



BOISE STATE UNIVERSITY
ENVIRONMENTAL HEALTH, SAFETY
AND SUSTAINABILITY

STANDARD OPERATING PROCEDURE

VOLTERA ONE PCB PROTOTYPE SYSTEM

College/Dept: College of Engineering


Building/Room: RUCH106

Laboratory Name: Idaho Microfabrication Lab

Revision: A

Revision	Details	Date	By
A	Updating original document format	9/14/21	P. Miranda

Approval

Authored by: _____ Name, Title	Pete Miranda, IML Director	 Signature	091421 Date
Reviewed and Approved by: _____ Name, Title		_____ Signature	_____ Date

Procedure Type

Process/Protocol
 Chemical
 Hazard Class

Brief Overview

The Voltera One PCB Prototyping system is designed to provide the user with a quick and easy method for producing PCBs for many different types of applications including R&D, academics, hobby, etc. The system is designed to print conductive traces, dispense solder paste onto printed pads and after components are placed by hand, to reflow the solder paste for component attachment!

Scope

This document covers basic operation of the Voltera used in a typical fashion with commonly printed silver conductive traces and eutectic solders such as Sn60/Pb40 or similar.

Potential Hazards

Chemical
 Thermal
 Hydraulic
 Electrical
 Slip/Trip
 Biological
 Mechanical
 Radiation
 Pneumatic
 Fire
 Fall
 Other

Hazard Specifics: Heated board platform, moving dispense head, solder paste handling hazards, conductive silver handling hazards

Engineering Controls (EC)

Fume hood Biosafety Cabinet Other Local Exhaust Shielding Other

EC Specifics: < if necessary, identify type, location, flow requirements, material requirements, etc.>

Training Requirements – except for classroom lab safety, must be completed prior to performing the procedure

Classroom Laboratory Safety Awareness Radiation Worker
 Online Safety Topics
 (specify):
 Lab/Work Group Specific Training IML Safety Training
 (specify):
 Other (specify):

Safety glasses Safety goggles Face shield & safety glasses
 Lab coat Apron Tyvek suit
 Gloves Leg coverings Hard hat
 Respirator Shoes Fall protection

PPE
 Description:

Eyewash Fire extinguisher First aid kit
 Safety shower Fire blanket Spill kit
 Drench hose

Reference: <https://www.voltera.io/support/quickstart>

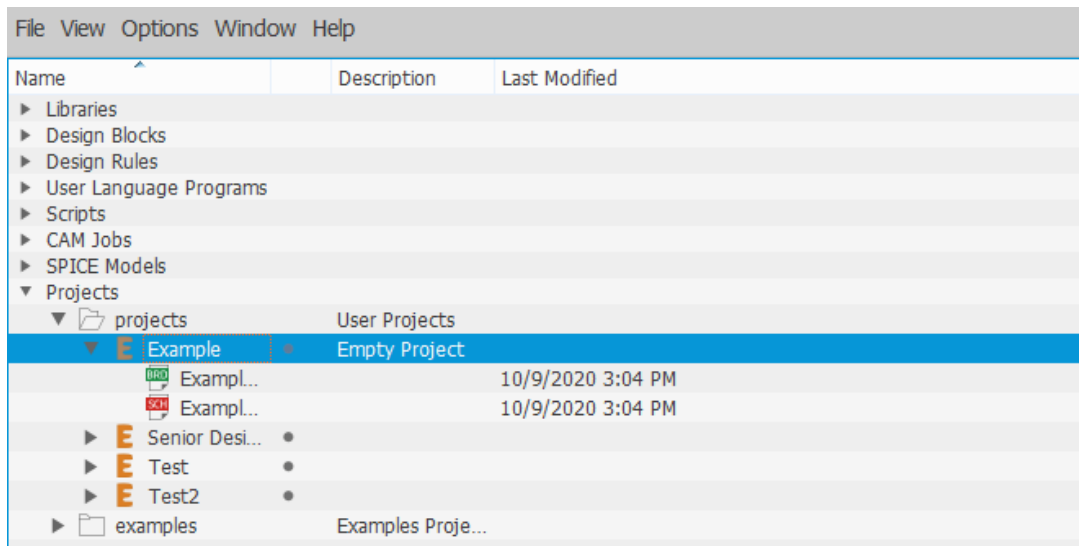
1. Draw Circuit

Note: There will be two references to the word board. If it is written in all lower case letters, it is referring to the part that is being placed into the Schematic file to help with dimensioning. If it is written as "Board" it is referring to the file.

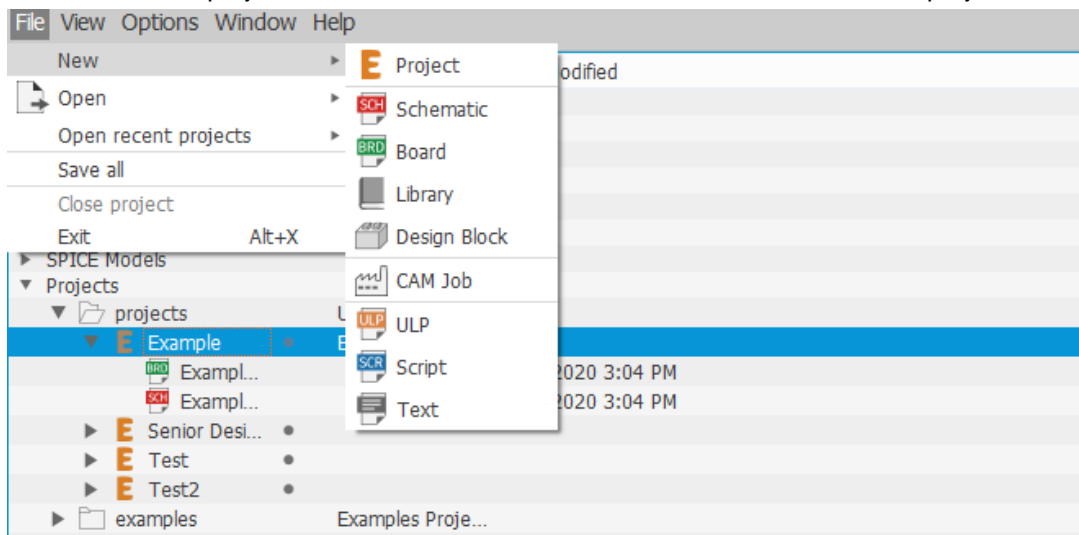
The first step in making a print is to draw the circuit. There are three software packages that Voltera recommends using: Altium, Eagle, or KiCad. The IML uses Eagle so below are the basics on using the Eagle software.

The circuit will be drawn in a Schematic and Board. These files are located under project folders. You can use the same project folder for multiple Schematic and Boards, but only one project folder can be open at a time and only one Schematic and Board can be open at a time. If you try to open another one it will close the file that is open.

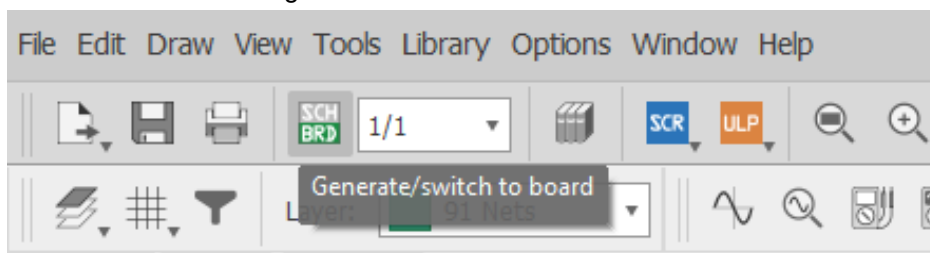
VOLTERA SOP



To start, make a project folder. Then make a Schematic and save it under the project folder.



