

# Eagle manual (basics)

## In this manual you will find:

A basic tutorial on how to work with Eagle to make the LedUp Kids developed at De Creatieve STEM.

In this tutorial you will learn:

- finding your components in the libraries
- adding them to a schematic
- wiring schematics
- supplies
- drawing the board
- wiring the board
- giving the components a value and name
- adjusting the board according to the purpose (vinyl, etching, milling)
- 

## 1. Figure out what your board needs to do

As with everything, this is the most important question: what do you want your board to do? Is it just a LED blinking? A sensor collecting data and showing it to you? Or something much more complex?

In the case of the LedUpKidz we want children to be able to program their pcb and learn about smd soldering and programming. For that, we keep it basic, some LED's, switch, ATtiny85. No special components.

## 2. Make your BOM

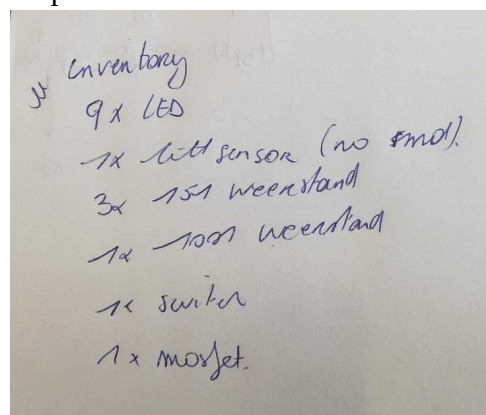
Almost as important as the first step is to make your bill of materials (BOM). This makes it easy to order components, ask someone about components but also use it as a checklist to see if you have added everything to the board.

You need to see the BOM as a communication. If you walk up to someone in a FabLab and ask for a Attiny micro-controller they will ask you wich one.

You'll need to be able to say Attiny44, Attiny45, Attiny85 or even any other Attiny that's out there in the libraries and stock.

For the LedUpKidz V1 we'll need:

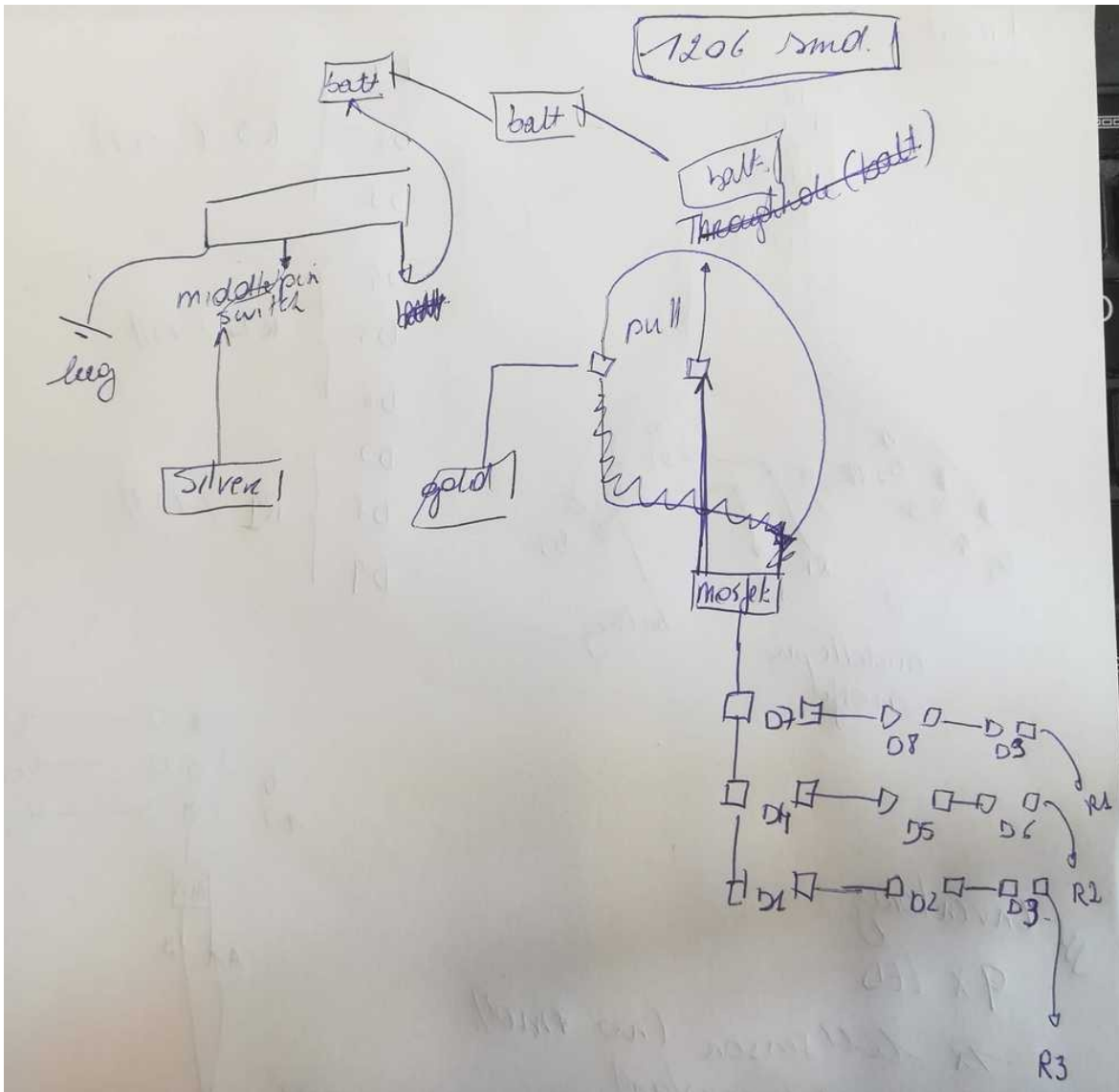
- 1x Attiny85-20SU (1Mhz clock)
- 1x Linx Coin Cell Battery holder
- 1x C&K Slide Switches
- 6x RED – BIVAR standard LED smd 0603
- 6x Yageo Thick Film resistors smd 330 ohm
- 1x pcb board



