PCB Design Laboratory Manual

Introduction to Product Development and Management for Engineers
GNG 2101
Faculty of Engineering
University of Ottawa
Winter 2019

Checks required by TA:

Group Member	Board design

Dr. Hanan Anis
Dr. David Knox
Dr. Patrick Dumond

Objective

The purpose of this lab is to teach students how to design a printed circuit board (PCB) based on a circuit schematic. They will use Eagle to recreate the circuit and design a PCB with the corresponding components. Making custom libraries and parts as well as proper identification and layout will be introduced.

Background

A Printed Circuit Board is a board that has pads that are connected with traces (lines) which make a circuit. It allows electrical components to be connected and get power when they are soldered to this board. Solder is metal that glues the component to the board and it provides the connection between them. A PCB is made up of many layers of different materials, as shown in Figure 1.

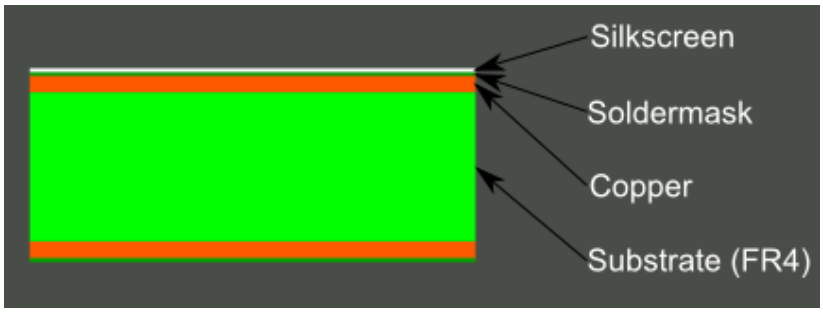


Figure 1. PCB composition

The substrate (FR4) is fiberglass which gives the board rigidity and insulates each side from the other. The copper is the conductive material in the board. It can be only on one side of the board or more commonly on both sides to have a double sided board. The next layer, the soldermask, is placed on top of the copper to avoid unwanted contact from other conductive elements. The white silkscreen is then placed on top to indicate the position of different components or any notations you would like to add.

Figure 1 shows a 2-layer board, but more inner layers can be created by sandwiching multiple 2-layer substrates. If connections need to be made between layers, then holes need to be drilled then plated through inside with metal to make the connection between the layers (Figure 2). These holes are called vias. We are making a 2-layer through hole board, which means our components will have pins that pass through the PCB and will be soldered on the opposite side.

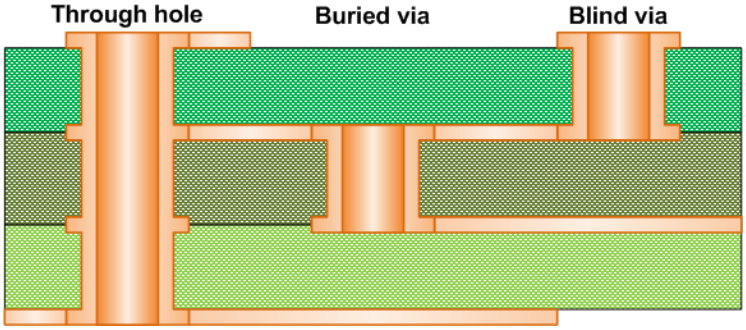


Figure 2. PCB internal routes

Through hole vias are the most common and the cheapest, since it is easiest to drill a hole all the way through a piece of material with layers that have already been sandwiched together.

Designing and using a PCB should be the last step after testing your circuit first on a breadboard, then on a prototyping board. These methods are used in the prototyping stages of your design while a PCB is more of a final product but is not always.

Eagle will be used since it's free, can be used on many platforms and has lots of support out there. It is also one of the industry standards. We will be going over the basics but feel free to look at Eagle (http://eagle.autodesk.com/eagle/documentation) as well as sparfkfun tutorials (https://learn.sparkfun.com/tutorials) for some more complicated functionality and more explanations.

Apparatus and Equipment Overview

The following components will be used in this lab:

- 1 x computer
- 1 x mouse (optional)
- 1 x internet (to download Eagle)

Pre-Lab Preparation

Before arriving in the lab, students should review the lab manual and familiarize themselves with the lab setup and procedures. Students are encouraged to use their own computers for this lab, provided that they have downloaded and installed Eagle (https://www.autodesk.com/products/eagle/free-download).

Prelab Questions

What does PCB stand for?

_____

_____

What keeps a PCB rigid?

_____

_____

What is the conductive material used in a PCB?

_____

_____

Explain how you would make a connection that started on one side of a board and then continued on another.

_____

_____

Why is it important to have a silkscreen on a PCB?

Overview

The following circuit diagram and parts list should be referenced throughout the lab, it is a circuit for a USB charger. Each team member should try to do this lab.

