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
- supplys
- 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 chidren 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

y inventory gx LED 1x litt sensor (no \$mol). 3x 151 weenstand 1x 100 weenstand 1x switch 1x mosfet.





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.

E		Control Panel - /home/amy/	Control Panel - /home/amy/EAGLE/projects/Fidget Anner/Fidget spinner - EAGLE 9.3.0 education	Control Panel - /home/amy/EAGLE/projects/Fidget spinner/Fidget spinner - EAGLE 9.3.0 education	Control Panel - /home/amy/EAGLE/projects/Fidget Anner/Fidget spinner - EAGLE 9.3.0 education	Control Panel - /home/amy/EAGLE/projects/Fidget Sinner/Fidget spinner - EAGLE 9.3.0 education
File View Options Window Help						
Name	 Description 	Home Preview	Home Preview	Home Preview	Home Preview	Home Preview
▶ Libraries						
Design Blocks						
 Design Rules 						
 User Language Programs 						
 Scripts 						
▶ CAM Jobs						
 SPICE Models 		the second descent and the first second descent second				
▼ Projects						
▼ 🗁 projects	User Projects					
Fidget spinner						
🔻 🖺 Fidget spinner						
tidget.brd						
tidget.sch						
Hello World (Greta Sup)						
Hello_world_0						
E Hello_world_1	•					
▶ □ LedUp						
Project abdon new						
helloworld2						
Hello_world_1.tar.gz						
B abdon board.svg						
amyboard.svg						
examples	Examples Pro					

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. Therfore 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 SparkFunlibary. 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.

Namo	* Description			
h 10inch	10 Inch Slot Eurocarde			
► 40vv	CMOS Logic Devices 4000 Series			
P 40XX	Alive Series Devices			
P 41XX	41xx series Devices			
▶ 45xx	CMOS Logic Devices, 4500 Series			
▶ 74ac-logic	 TTL Logic Devices, 74AC11xx and 74AC16xx Si 	Name 👻	Description	
▶ /4tti-din	 TTL Devices with DIN Symbols 	▶ 751xx 🗃	75xxx Series Devices	
▶ 74xx-eu	TTL Devices, 74xx Series with European Symbol	► IQD-Frequency-Products ■	Crystals and Oscillators	>NAME
▶ 74xx-little-de	Single and Dual Gates Family, US symbols	SparkFun-IC-Microcon	SparkFun Microcontrollers	
 74xx-little-us 	Single and Dual Gates Family, US symbols	▼ SparkFun-LED 🚔	SparkFun LEDs (urn:adsk.eagle:library:529)	
▶ 74xx-us	TTL Devices, 74xx Series with US Symbols	▶ 7-SEGMENT-1DIGI	7-Segment Display, single digit, 1"	
▶ 751xx	75xxx Series Devices	▶ 7-SEGMENT-4DIGI	4 Digit 7 Segment Display	
IQD-Frequency-Produce	ts Crystals and Oscillators	▶ 7-SEGMENT-4DIGI	4 Digit 7 Segment Display	
 SparkFun-IC-Microcon. 	SparkFun Microcontrollers	▶ LED	LED (Generic)	ZVILUE
SparkFun-LED	SparkFun LEDs (urn:adsk.eagle:library:529)	► LED-BLUE	Blue SMD LED	
SparkFun-LED	SparkFun LEDs (/home/amy/EAGLE/libraries/Sp	► LED-GREEN	Green SMD LED	2mm
▶ adafruit		► LED-IR	Infrared LED (IB)	J1. 0.0381
▶ allegro	 Allegro MicroSystems, Inc 	LED-PINK1206	Pink SMD LED	LED-RG (Version 1)
▶ altera	Altera Programmable Logic Devices	► LED-RED	Bed SMD LED	
altera-cyclone-II	ALTERA Cyclone II FPGA	EED-RGR	IED-BGB	LiteOn Technology Corporation LTST-C195KGJRKT - LED
altera-stratix-iv	 Altera Stratix IV 	► LED-RGB-CA	LED-RGB Diffused Common Anode	GREEN/RED CLEAR 0606 SMD
▶ am29-memory	Advanced Micro Devices Flash Memories	ELED RGP CC	IED RGB Common Cathodo	Color: Red/Green
▶ amd-mach	AMD MACH4/MACH5 Family (Vantis)		Libor Technology Commention LTCT CLOSKCIPKT, LCD CDCCNIPCD CLCAD GOOS CND	Lens Transparency: Clear Lens Style/Size: Rectangle with Flat top, 110 x 15 mm
analog-devices	Analog Devices Components	LED-RGLIST-CI95	Theory recimining Corporation ETST-CLASKOJKKT - LED GREEN/KED CLEAR 0000 SMD	mC Rating45mcd Red, 35 mcd Green
▶ atmel	AVR Devices	EED-WHITE	White SMD LED	Current: 20mA Red, 20mA Green
▶ avago	AVAGO Technologies	EED-TELLOW	TEIDW SMD LED	Packages: 0606
battery	Lithium Batteries and NC Accus	▶ LED_BARGRAPH_10	Young Sun LED Tech - YSLB-10251G5-10 10-Segment LED Bar Graph	
, outery		LED_RING	Top Up Industry Corp 16 LED Ring - LED-CCLA	Attribute * Value
Pads Smds Description		RGB_BAR_GRAPHCA	SparkFun 48 Segment RGB Bar Graph - Common Anode	PROD_ID DIO-12140
Course Ma		▶ SFH4641	OSRAM SFH 4641-Z - Infrared Emitters - High Power Infrared 940nm	
		SMA-B500	American Opto Plus LED Corp SMA-B520LB	
		W52812"	WS2812 RGB LED w/ WS2811 driver	
		WS2812B	WS2812B SMD addressable RGB LED	v
		Pads V Sr	nds 🔽 Description 🔍 Previ	ew
		Search D		*
		Attailustas (2)		-
		Attributes tar		•
				Onen Library Manager