In this case you won't find the components in the FabLib for eagle, but you'll need to download and add the sparkfun library.

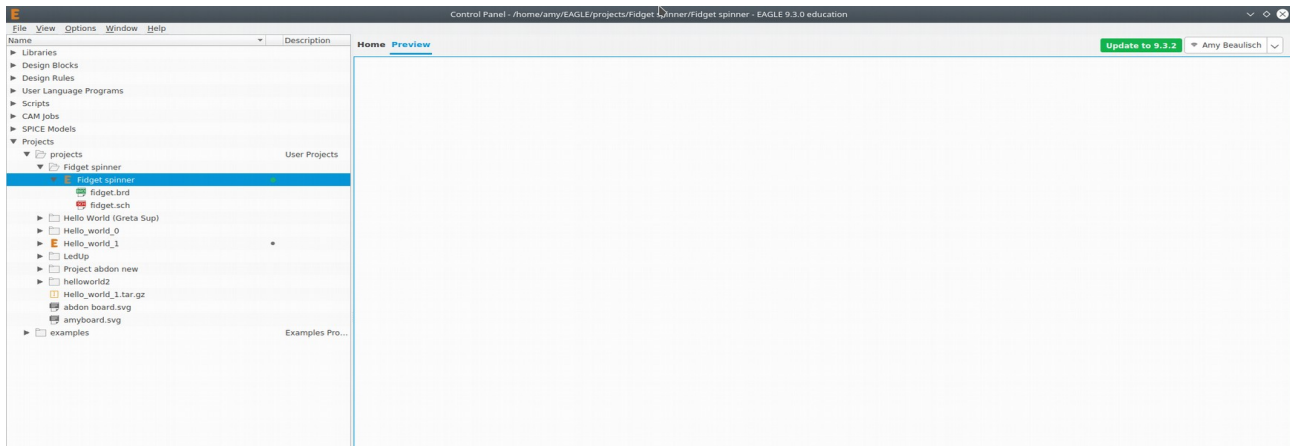
### 3. Sketch your circuit on paper

If it's an easy circuit and you have experience in pcb drawing, you might skip this part, but as you are reading this, make a sketch or 2 on paper. It will help you when drawing digitally.



