use in your projects. The following link will show you how to install them.

<https://www.autodesk.com/products/eagle/blog/library-basics-install-use-sparkfun-adafruit-libraries-autodesk-eagle/>. These are installed on the lab stations.

If a component does not exist, it may necessary to build a custom library component. We won't do that in this tutorial, but you can find a tutorial here.

The Board

With the real components specified and their connections, we are ready to do the layout design for our printed circuit board. Save the schematic and then select "File→Switch to Board". You should see something like the following with a default board in black and your connected components to one side.

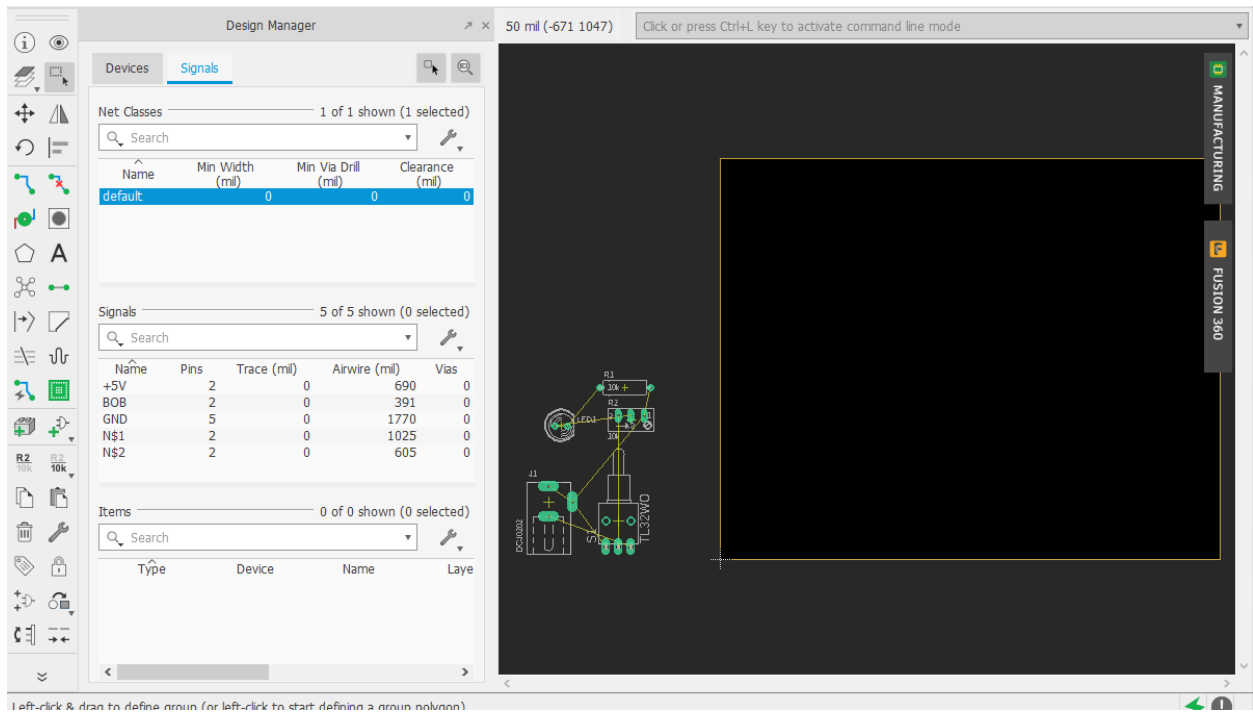


Figure 7

Size the Board

The default scale is metric. We will work in inches for this design. To change the scale, use the expansion arrow on the grid icon. Turn on the display, change the scale to inches, adjust the grid size to 0.05 inches and the Alt to .005 inches.

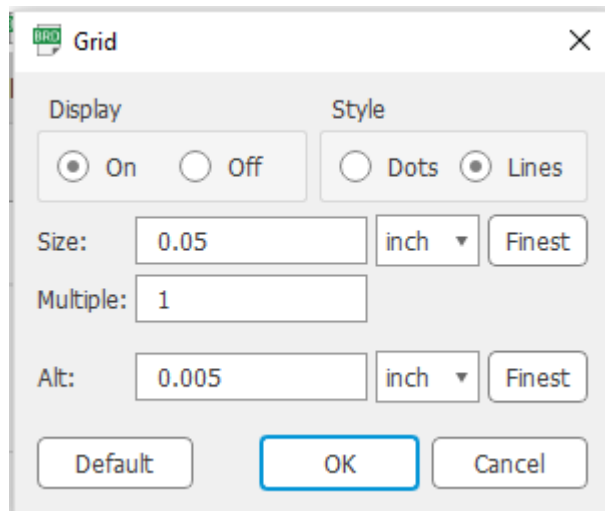
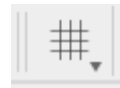


Figure 8

The default board size is bigger than we need. Adjust the size to 1.5" by 1" with the Move tool. Notice the upper left corner of the board display provides the coordinates of the cursor with respect to the lower cross hair index in the lower left corner. Be sure to keep your boards lower left corner referenced to this point.