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 microcontrollers, 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 loose a lot of time as you have to figure out how the lines will run.

To go to the board use this button:

File Edit Draw View Tools Library Options Window Help Image: I	SCH			
$ \begin{array}{c c c c c c c c c c c c c c c c c c c $	<u>File Edit D</u> raw	<u>View T</u> ools <u>L</u> ibrary <u>O</u> ptions <u>W</u> indow <u>H</u> elp		
Layer: 91 Nets Generate/switch to board 4 ∞ \otimes \otimes \otimes \otimes \otimes \otimes \otimes \otimes	₿₽₽₿			
	tayer:	91 Nets Switch to board $\sqrt{2}$ $\sqrt{2}$ $\sqrt{3}$ $\sqrt{3}$	4TTINY85	

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.



Created by Amy Beaulisch February 2019





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 guidline 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 yellor line surounding 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.





Created by Amy Beaulisch February 2019





#_ Layer:

Dev

Devi

Q,

<All <Bo

(i) (i)

Ø .

+ /⊾

*

А

€

* *

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 \rightarrow export \rightarrow image. Once you click on image, another screen opens.

File Edit Draw Yiew Tools Library Options Window Help							
[⊒,	🗄 🖶 🖽 🎟 🕮 🔤 🖳 ର ର ର ର୍ ବ X 🖛 🗢 🗢 🗍 💿	$\mathcal{P}_{\text{UMV}}^{\text{PMSON}} \equiv \mathcal{R}_{\text{OOH}}^{\text{PMSON}}$					
#	Layer: 🚺 1 Top 🔹						
	Design Manager * × 50 mil (-426 2885) Click or press Ctrl+L key to activate command line		Ŧ				
(i) ®	During Circula D. D.		-				
Ø	Devices signals	de Devices>	0				
+++ //	Device Sets	Devices>	MAN				
	Q Search		IUFA				
+) =			CTUR				
23	<all devices=""></all>		ING				
* *	<bottom devices="" side=""></bottom>						
		<u>File</u> <u>jects/Test/testx.png</u> <u>B</u> rowse					
O A	Export image						
20	Eile Browse		ISNE				
	Clipboard <u>M</u> onochrome	Resolution 1000 🗘 dpi	ON 3				
17 12	Resolution 1000		8				
≣\≣ ปป	Image Size 1649 x 2037 pixel	image Size 1649 x 2037 pixel					
	Devices 16						
R2 R2	Q_Search		•				
BB	Name * Foo		FUS				
	D2 LED-06	LED-06 V OK O Cancel	ION				
	D3 LED-0603 D4 LED-0603		TEAN				
1	D5 LED-0603	150,0603					
۵ 🔇	G1 10MM_SM_C 10MM_SM_C	LED-0603					
10 6	R1 0603-RES	LED-0603					
CE	R2 0603-RE5	10MM SM C 10MM SM C					
1							

- 1. Select where you want the picture to be saved. Choose a logica 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 \rightarrow 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.