## 4. Create a new project folder and schematic

Starting a schematic, board or circuit in Eagle means starting a new project. Once you open Eagle, you can right click on the tree that contains 'projects'. First start a project, once you have that, you can right click again and make a new schematic. If you know that you'll have multiple boards in one project, you can first make a folder.



## 5. To add or not to add, that is the question

The next step depends on how familiar you are with components. If you, just like me at the time, work with Eagle for the first time during the time of a course, or a specific project, you might want to add a library immediately (1). It can however be that the standard libraries are good enough and contain all the components that you need (2).

In case of the first (1) option, follow the next steps.

The **first step** here is to find the libraries online. Both sparkfun, addafruit, FabLabs, electronic component companies... have their libraries that you can download. Often they are already specified on subject. Resistors, diodes, micro controllers. Therefore it is important to know that both a LED and a Zener are diodes.

*\* Add links to common libraries\**

The **second step** is to add those libraries to you Eagle Library.

*\* search for the library icon\**

*\* the tab 'in design' will show you all the libraries that you use for the schematics\**

*\* the tab 'in use' will show you all the added libraries that you can use\**

*\* the tab 'available' will let you browse though (eg) you computer to find libraries \**

In this case, we downloaded the librarie and put it in the folder 'libraries' in the main folder of Eagle. We use the tab 'available' and the 'browse' button to add the libraries to our 'in use' tab.

Once you selected the right librarie, you click open and it will be added to the 'available' tab. Select the library and click 'use'. Once you've done that, you'll find the library back with the others and you can find your needed component.

In case of the second (2) option, please skip this.

## 6. Search your components in the Eagle Library

One of the hardest parts as a beginner is to find components in the library. If you are lucky and the company, project or school you are making the project for has a specific library, focus on that one and never mind the others.

Remember the BOM that we made in step 2? It's time to use that one again. Now, there are some important things that I didn't mention before. But they are becoming important now.

SMD = surface-mount device

SMT = surface-mount technology.

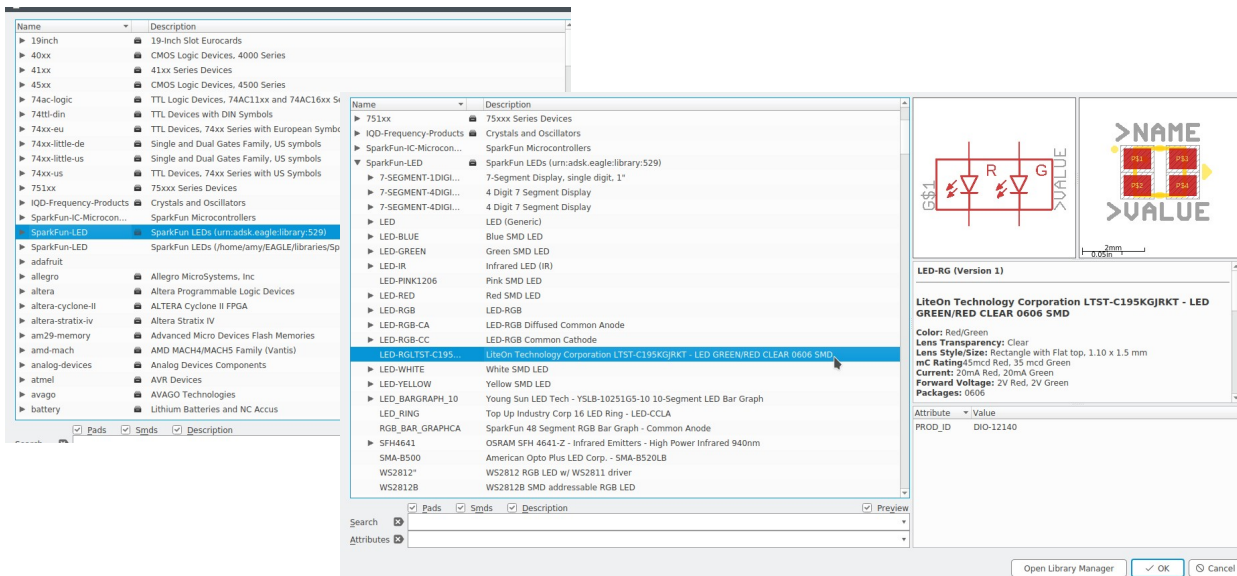
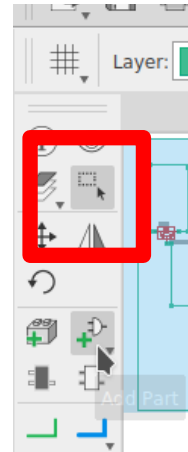
1206 = one of the possible (and more common) SMD packages. Size can be in metric and in units.