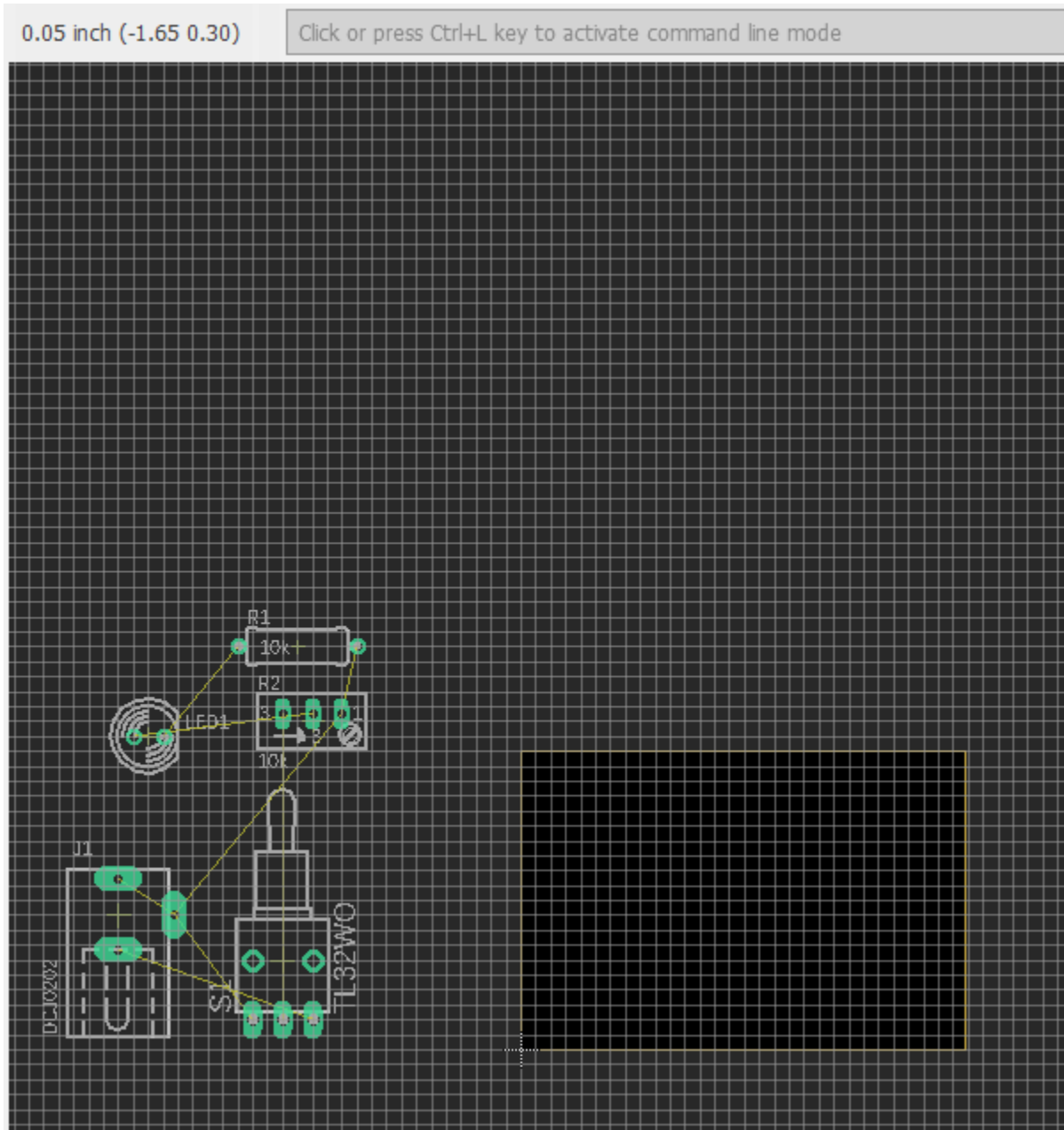


Figure 9

Locate the Components

The location of your component will depend on your application and component needs. In this case the only concern is to locate the switch and power jack along

the perimeter of the board. Using the Move tool, locate the components similar to the following figure.