To make a Board click 'generate switch/board'.

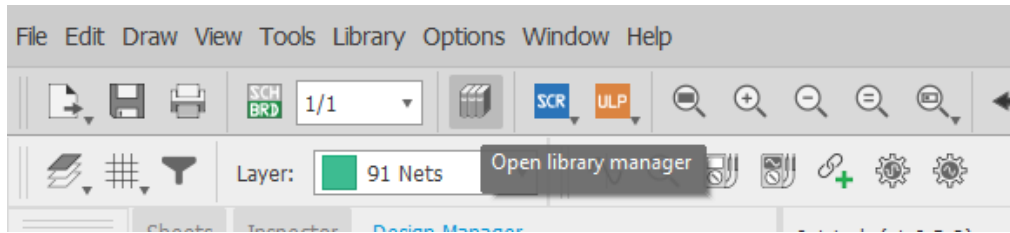


Both the Schematic and the Board must be open for them to sync. DO NOT work on one if the other is not open. The program will warn you if this error does occur.

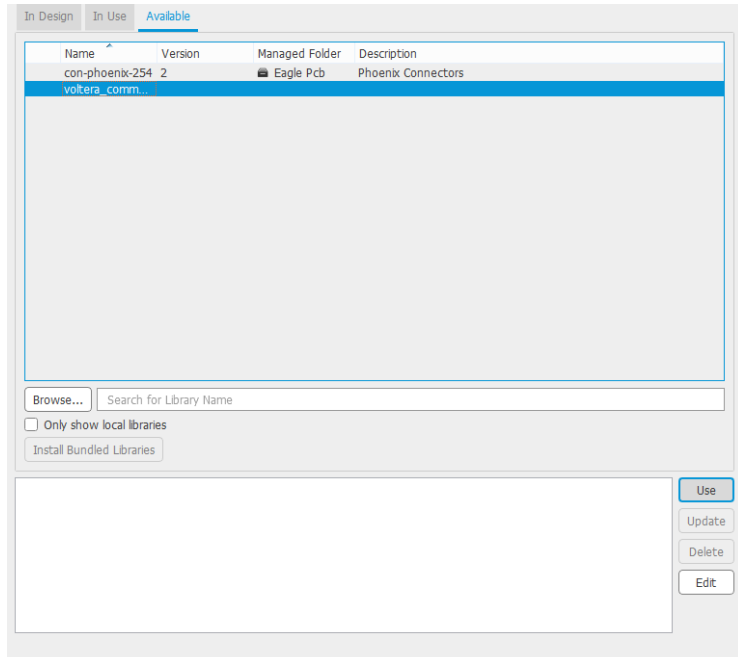
F/B Annotation has been severed!

VOLTERA SOP

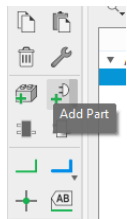
The first step is to put the substrate outline in. The program already has 2x3 inch board, 3x4 inch board, and 4x5 inch board for the PCB they sell. If the dimensions of your substrate do not match this you will need to make a new device (reference section below). To insert the board click 'open library manager'.



Go to the 'available' tab and click 'voltera_community' then 'use'.

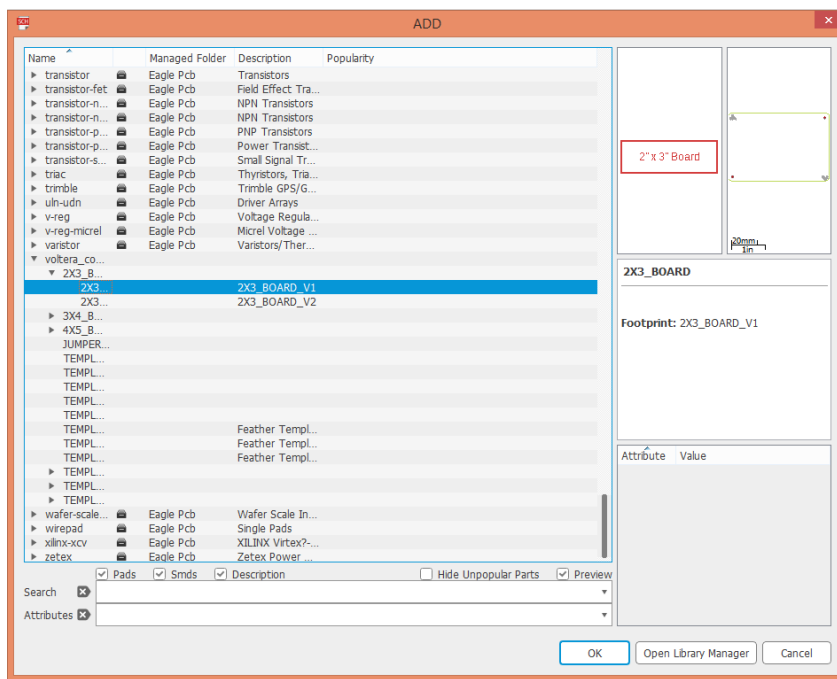


Next open 'add part'.

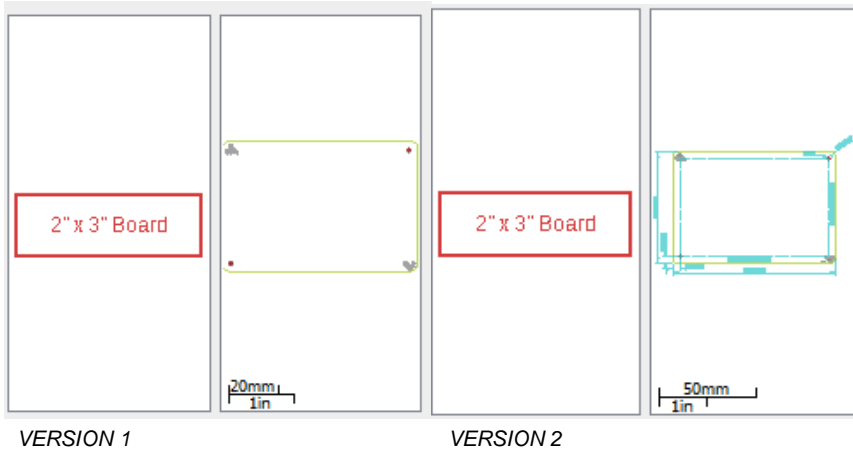


Scroll down to 'voltera_community' and select the board.