Packages = a set of components that fits a specific size. Size can be in metric and in units

Through hole = a component that goes all the way through the board and might connect a second layer of components. Mainly used before 1980's.

To add components to your schematic you need to click on the 'add part' button.

Once you found this, you can choose components out of all the added libraries. We, for example want to add a LED from the SparkFun library. This means that we'll click on the little arrow so that all the components will be shown. From there it's up to you to pick the right component with the right package and to see if its SMD or through hole.

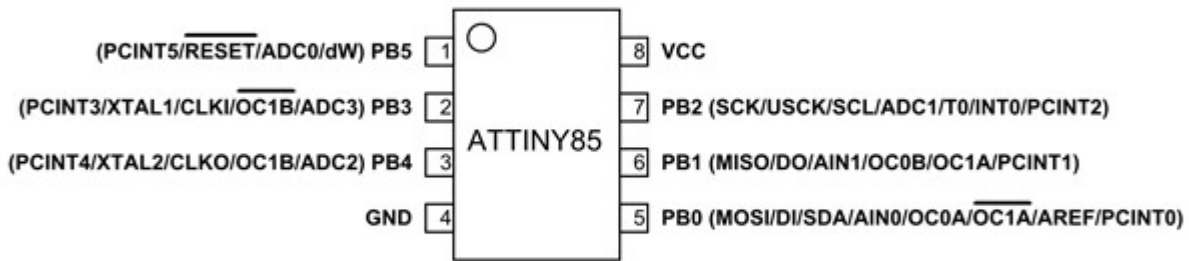


You will see that in some cases, you'll see the schematic drawing (first drawing to the right) and the drawing that will be used on the PCB itself or as we call it: the footprint (right). Make sure the footprint fits the components that you have!