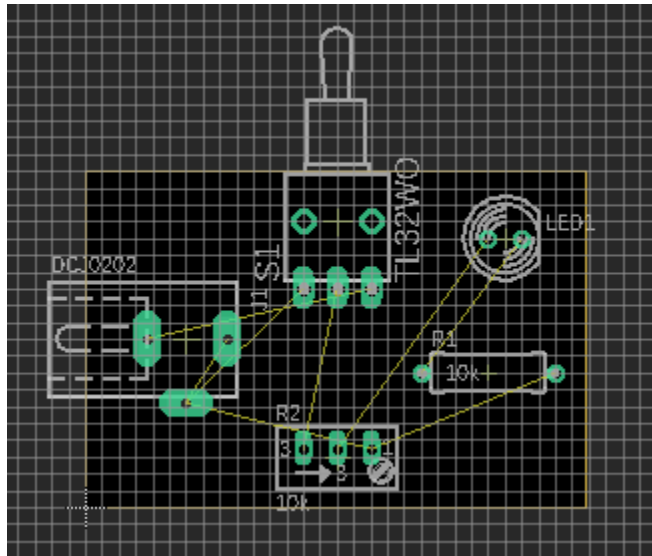


Figure 10

Layers

The printed circuit board description and implementation can consist of multiple layers. The simplest board can be just one layer providing the connections between the components. This board is simple enough that we could do that. Boards typical have at least a bottom and top connection layer that will make routing much easier. They also have a layer of material that protects the copper connections and prevents inadvertent solder shorts. There is also layer that can be used to label parts and provide component outlines facilitating board assembly. Another layer provides mechanical information about the board size and other features such as mounting hole locations and size. Here is a typical list of the layers with a brief description.

- Top – The top signal routing layer.
- Bottom – The bottom signal routing layer.
- Pads – Connections to components. These go through all physical layers.
- Vias – Holes to connect different layers. Discussed later.
- Unrouted – Shows the wires that have not been set to the top and bottom layers yet.
- Dimension – The board outline.
- tPlace/bPlace – Top and bottom layers for silkscreen information.

- tOrigins/bOrigins – Crosshairs allowing interaction with components.
- tNames/bNames – Shows names corresponding to components.
- tValues/bValues – Shows values corresponding to components.
- tStop/bStop – Solder resist layers.

Mounting Holes

We will provide some mounting holes for our design. Before we layout the circuit connections, we need to locate and size the holes. We will use the hole tool to do this. Select the hole tool. You will see a Drill size window open on the same tool bar as the grid tool.

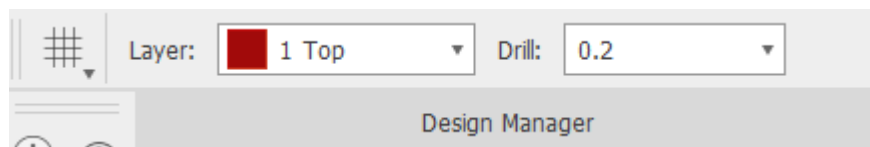


Figure 11

We are going to use a #8 screw to mount our board. A number 8 screw had a diameter of 0.16 inches. To provide a little extra room, we will specify the diameter to be 0.2 inches. You may need to manually enter this in the Drill size window. You should see a symbol that looks something like this.



Figure 12

The center circle is the drill hole. The outer circle represents the screw head diameter. This will depend a bit on the type of screw, but in our case we can assume it is a good approximation. You use this approximation to provide space around your other components. Place these in the four corners adjusting your other components for clearance. You should end up with something like this.