VOLTERA SOP

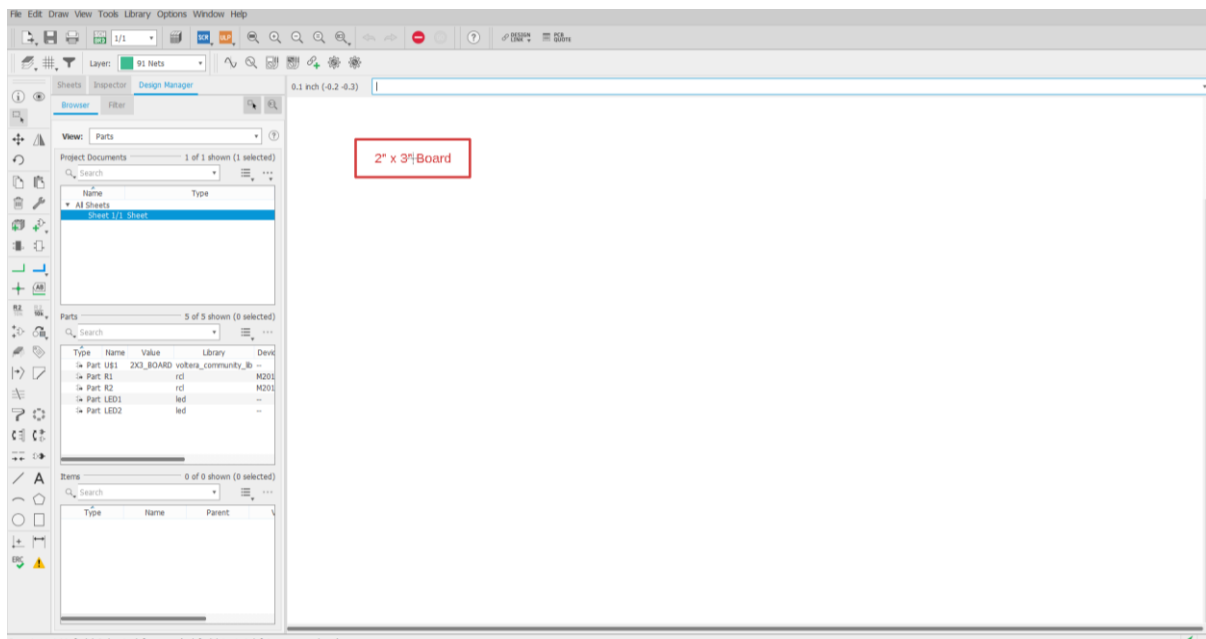


You can use version one or two depending on your preference. The footprint will vary in the Board.

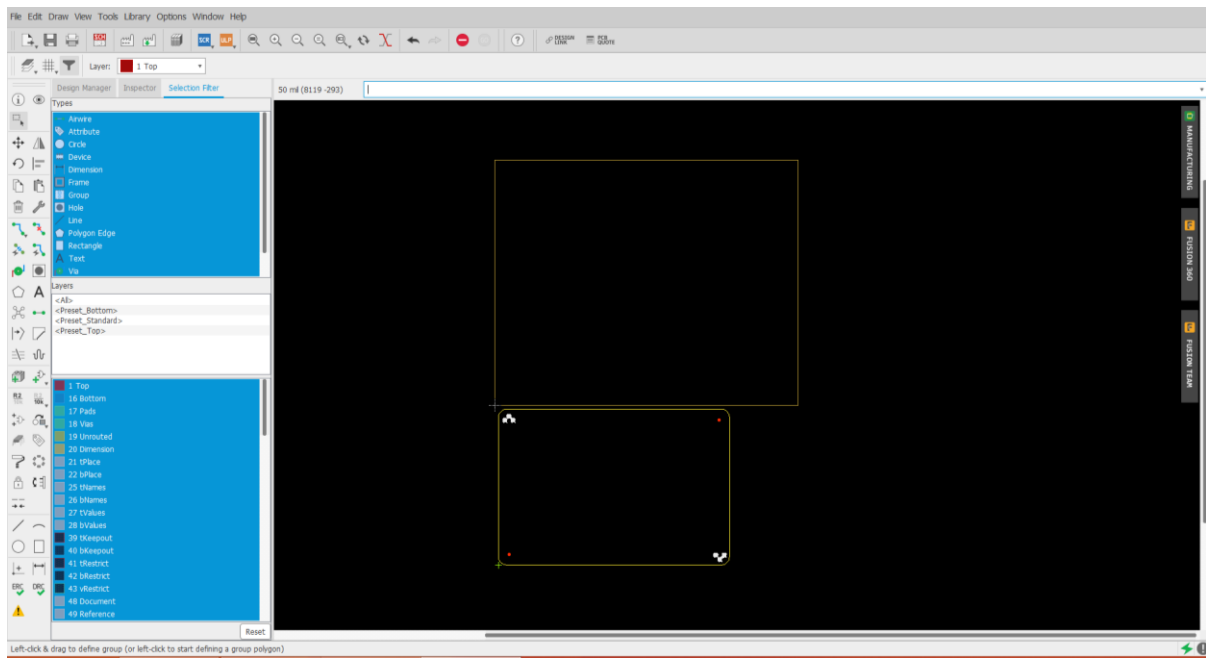


Place the board wherever in the Schematic. The placement of components in the schematic does not affect the Board. The Schematic is used to get an outline and the Board is where things need to be precisely placed. Once you have one board hit the 'esc' button and then press 'cancel'. If you open the Board you should see it is also in there.

VOLTERA SOP

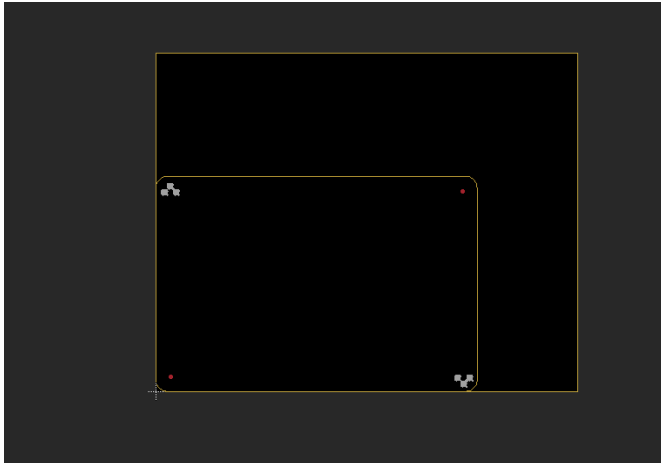


The 2 x 3 board in Schematic

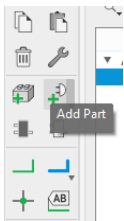


The 2 x 3 board in Board

The placement in Board is important. The corner of the board should align with the cross hairs. To move the board click and drag by its crosshairs. Release when it is in the right position. The outside box is the area that will be printed. You can change the size to match the template or just ensure you draw within the designated space.



If there is a component you would like to put in your circuit (i.e. resistor, LED, etc.), place it in like you placed the board, by clicking 'add part'.



The image below is helpful advice on how to optimize your search for components.

Search pattern can be one or more words, separated by blanks. These words are searched case insensitively in the device names and descriptions (if *Description* is checked), and must all match.

The wildcard character '*' matches any number of non-whitespace characters, while '?' matches exactly one of these characters.

If *Pads* is checked, devices that contain PADs will be included in the search.

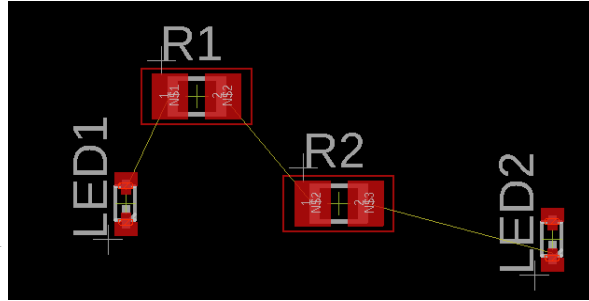
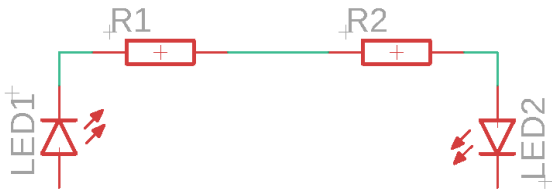
If *Smds* is checked, devices that contain SMDs will be included in the search.