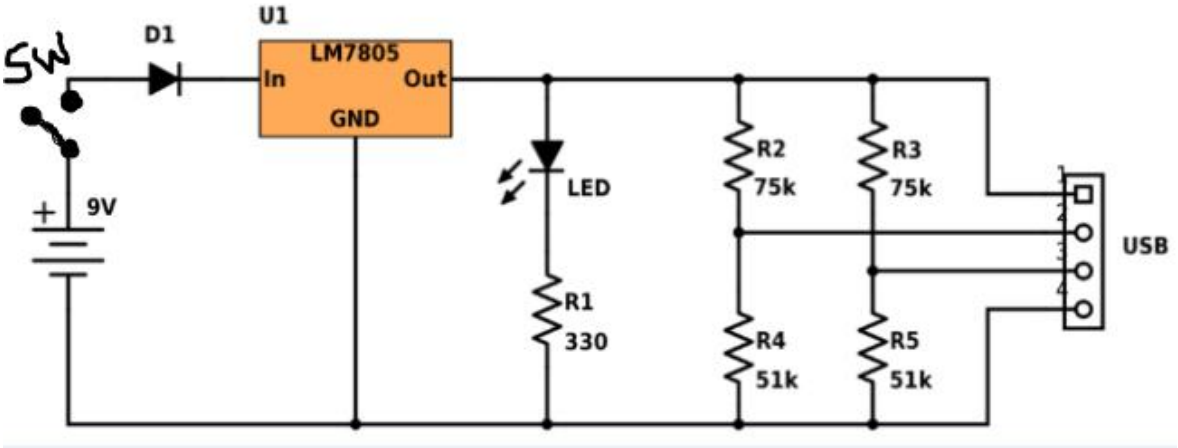


Figure 3. Circuit diagram

Table 1. Parts list

PART	VALUE	DESCRIPTION	Didgikey part num
U1	7805	Voltage Regulator	LM7805CT-ND
LED	Standard Output	Light-Emitting Diode	C5SMF-BJE-CR0U0451-ND
R1	330 Ω	Resistor	CF14JT330RCT-ND
R2	75 kΩ	Resistor	CF14JT75K0CT-ND
R3	75 kΩ	Resistor	CF14JT75K0CT-ND
R4	51 kΩ	Resistor	CF14JT51K0CT-ND
R5	51 kΩ	Resistor	CF14JT51K0CT-ND
D1	1N4004	Rectifier Diode	1N4001-TPMSCT-ND

USB	Type A Socket	Solderable	36-921-ND
9V	9V	Battery Connector	36-232-ND
SW	-	ON/OFF switch	563-1565-ND

Terminology

- Schematic/diagram: visual representation of a circuit with symbols
- Symbol: representation of a component visually
- Package: representation of a component based on its physical configuration
- Device: part with linked symbol and package
- Library: add new parts in a custom folder (library) to keep track of them
- Eagle Schematic/Board: you can switch between the schematic (symbol) and board (package) in Eagle

Eagle Tools

- Group : select one or many components
- Move : click on the little + on each component to move them
- Rotate : click on a component to rotate 90^{o}
- Add : add new component