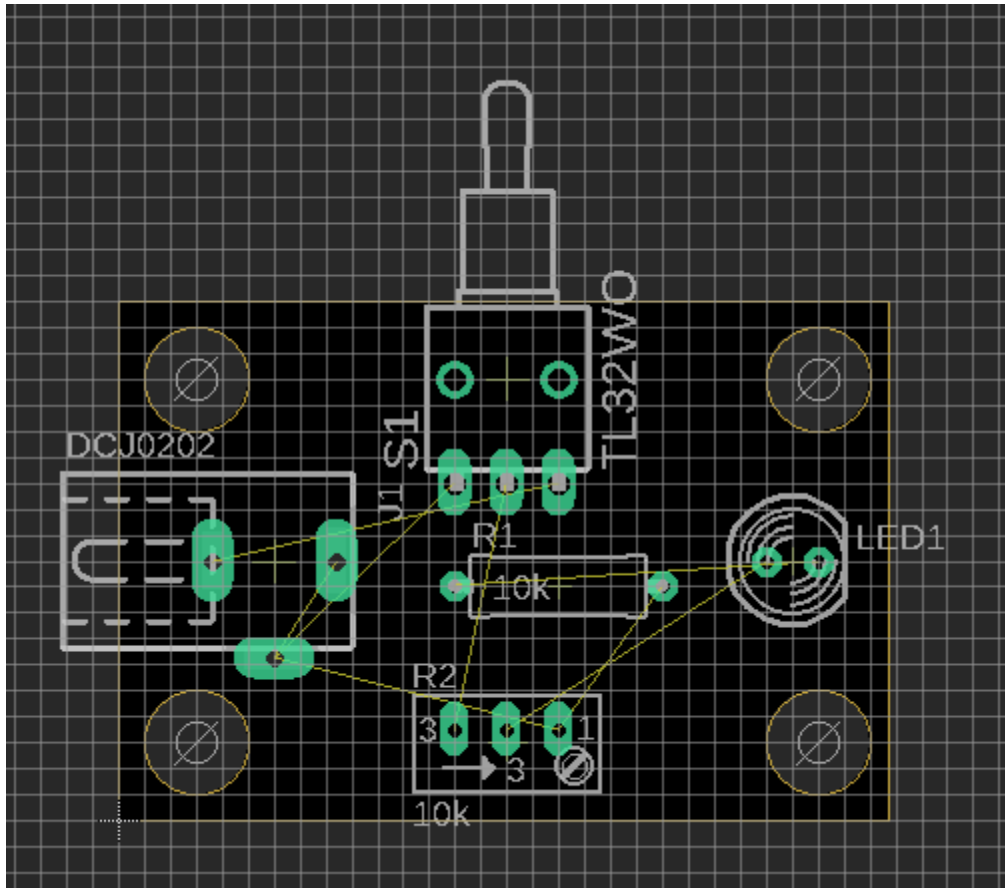


Figure 13

Routing the Connections

We are not ready to layout the connections. You can do this manually like running wires in schematic entry or use an auto-route function. Typically, the auto-routing function is used in conflicts that the auto-router can't resolve and is done manually. You can also manually make other adjustments that you know will be needed because of other circuit demands.

Let's start with the auto-router by selecting the tool shown at the left. The following window will appear. Use the default settings and continue and acknowledge the following windows.



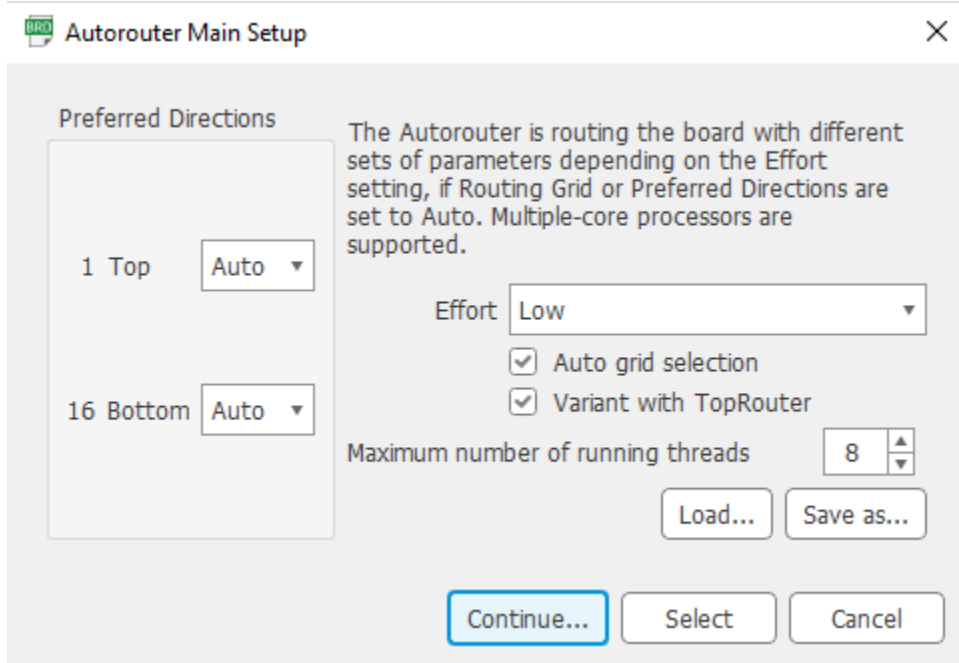


Figure 14

The completed routing will look something like the following, but can be different based on your component layout.