If attribute search patterns 'name=value' (e.g.: tolerance=5%) are given, these patterns have to match additionally. An attribute search pattern without the character '=' is searched in the attribute names and values.

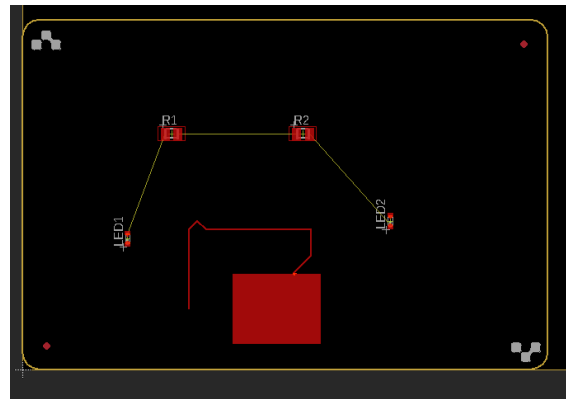
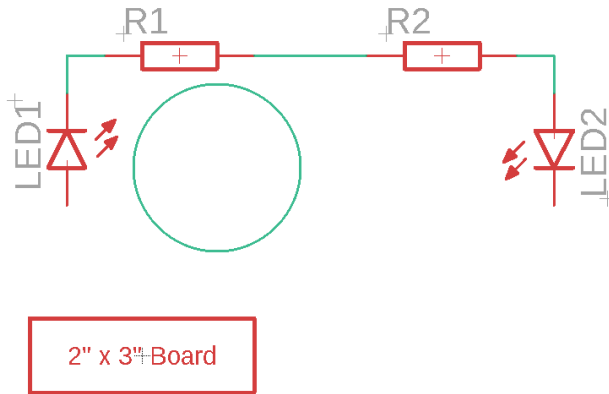
To find all NAND devices from the 74xx series, enter: 74* nand

VOLTERA SOP

When placing the component, you can right click to rotate. Again, their placements in Schematic will not affect their position in the Board, but connecting them with a line will automatically connect them the proper way in the board.



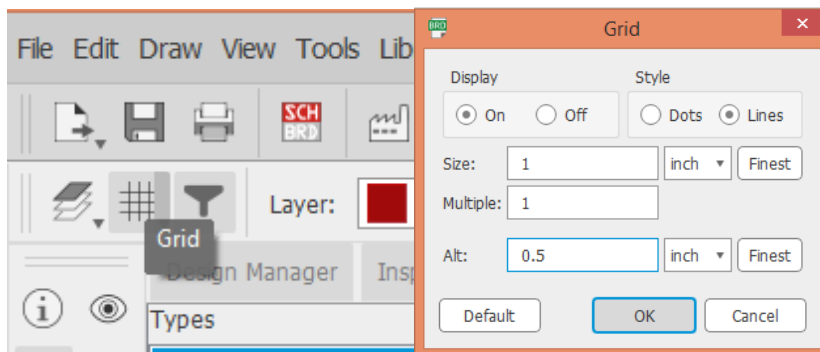
If you draw a shape or a line (not connecting components) in the Schematic it will NOT appear in the Board, but if you draw any shapes or lines on the board it will appear in the print.



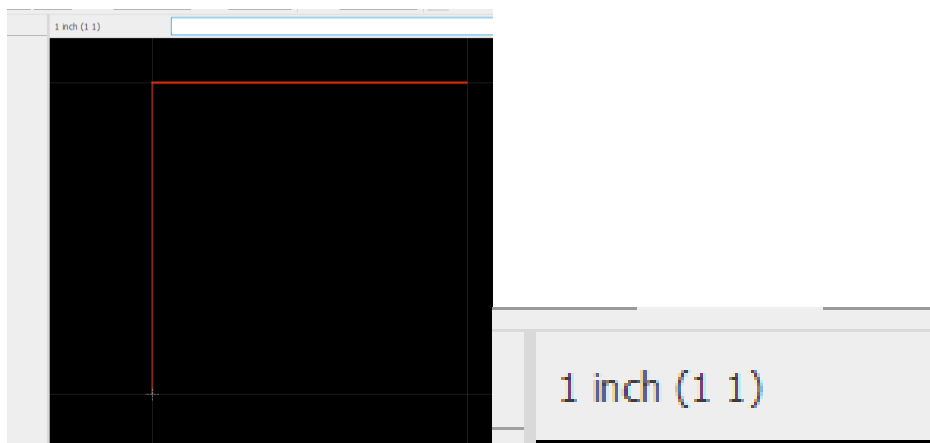
The circle in the Schematic won't appear in the Board

The square and line drawn in the Board will appear in the print

There is no way to dimension your circuit. The only way to precisely place components and draw is by using the grid. Click 'grid' and set it to the size and units that will allow you to achieve your dimensioning. Make sure you click "display" to see your grid lines. If the display is on and you can't see the grid, it is because it is too small. To see it - Zoom in.



As you move along the grid, in the left hand corner it will tell you where the cursor is within the grid. For example, the first line is one inch tall and the second is one inch wide.

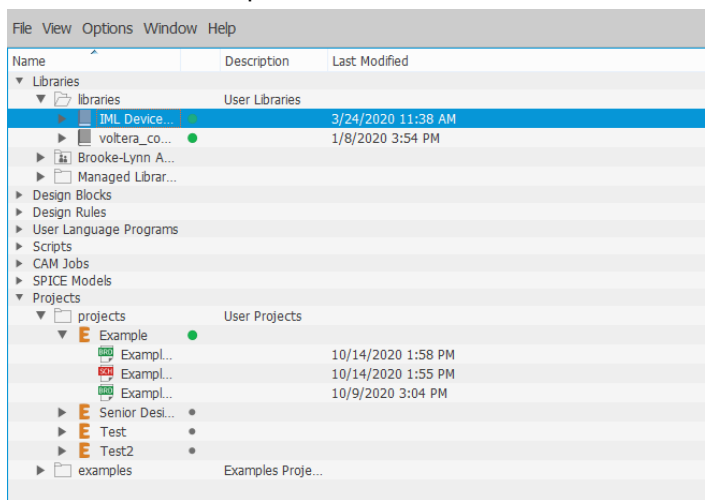


2. Making a Device

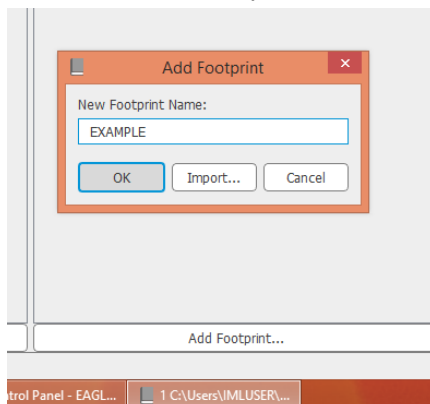
Note: If you do not need to create a device, skip to the next section.

A device can be made for components and boards (i.e. 2" x 3" Board). A device consists of a symbol and a footprint. The symbol is what will appear in the Schematic and the footprint is what appears within the Board.

To make a device, open 'IML Devices'.

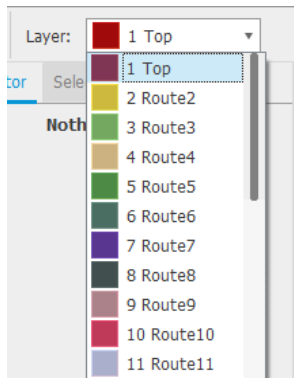


Next, click 'Add Footprint' and name it the same as what your device will be. Click 'ok'.