## 7. Connect all the components together according to your schematic

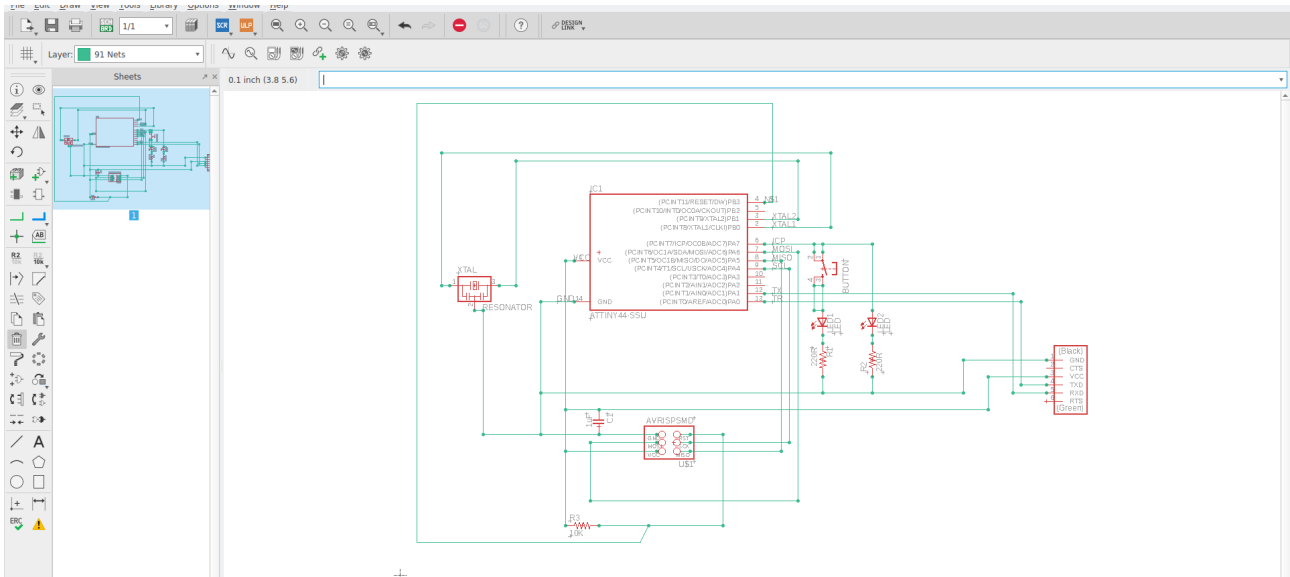
The next step is to wire or connect all the components to each other. If your work with micro-controllers, LED's be sure to also open the datasheet and read what it says. Especially LED's are very commonly used and we might forget that there are rules applying to them to.

For micro-controllers, know that not every pin is the same. It really depends there where you connect what in order to make your circuit work. Below is an example of the layout of the pins of a ATTINY85 micro controller. The little dot in the upper left indicates how the component should be placed.



source: components 101

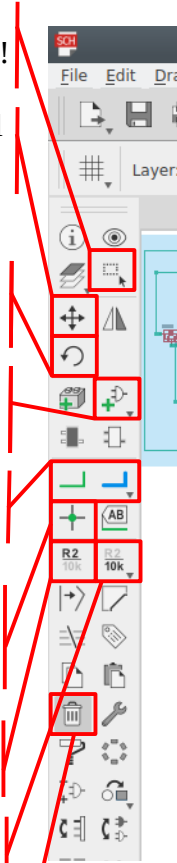
Once you've connected all the parts together it can look like this:



For now you don't have to worry about crossing lines. Just make sure everything is connected. See tips and tricks below

## Tips and Tricks:

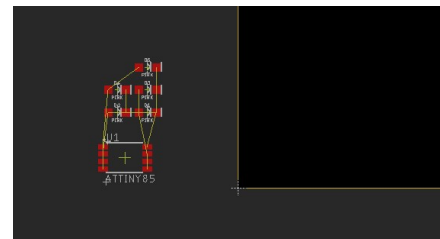
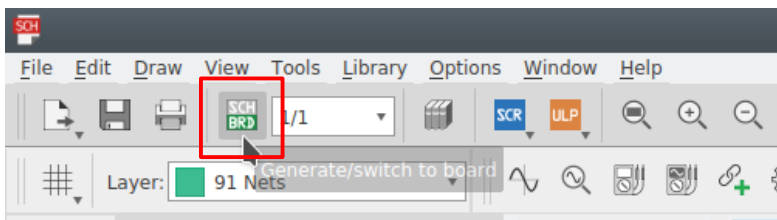
1. Select a component. Maybe the most important tool of all: the selection tool!
2. Move! Sometimes we need to move lines or components. Use the move tool for this. Also watch that there is a small cross in each component, use this to move the tool.
3. Rotate the tool Sometimes the connections stand or fall simply by rotating the component. In the schematic it is still okay, but this becomes very important when designing the board!
4. Add a component from the libraries.
5. Draw lines. If you want to draw lines, you can do this in green or blue. Just select the tool and click and drag.
6. Make sure all the components are actually connected. Use the 'junction' tool for this. If you use it, all the selected connections will have green dot's now.
7. Add a value. Add values to the components. Is it a 20K resistors or a 10K one?
8. Add a name. Name the components on the schematic as you'll need it later.
9. We all make mistakes, as we should, so sometimes we want to delete a line. Select the 'garbage' tool and delete the line, component or whatever you want.