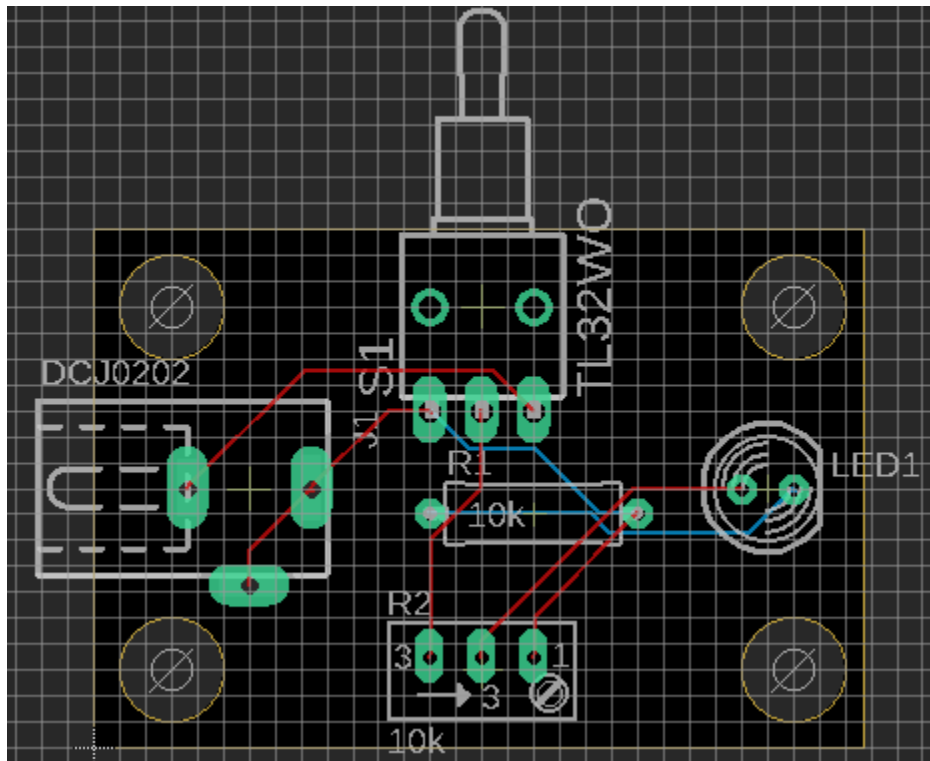


Figure 15

Notice that the router is using red and blue wires. The red wire on the top layer and the blue wires are on the bottom later. The top and bottom connections are joined were necessary at the component through holes. There are enough in this design to accommodate the connections. In more complex designs or designs that do not use through hole components (surface mount components), it will likely be necessary to use a via. It is a hole with a copper lining to provide a connection between the bottom and top layers. The via tool is used for this purpose and shown on just to the left.



Labeling

We can now inspect the labeling and assign them to the appropriate layers. First we want to make sure we are using “vector fonts” and “persistent drawing”. This is set under options→User interface.

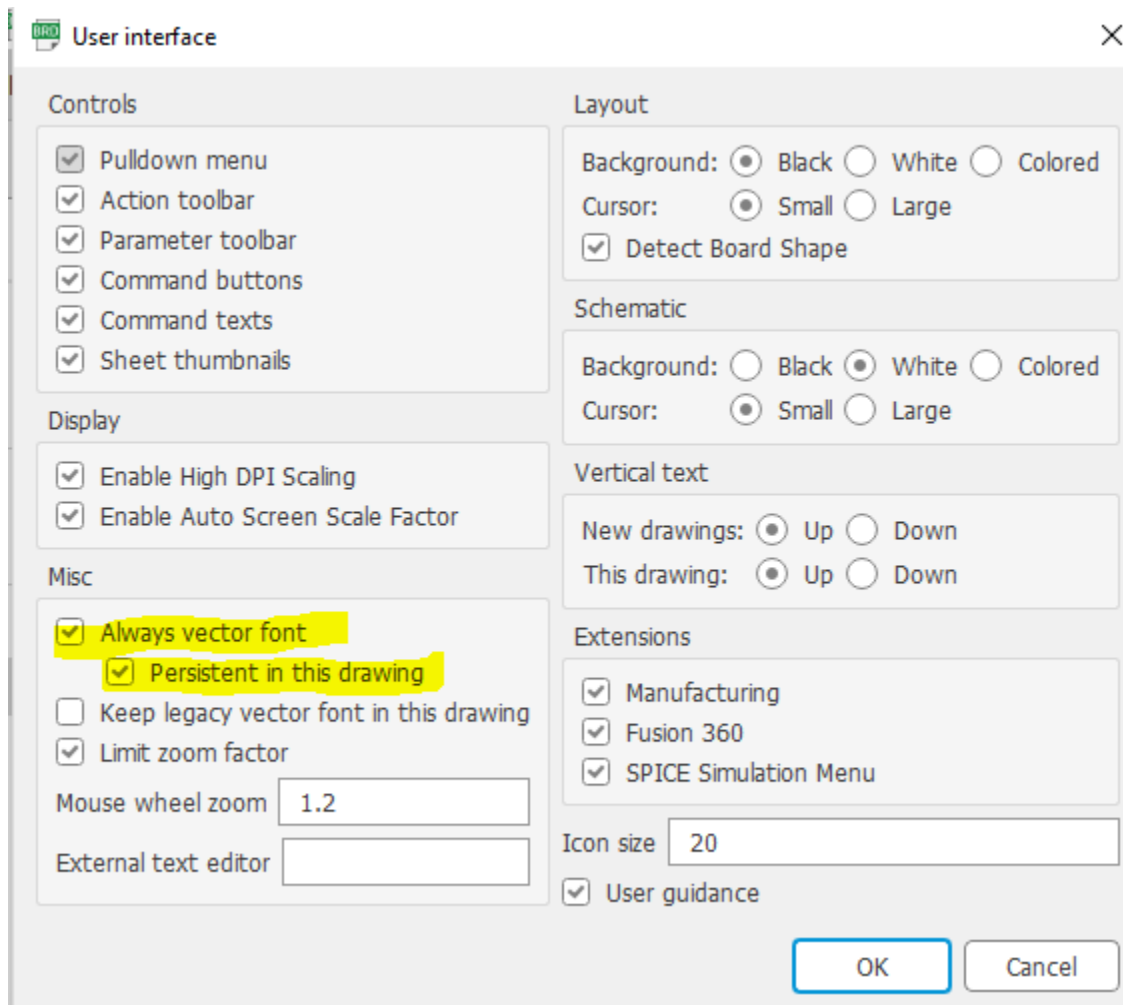


Figure 16

The labeling is provided in several layers: tPlace, bPlace, tNames, bNames, tValues and bValues. Lets look at what labeling is currently be used in our design with the “layering tool” shown at the left. You will see something like this.