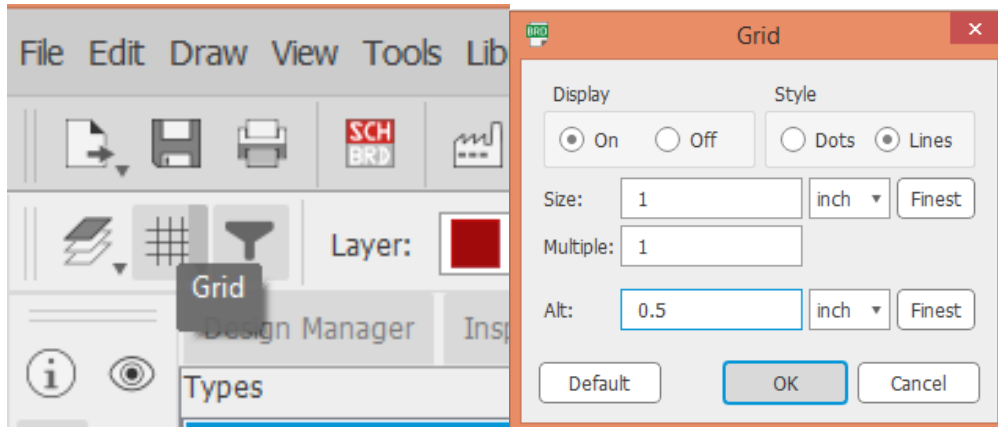


VOLTERA SOP

Before you begin to draw the footprint make sure the layer is set to '1 Top'.



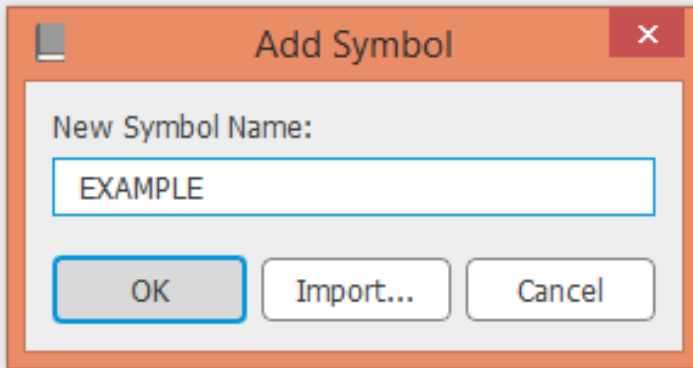
Turn on the grid to get accurate dimensions.



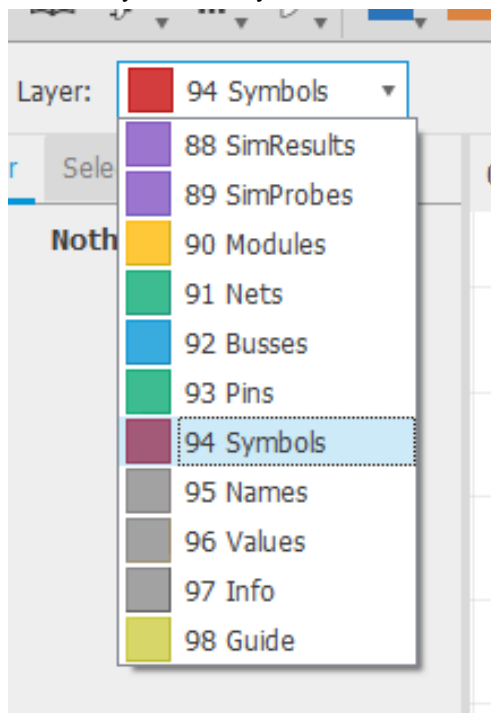
The grid is set to 1 inch. That means the box below is one inch by one inch.



The next step is to make a symbol. Click 'add symbol' and give it the same name as the footprint and click 'ok'

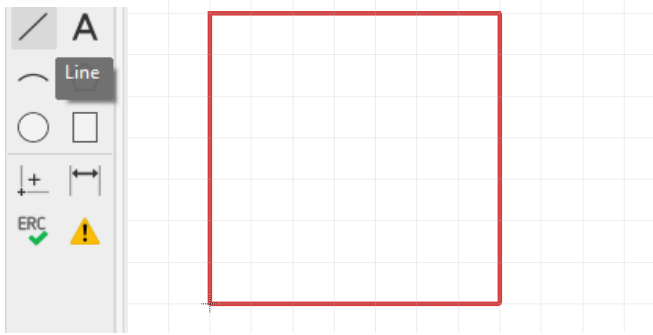


Set the layer to '94 Symbols'.

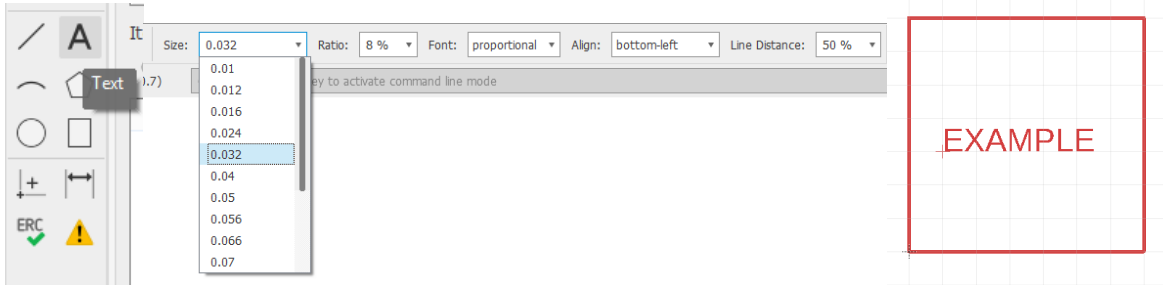


Create a box of any size using lines

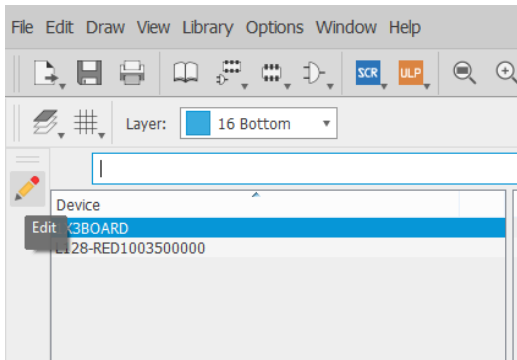
VOLTERA SOP



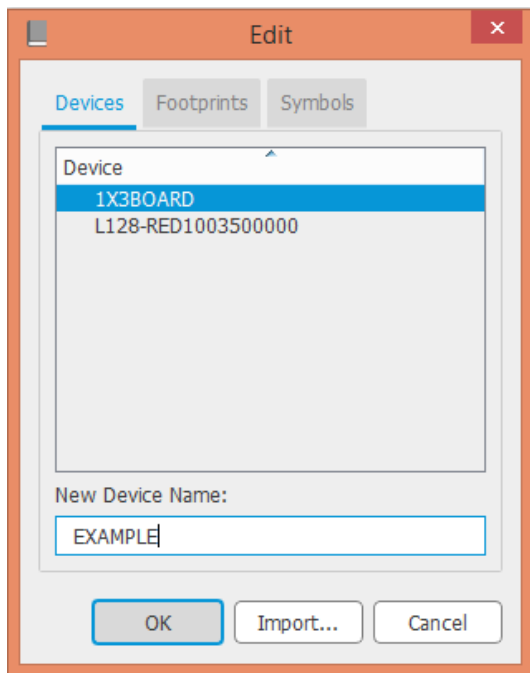
Add text to describe what the object is and make sure the font is legible by adjusting its size.