Procedure

Part A – Making the schematic

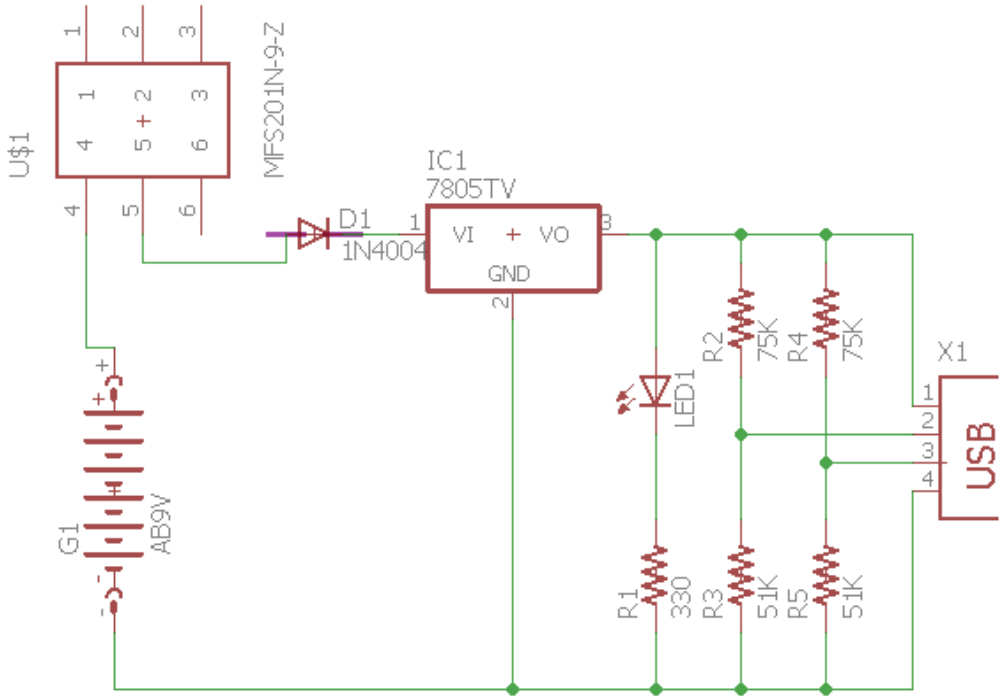
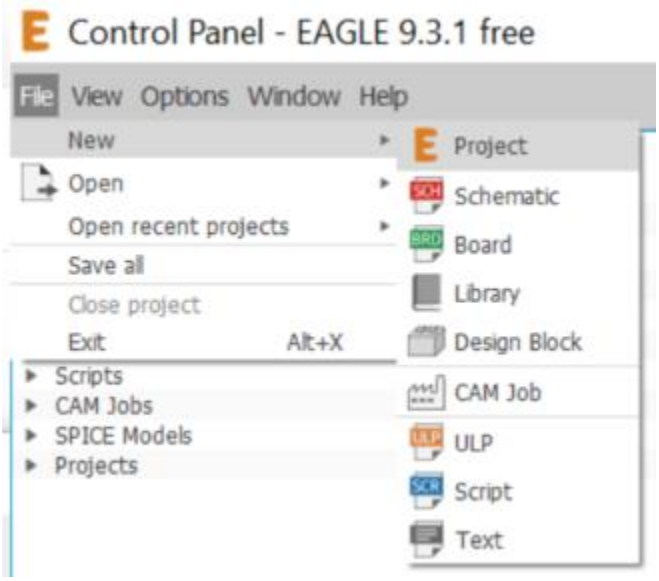
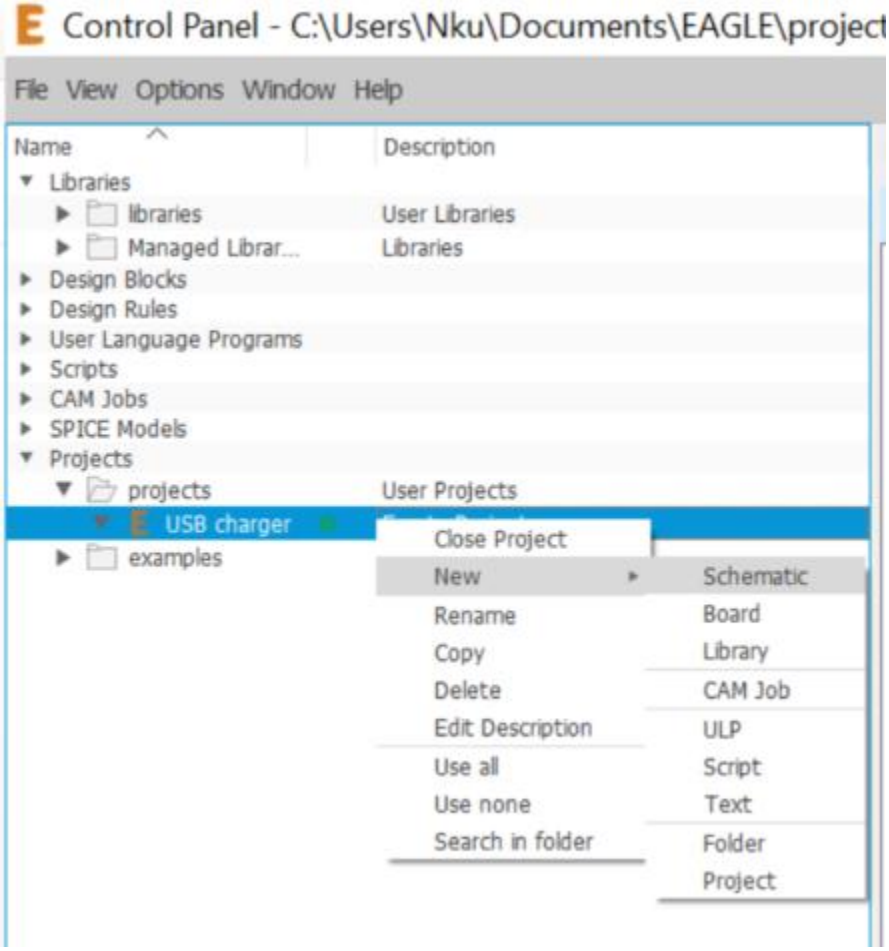


Figure 4. Circuit schematic in Eagle

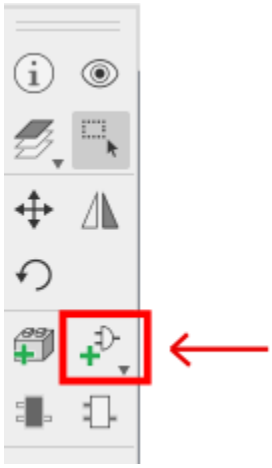
1. Open Eagle and create a new project, File>New>Project, name it USB charger or something similar.



2. Right-click on your new project to start making your schematic, New>Schematic.



3. To make the circuit you will need all the components. Click on the 'add a part' button to open the existing ones in the available libraries. You can search 'resistor' for example or a part number like 'LM7805'.