## 8. Make your board

Once you have the schematic, it's time to move on to the board. This is often the place where you lose a lot of time as you have to figure out how the lines will run.

To go to the board use this button:

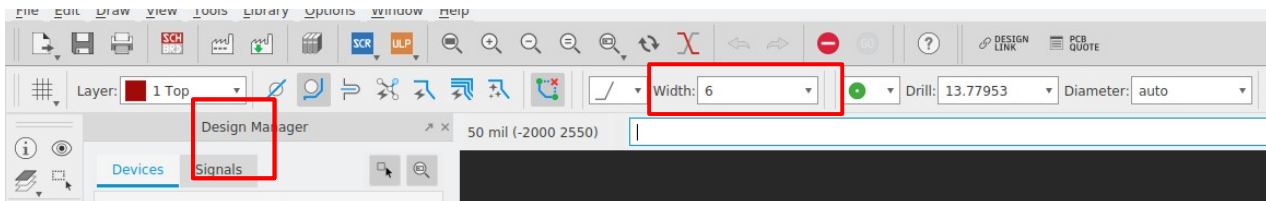


When you arrive in the 'board' screen, it's up to you to place all the components where you want them to have. At first it will all look like this: (see above)

To place all the components where you want them, use the same icons as when you were making the schematic.

## Tips and tricks:

1. Change the line thickness of the traces. If you want to make your pcb's on the vinyl cutter this will be important as they can't always go very thin.

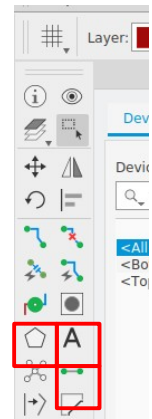


2. Layers. When exporting the board this will become important to see or not see extra information on the board.

3. Add a route. You've connected all the components in the schematic, and those lines became a guideline now. Reconnect them with the 'route' function.

4. Ripup. Break routes that you made and change them.

5. Autoroute. The opinions are double on this one. Some say to use it, others say don't. I mostly used it as a start and then rewire most of the components anyway.

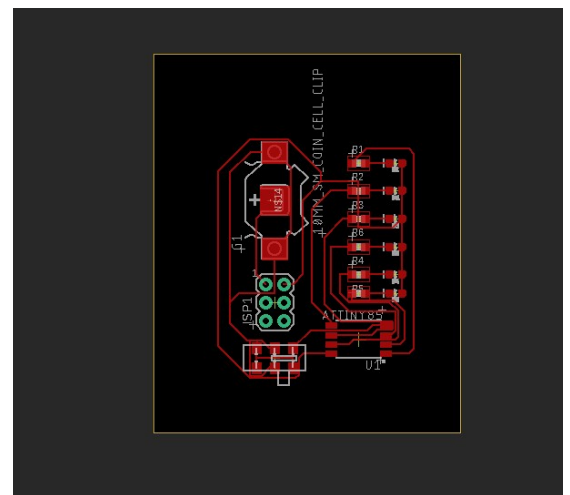
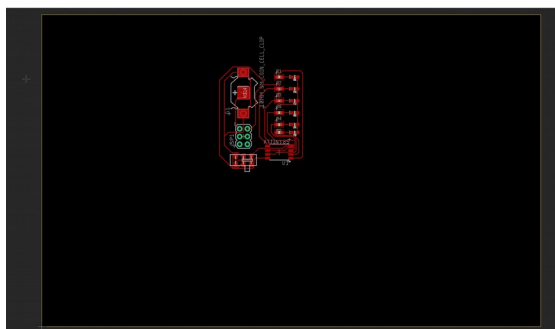