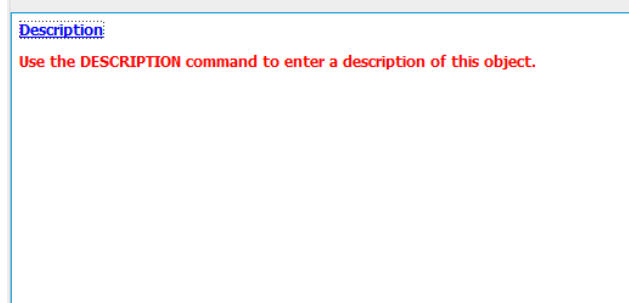
Once the footprint and symbol are made click 'edit' (the pencil).



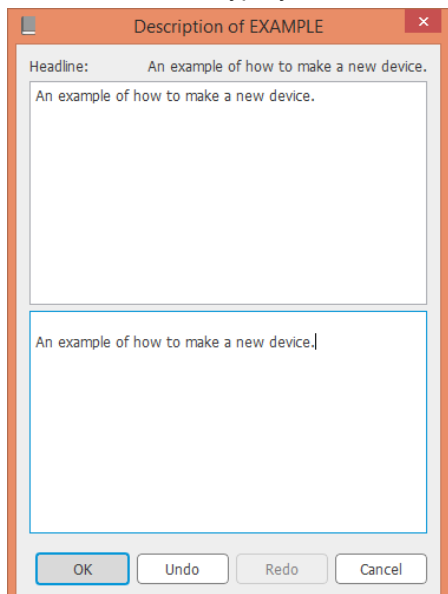
Type in the new device name and click 'ok'.



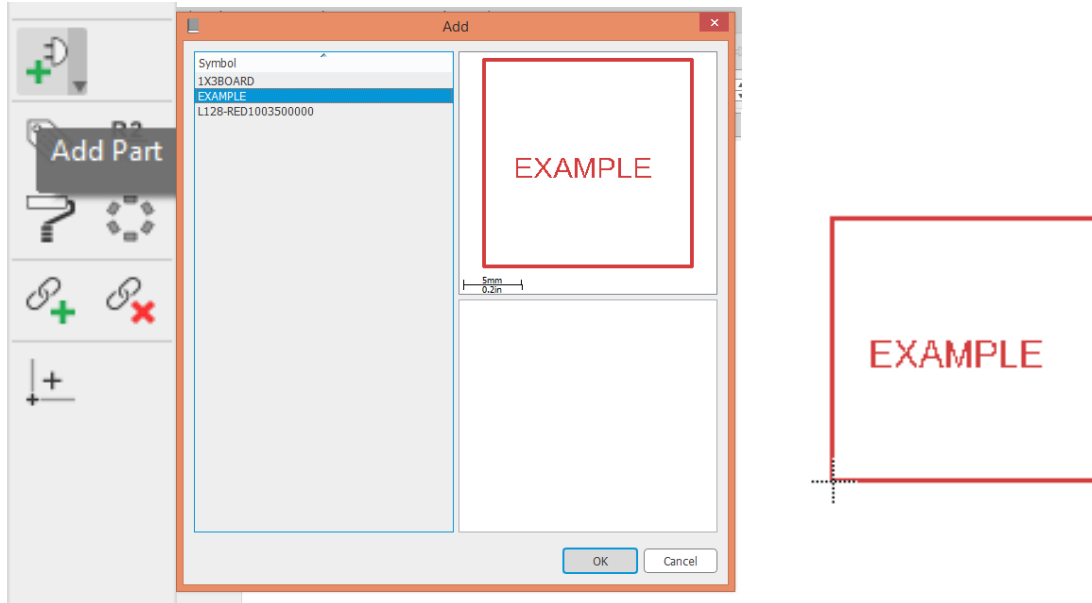
Click on 'Description' in the blue letters.



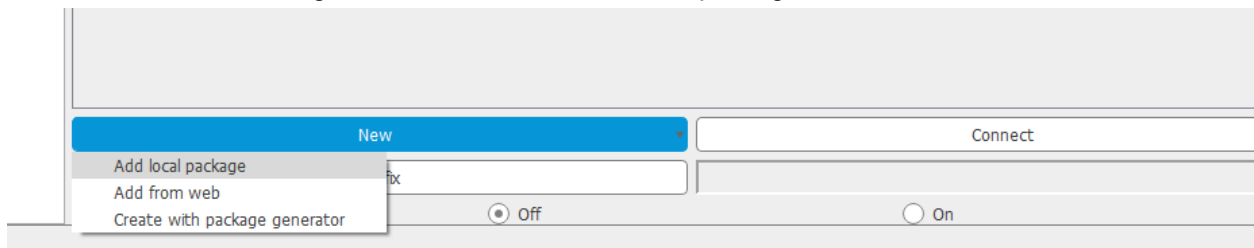
In the bottom box type your device description and click 'ok'.



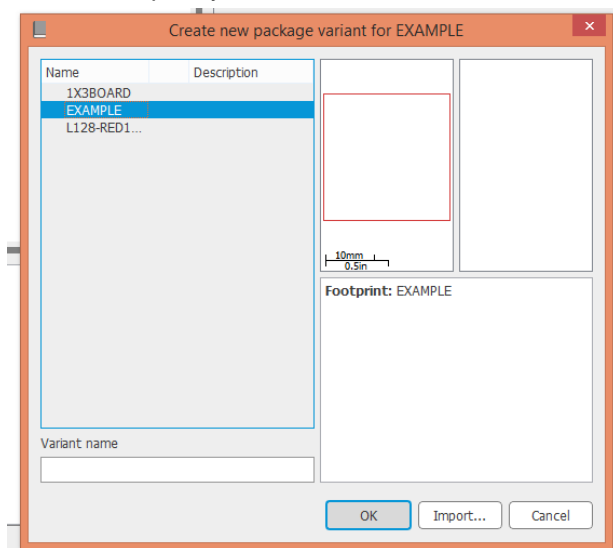
Click 'add part' and select your symbol. Align with crosshairs.



Go to new in the bottom right hand corner. Click 'add local package'.

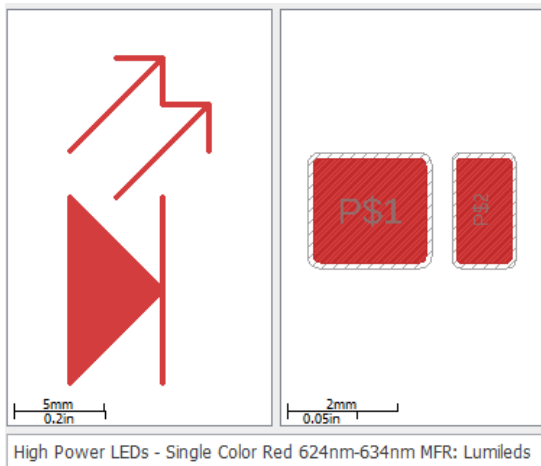


Add the footprint you created.



Once the symbol and footprint are added to the device it can be used in your designs. Devices aren't just for defining the print area; they can also be used for components. Although Eagle is preloaded with

several components, they may not have the one you need. If you know the product dimensions, you can make your own. **Most SMT (surface mount technology) components have a recommended pad design and can be found in the part datasheet.** Below is an example of a device created for a red LED.