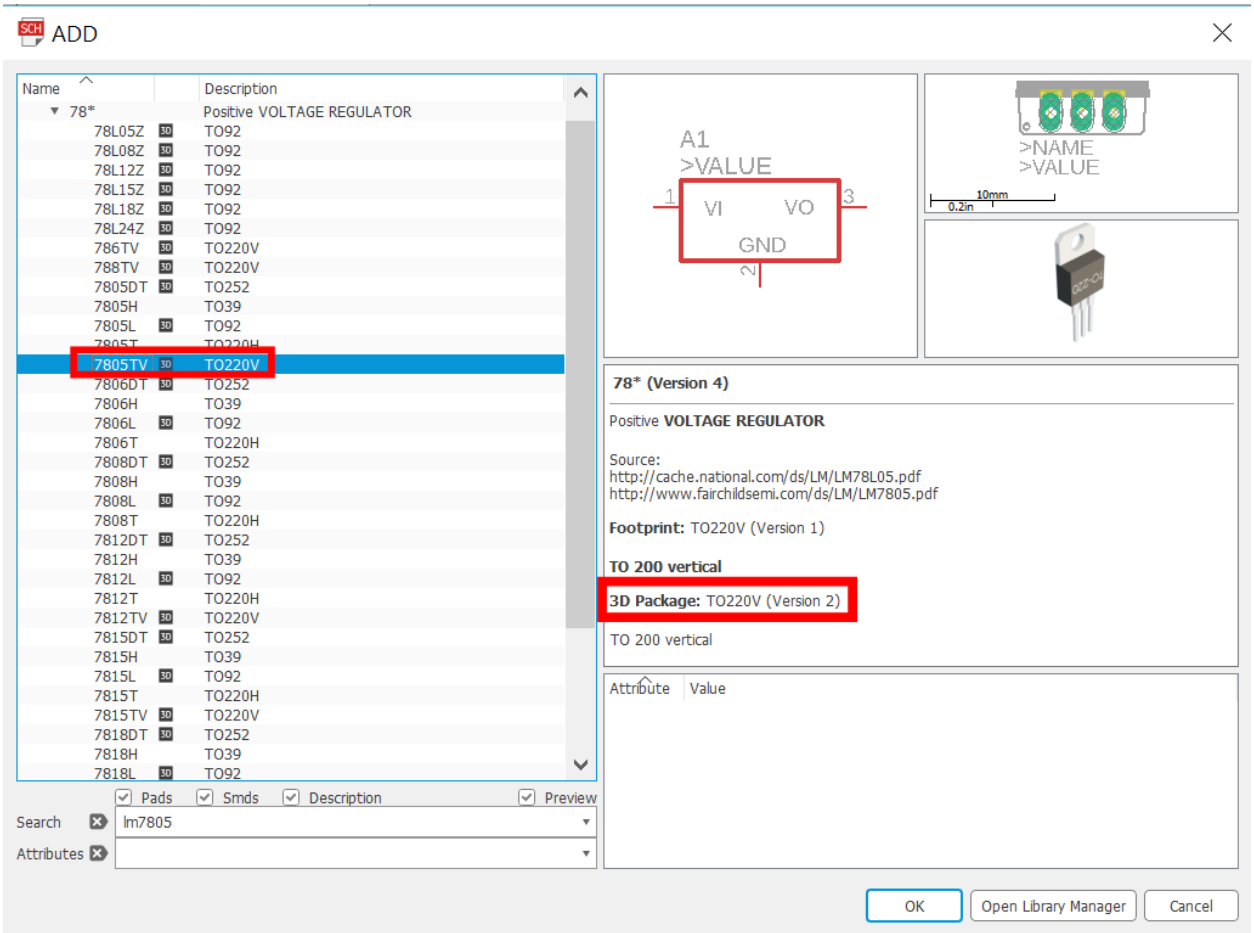


4. When making the schematic, for our purposes, the symbol in the schematic doesn't matter. However when it comes to the board the size of the part needs to correspond with the actual one that we have so it can be soldered to the PCB. To do that, you can google the digikey part number to find the package (it's the size, orientation, etc of the physical

part). Ex: package of the voltage regulator is TO-220 which can be found in the Eagle library.

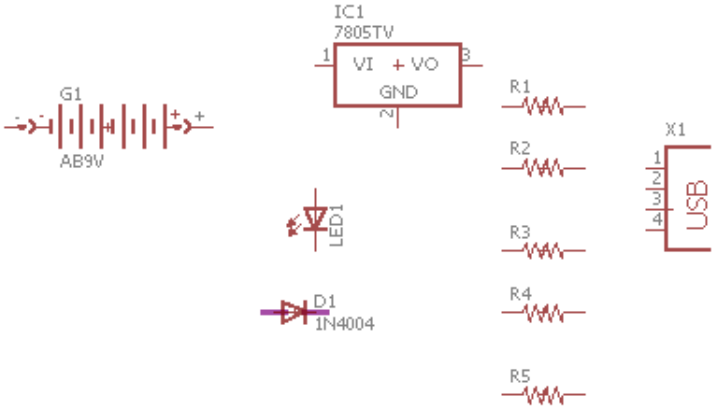
Operating Temperature	-40°C ~ 125°C
Mounting Type	Through Hole
Package / Case	TO-220-3
Supplier Device Package	TO-220AB