Eye Icon	#	Name
<input type="radio"/>	17	Pads
<input type="radio"/>	18	Vias
<input type="radio"/>	19	Unrouted
<input type="radio"/>	20	Dimension
<input type="radio"/>	21	tPlace
<input type="radio"/>	22	bPlace
<input type="radio"/>	23	tOrigins
<input type="radio"/>	24	bOrigins
<input type="radio"/>	25	tNames
<input type="radio"/>	26	bNames
<input type="radio"/>	27	tValues
<input type="radio"/>	28	bValues
<input type="radio"/>	29	tStop
<input type="radio"/>	30	bStop
<input type="radio"/>	31	tCream
<input type="radio"/>	32	bCream
<input type="radio"/>	33	tFinish
<input type="radio"/>	34	bFinish

Figure 17

Determine the layers that are currently providing layers by the eye just to the left of layer names on an off. You should find that the tPlace, tNames and tValues are providing labeling for component outline, name and value on the top layers. It typically sufficient to provide the outline and schematic part reference (ie J1). So we are interested in layers tPlace and tName. You can change the label locations by grabbing the label cross hair with the Move tool. You can also rotate the label by right clicking a selected label. You may also want to add label to the board. Use the text tool shown to the left. A text box will appear. Enter your text and click OK. Select what layer you want it in with the layer window upper left. Select the size with the Size window just to the right of the Layer window.



Figure 18

The final version will look something like this with the “standard layers” enable in the “layering tool”.