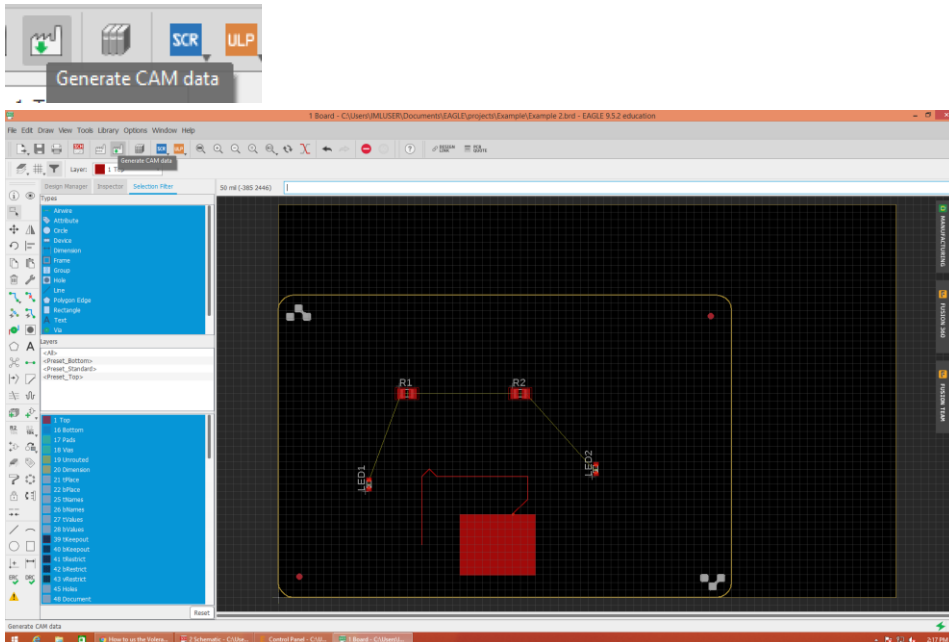


3. Exporting Gerber File

The file needs to be exported to a gerber file. Below are links to instructions on how to export with each software.

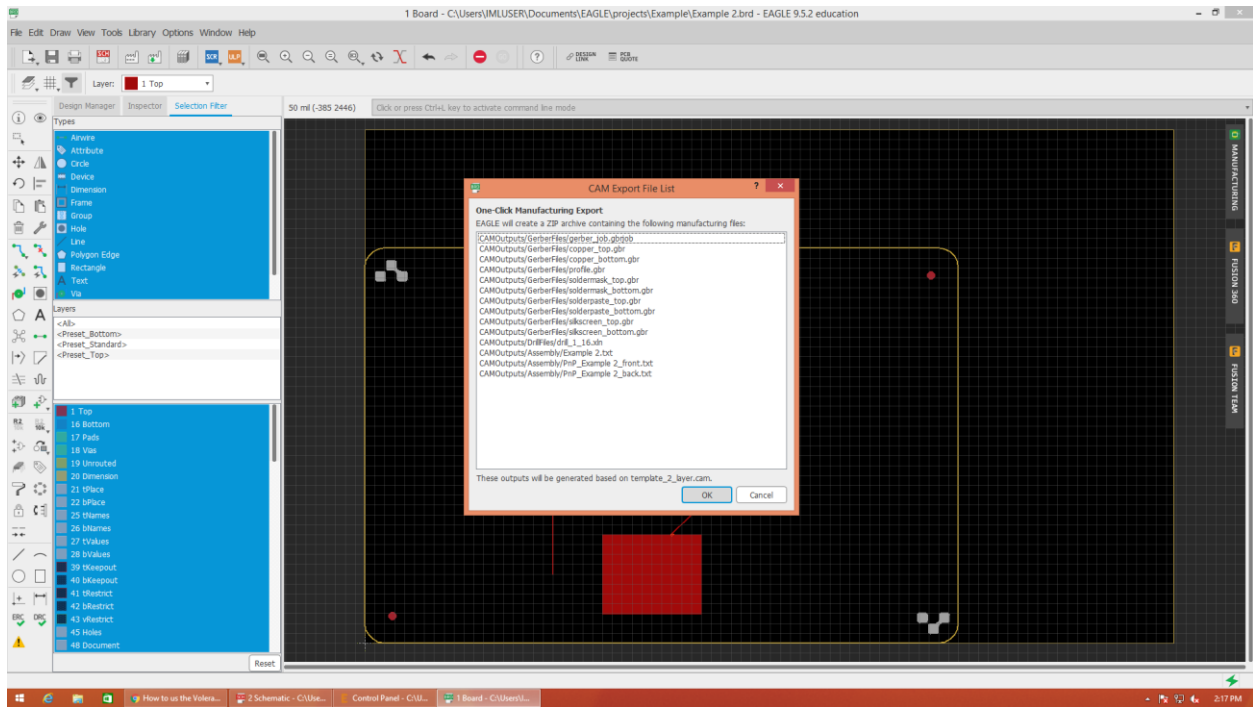
- Altium: <https://www.voltera.io/support/export/altium#main>
- Eagle: <https://www.voltera.io/support/export/eagle#main>
- KiCad: <https://www.voltera.io/support/export/kicad#main>

When the board layout is complete, the gerber files can be exported. Click 'generate CAM data'.

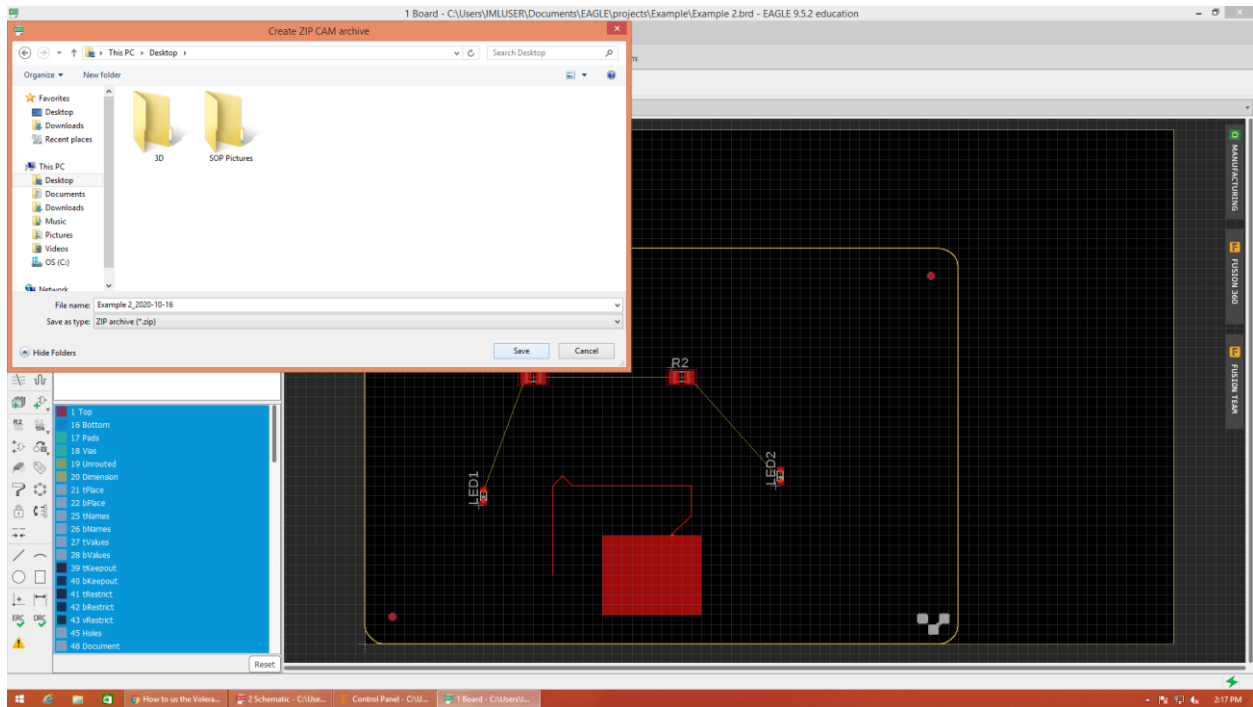


Then click 'ok'.