Report an Error



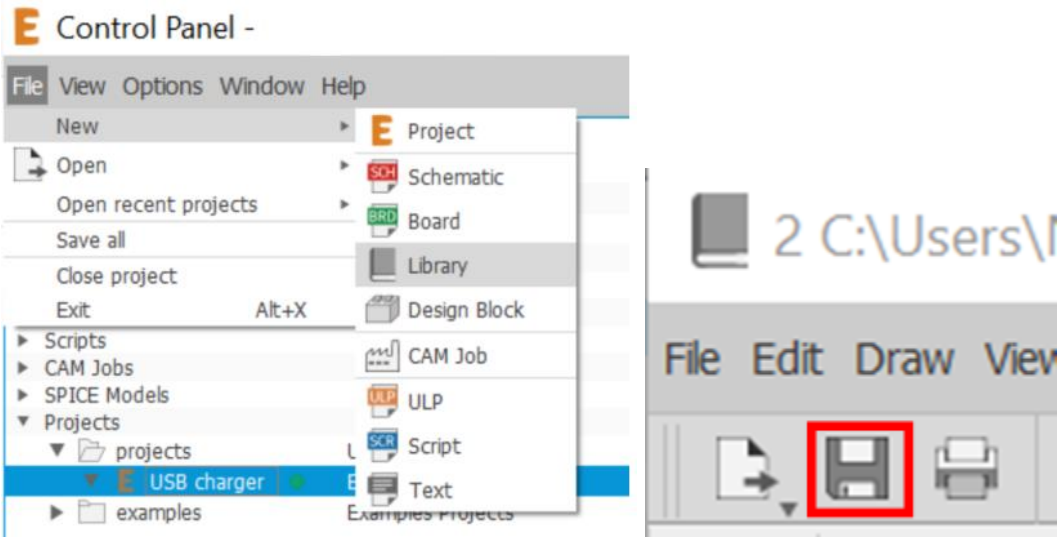
5. Refer to the parts list to get all of the proper parts on your workspace, search in Eagle for the part number, part name or package to find the right one. If you were to go through this process on your own you would have to find all the packages in the datasheets of all components but here is a list to help you for this lab:
    a. Voltage regulator- **LM7805**, use package TO220V (vertical orientation to minimize space)
    b. LED- **led5mm**
    c. Resistors- **resistor**, pick from the R-US list anything bigger then 207/7 (they have a power rating of 0.25W which is package 207 and they are 6mm long)
    d. Diode- **1N4004** (DO41 package)

e. USB socket- **PN87520**
f. Battery connector- **AB9V**
g. Switch- you will notice that you can't find a proper part, you will make your own

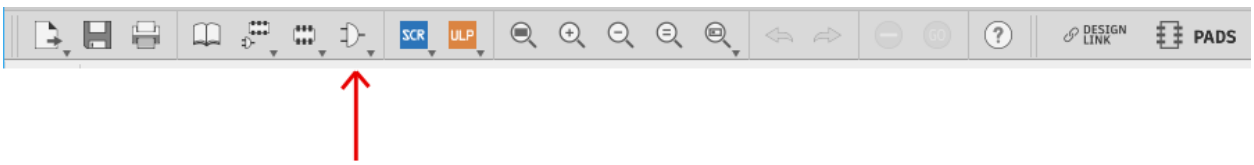


Part B – Making a custom component

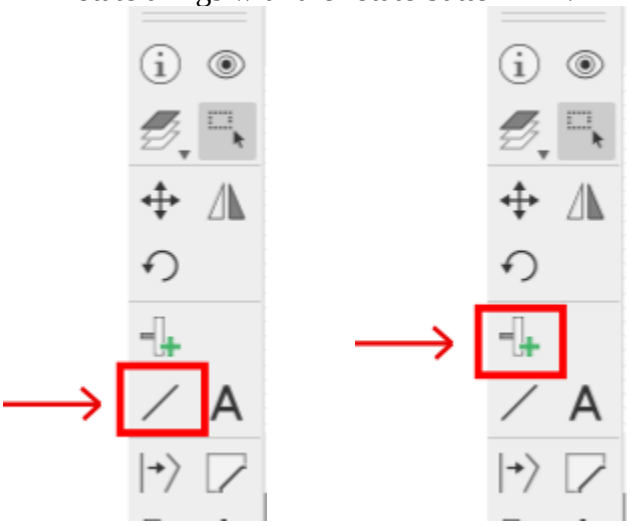
1. First open the datasheet of the part you are creating (the switch MFS201N-9-Z).
2. You will create your own library. In the control panel, click File>New>Library. In the new window that pops up click save and name it (make sure to save it in lbr which is libraries).