## 9. Export your board

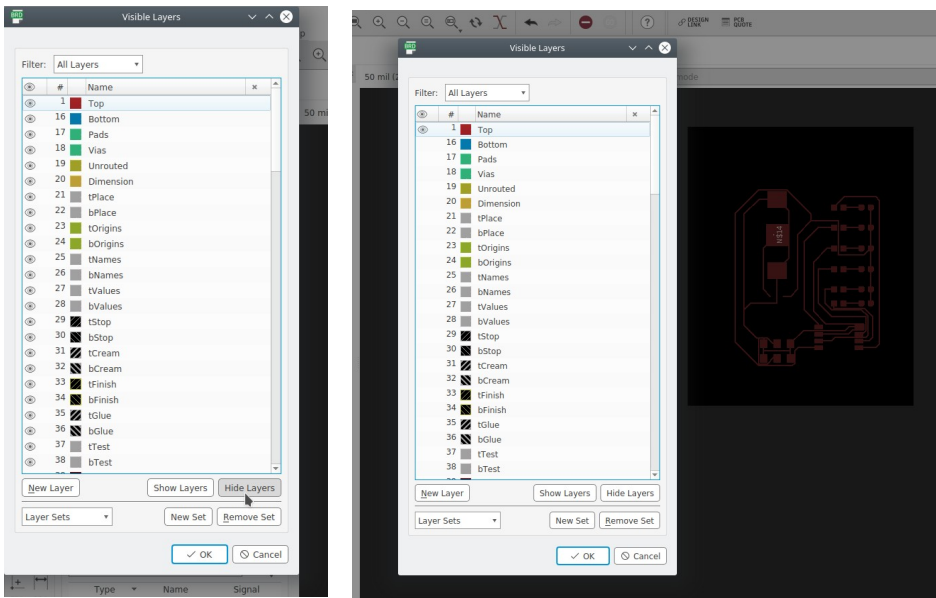
**I personally always work with etching or with the vinyl cutter which means that this last step might be different depending on your method of producing PCB's.**

Once your board is finished you want to export it and send it to the machine. The first thing that you have to do here is to resize the workplane. The workplane is the yellow line surrounding the circuit. If you would export now, the yellow line will be the outer side of the board. This is not what you want.

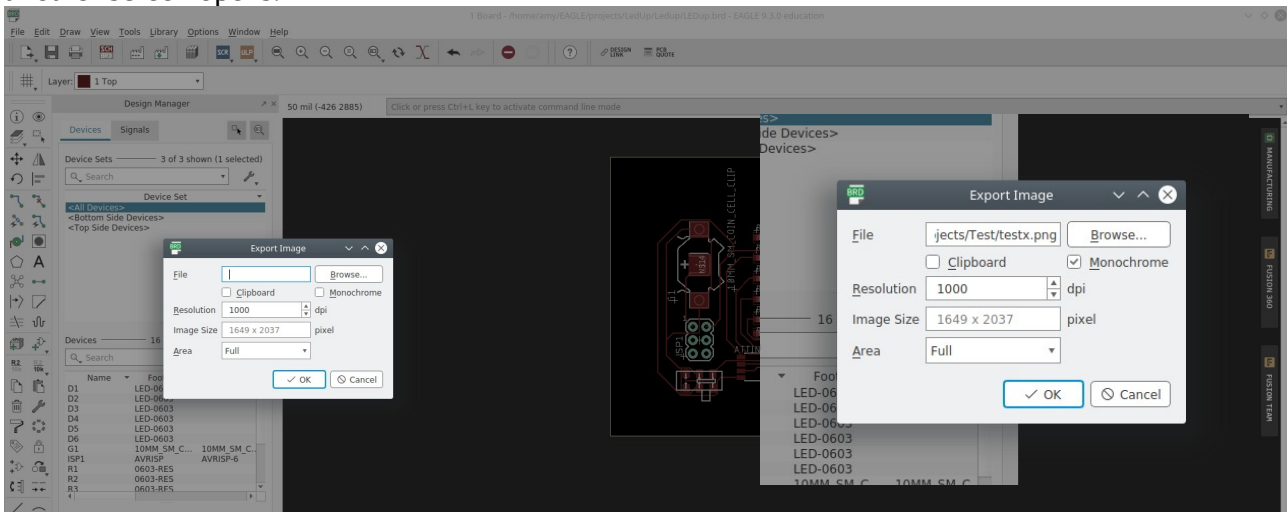
What you do want is the yellow rectangle as close to the board as possible. Drag the lines closer to your circuit.