VOLTERA SOP



Choose file location and click 'save'.

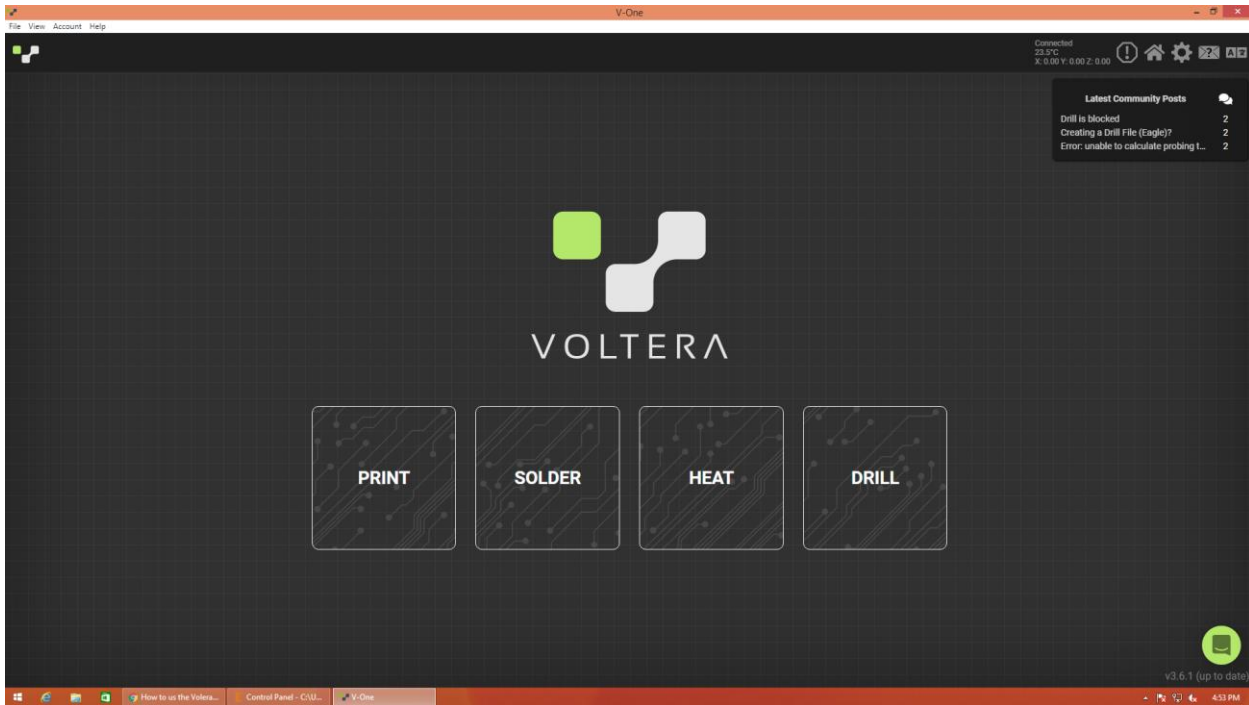


The file will be saved as a zip. Make sure to extract before going to Voltera software. Once you have exported there will be several files. The one titled "copper_top.gbr" will be the pattern for the conductor paste. The one labeled "solderpaste_top.gbr" will be for the solder paste.

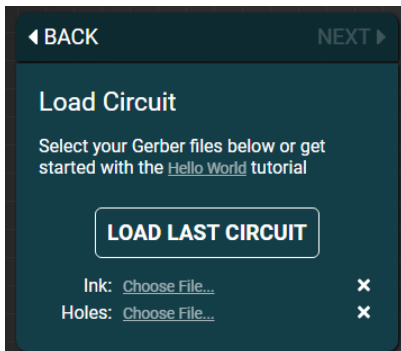
4. USING THE VOLTERA

Take ink out of cold storage 30 minutes prior to use.

Make sure the printer is on before you open the Voltera application. The screen will prompt you with four options. Click “print”

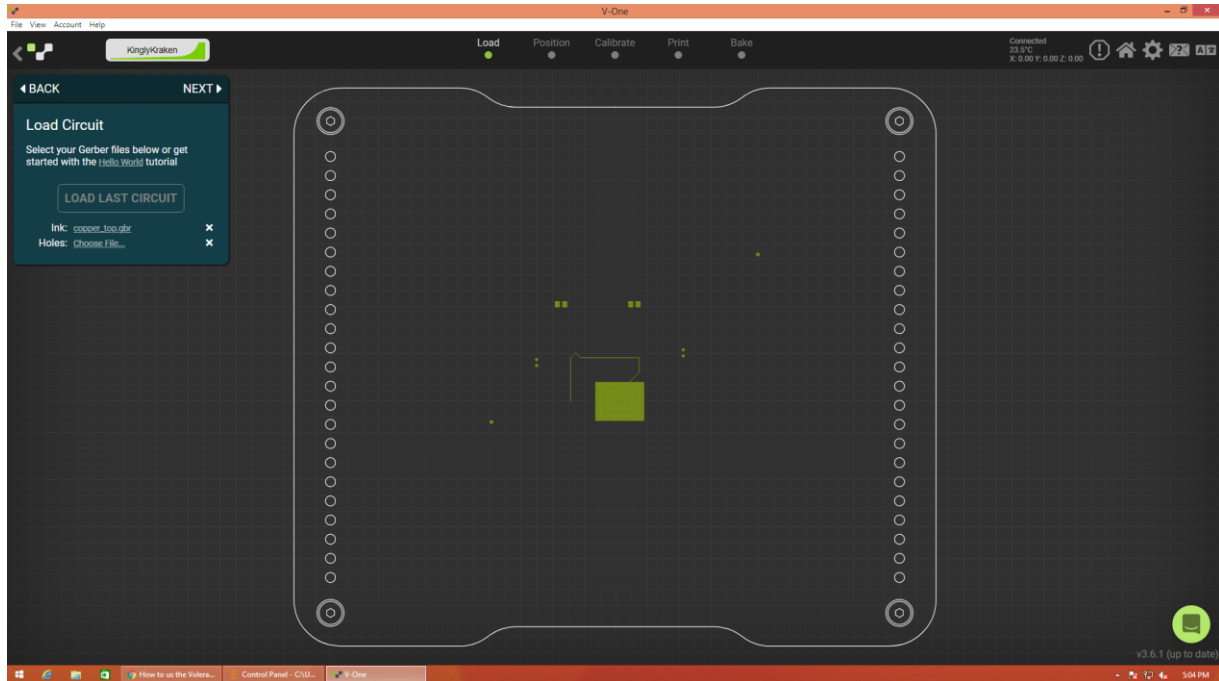


Load the file by clicking “choose file” next to ink.



VOLTERA SOP

Your circuit will appear as it will be printed. If anything looks wrong, now is the time to go back to Eagle and fix it (and export again). If your circuit is correct click “next” and the software will walk you through the rest of the process.



There are three nozzles you can use, the orange is 100 micron, the blue is 150 micron, and the black is 225.

After you have completed your printing, you will need to clean up your area and dispose of any waste as instructed by lab procedures.