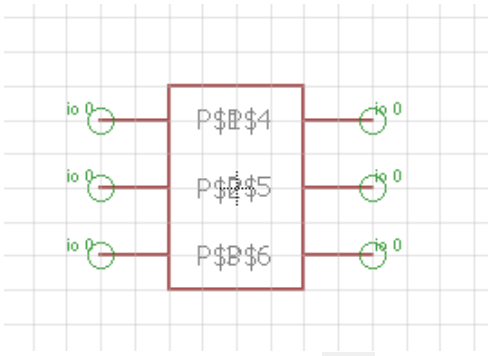


3. Start with the symbol of the part, click 'Add Symbol…' in your library and name it MFS201N-9-Z.

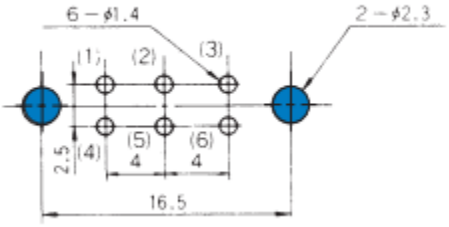


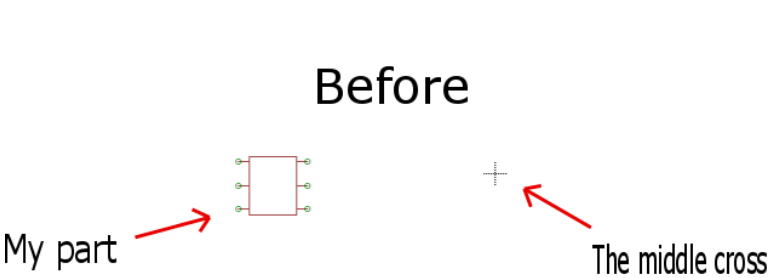
4. Create a box with ‘Line’ and add the correct amount of pins to it (6 in our case). You can rotate things with the rotate button . The size of this symbol doesn’t really matter.

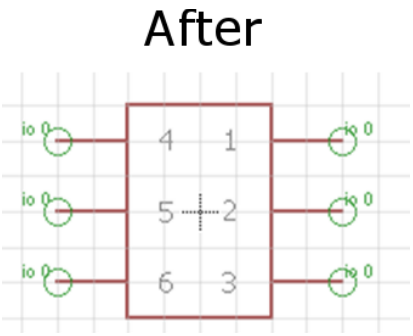




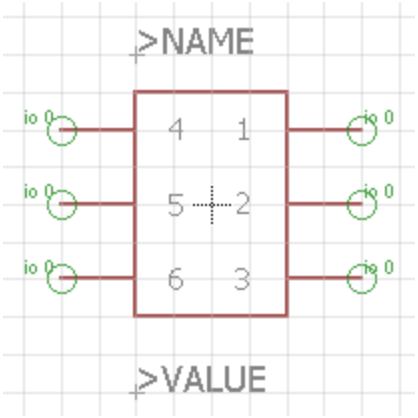
5. Name the pins so they correspond with the datasheet with the name button . Make sure the part is centered by selecting all of it with the group button and moving it to the middle cross. If you don’t see the middle cross, you may want to zoom out a little until it becomes more clear



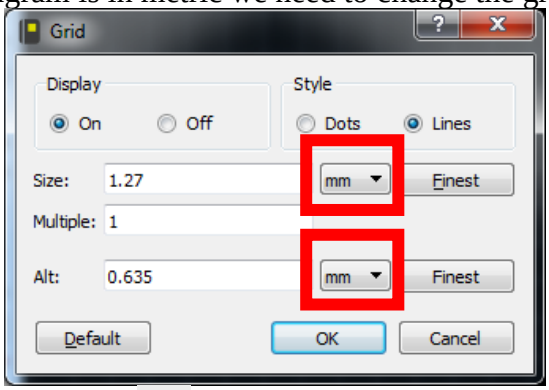




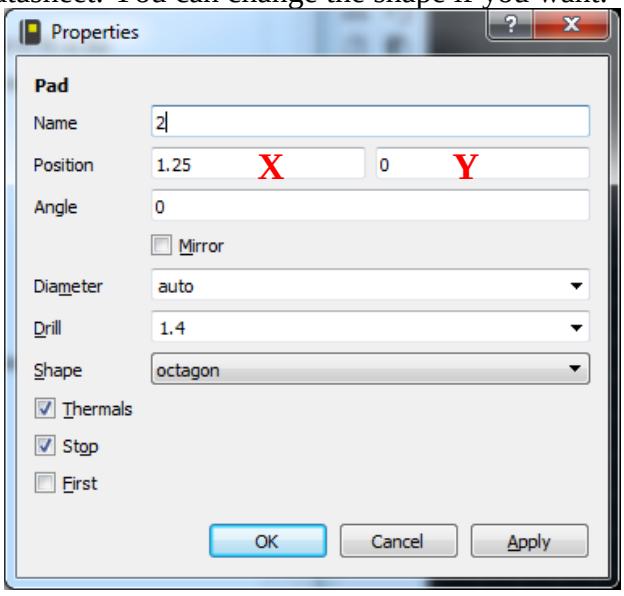
6. Click the text button to add name and value tags, then move them to Names and Values layers respectively with >Layer… by clicking on the tags.