Now that that's done, it's time to export. But before you export, go to layers. Hide all layers and only keep the first one on (the traces). Later you can repeat this step to, for example, have the names and values of the components, or other information.



Now that you've filtered out information, go to File → export → image. Once you click on image, another screen opens.



1. Select where you want the picture to be saved. Choose a logical location!
2. Select monochrome
3. Set the dpi to 1000
4. Click ok

## 10. Edit your drawing in Inkscape.

Depending on your next step you'll have to follow one of the 2 workflows here below.

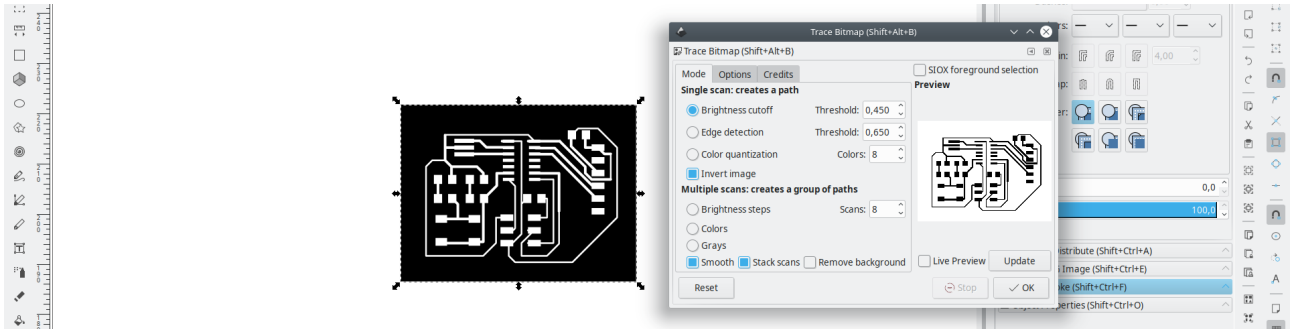
In general; import (or drag and drop) your newly made image into inkscape. Make sure you don't change the size of the png.



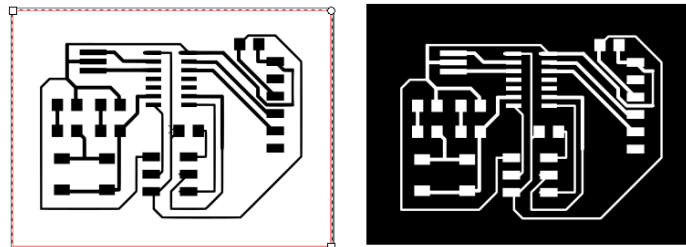
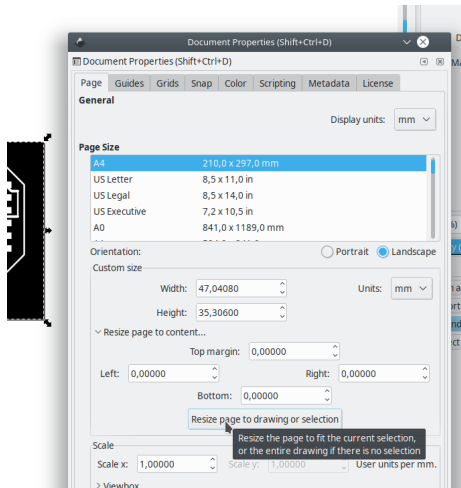
## For etching

For etching you will want black traces instead of the white ones you have now.

Go to Path → Trace Bitmap and invert the image. This will leave you with black traces. Drag them out of the original picture.



The second step is to keep the same size on paper as in the digital version, read, you want to have an outline that borders the circuit. To keep the same size, I do a 'resize page to content' on the original picture, delete the original picture and put the new traces in it. Add another square (in red here) and done you are.



## For vinylcutting

For vinylcutting you can either use the white picture with black traces, or the black picture with white traces.