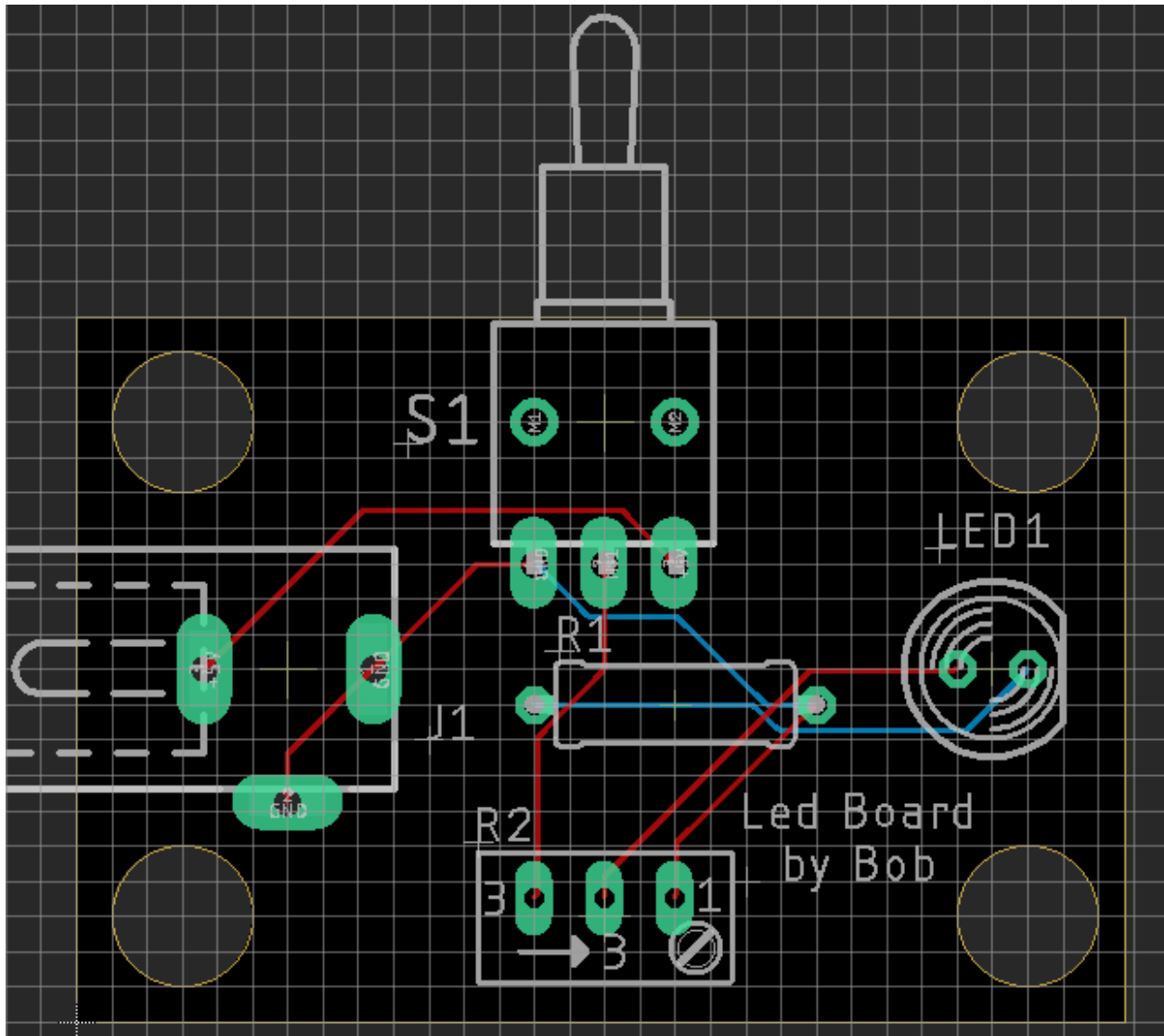


Figure 19

Errors and Design Rule Check

Eagle provides a tool to check your board against a set of design rules based on standard clearances and other parameters. First use the errors tool shown at the left. Hopefully you will not have any errors as shown here.



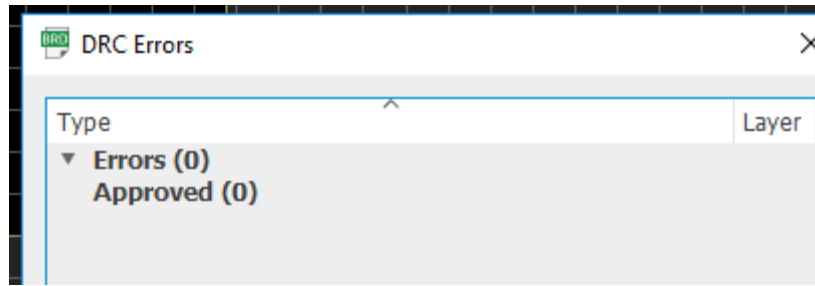


Figure 20

If not, you can click on each error and the tool will high light the error in the design canvas. You may have to research the errors on line if the description is not obvious. Next run the ERC shown to the left. In this case the following report was generated.

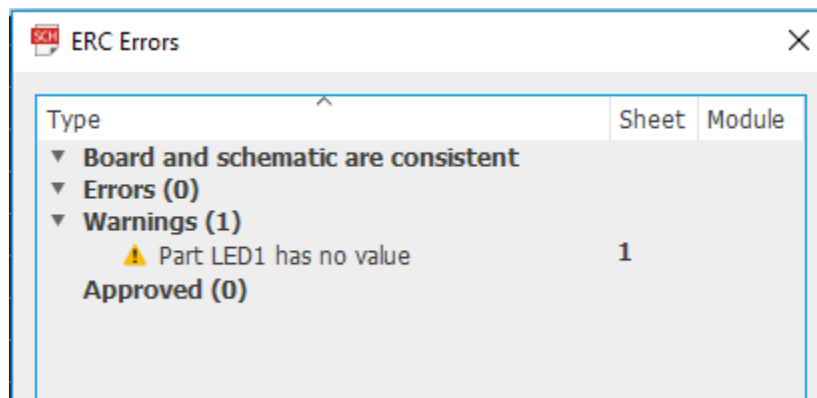


Figure 21

We are not concerned about a value for the LED so this check is fine.

Cam Processing and Gerber Files

We are finally ready to generate the files needed by most PSB houses. These consist of a set of files describing the connections, board labels, solder masks and a set of drill files used to describe the locations of the drill holes and their size. The first set of files are known as the Gerber files and the drill files are known as drill files or Excellon files for the CNC machine that uses them. The Gerber files consists of files with the listed extensions.

File Extension

.cmp
.sol
.stc

PCB Layer

Top Copper
Bottom Copper
Top Soldermask

.sts
.plc
.pls

Bottom Soldermask
Top Silkscreen
Bottom Silkscreen

The files typically come in two formats: Gerber RS-274D and Gerber RS-274X. The latest standard “X” contains all the information about a layer in a single file.

Creating the Gerber Files

Open the .brd file via the Control Panel. Open the CAM job listing in the Control Panel→CAM jobs→Legacy→gerb274x.cam. You should see something like this.