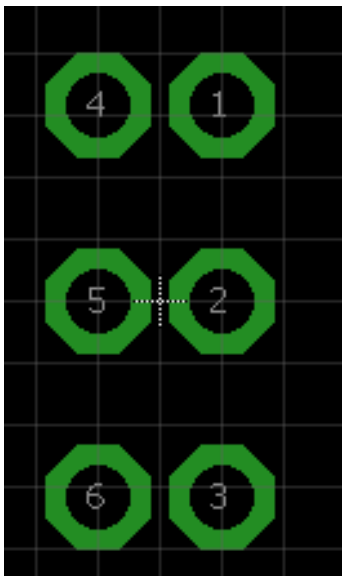


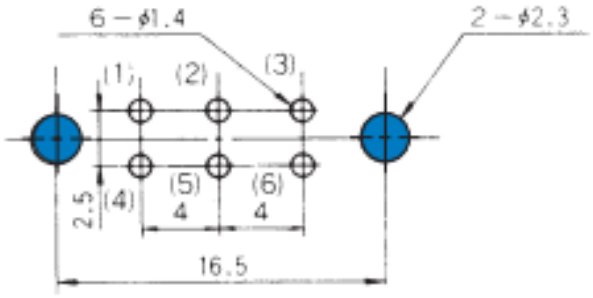
7. Save your symbol and start making the package (footprint) with the package button . Name it MFS201N-9-Z.
8. The size of the package is very important since our physical part will need to fit in the board. Since our diagram is in metric we need to change the grid to mm.



9. Place the right amount of pads on your workspace. Right-click on them to get to Properties. Change their name, position (x, y) and drill size to correspond with the datasheet. You can change the shape if you want.



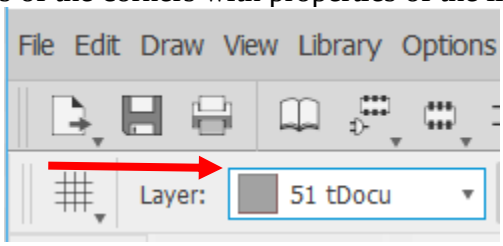


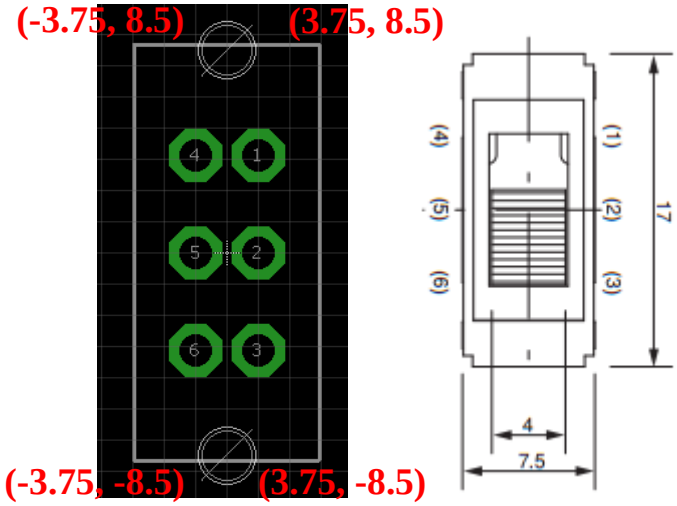


Pad	X	Y
1	1.25	4
2	1.25	0
3	1.25	-4
4	-1.25	4
5	-1.25	0
6	-1.25	-4
Drill top	0	8.25
Drill bottom	0	-8.25

10. Two drill holes also need to be added for this switch , change their properties as well. To add a different size of drill, click >Drill>…

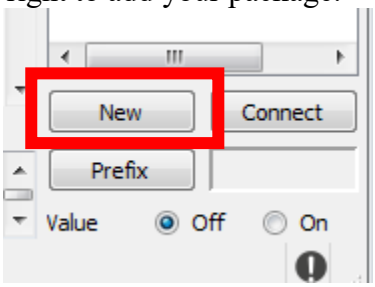
11. You should also indicate where the outlines of the part will be. Use line to draw a box around the pads and holes. Make sure the layer is tDocu before drawing the line. Adjust the coordinates of the corners with properties of the lines.