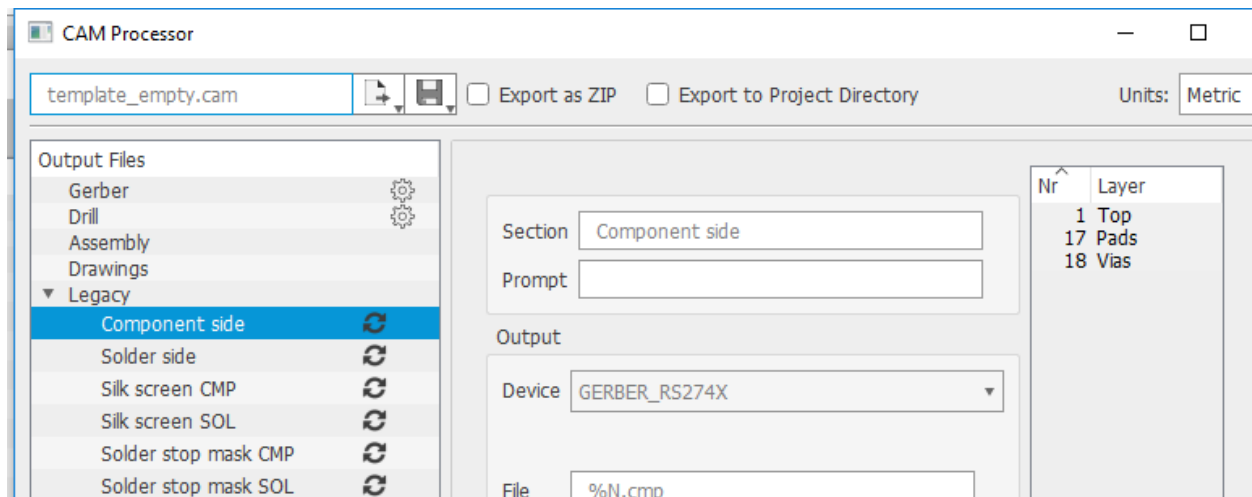


Figure 22

In the bottom right corner select the .brd file we have been working on. Process the job. You will be prompted for location to put the files. Use the default or create a new directory. When the job is down you should see something like this in the directory.

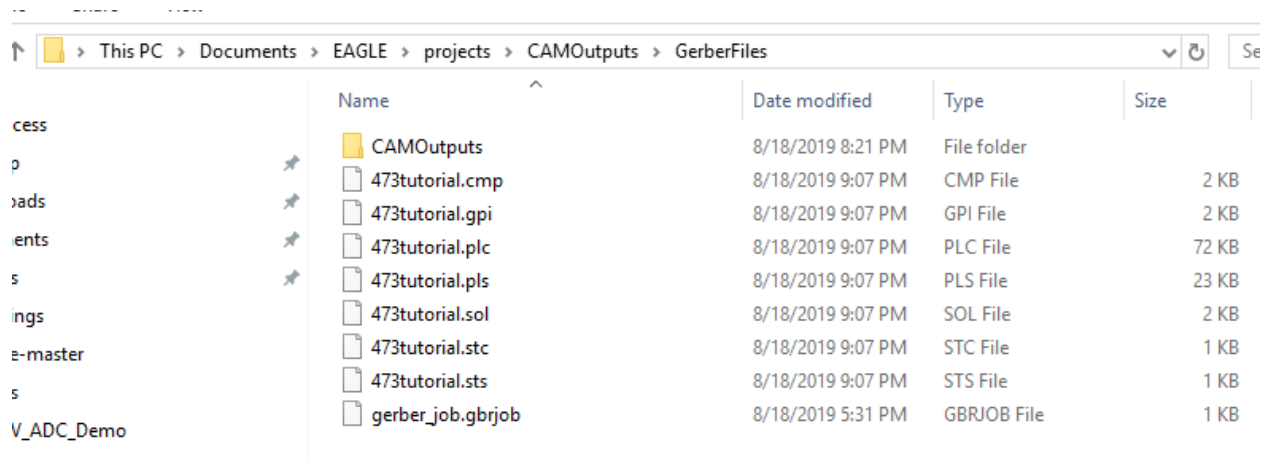
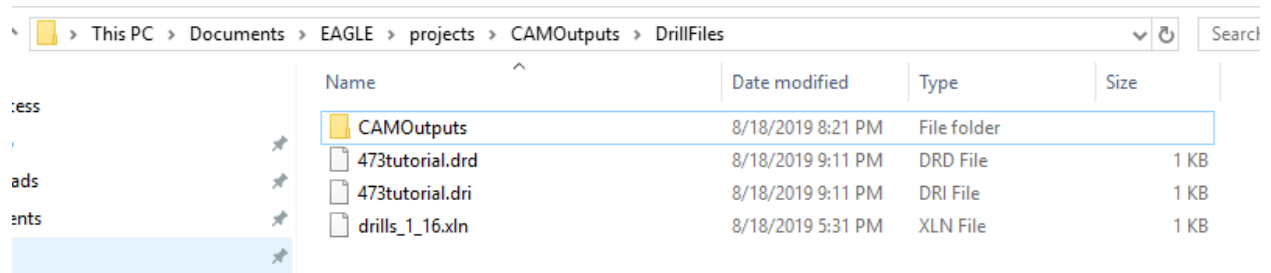


Figure 23

Close out the job. Next select the excellon.cam job in the Legacy cam job list. Select the .brd file as before. Process the job and select a different directory then you used before. When the job is done you should see the following files.



You now have all the files you need to provide a PCB house.

Alternate References

Auto CAD has an excellent tutorial and general reference at <https://www.autodesk.com/products/eagle/learning-center>. This is generally up to date and offers more detail then we have provided here.