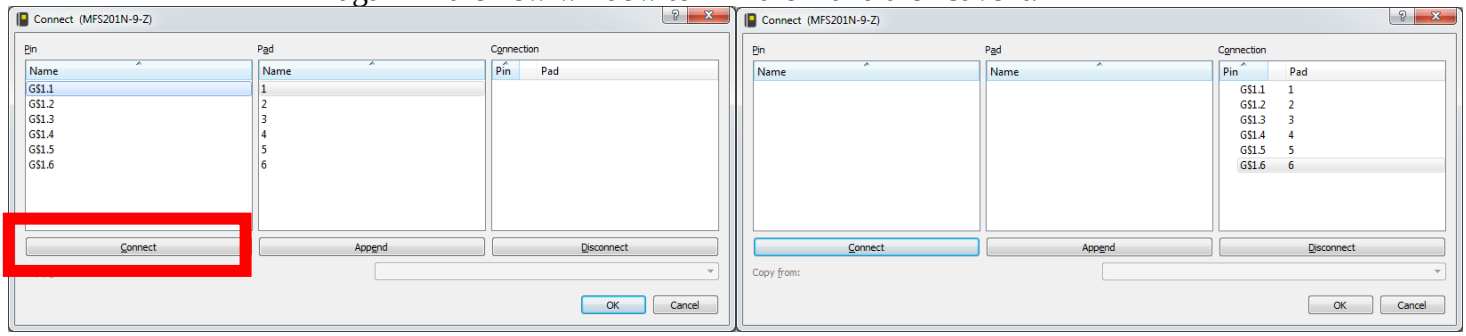


12. Add >NAME and >VALUE with the text button , placing them on the tNames and tValues layers respectively (in properties).
13. You are now ready to associate the symbol with the package. Save your package and open the device , name it MFS201N-9-Z.

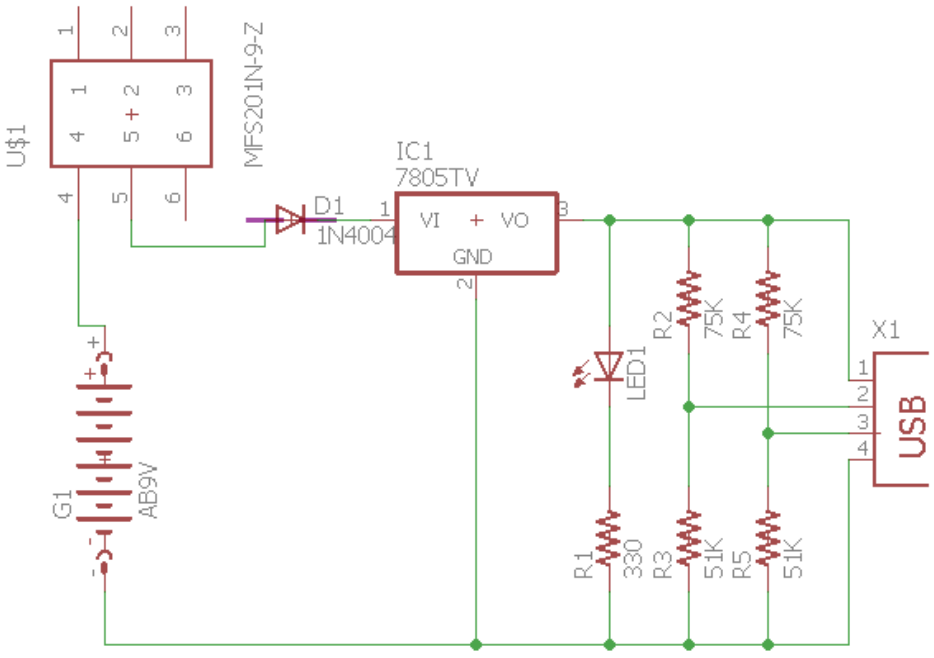
14. Add your symbol with the add button and place it in the middle of the workspace. Click ‘New’ in the bottom right to add your package.



15. Associate the pins in the symbol with the pads in the package with ‘Connect’. Click ‘Connect’ again in the new window to link them and then save it.

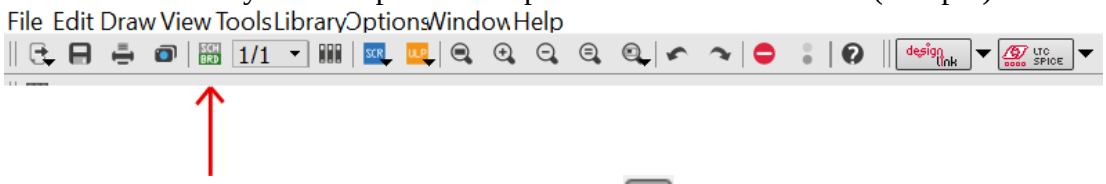


16. Finally, add this part to your schematic by navigating to the part in the control panel (Librairies>Name_of_your_library>MFS201N-9-Z) and click ‘Add’. *Your schematic needs to be open. Complete your circuit with the net wire button .



Part C – Making the board

1. Click on switch to board to see all the packages of the parts. Use the rotate and move tool to arrange them inside the white box while trying to visualise where the traces will be. Try and compact all the parts to make a small PCB (cheaper).



2. When your parts are placed, use the route tool to permanently place the copper traces on the board. We could use both layers but it is possible to only use one, choose the top layer for the traces (red). Make sure that they don't come too close to any other components or cross any other traces.
    a. Note: try to route your connections without moving your components to untangle all the wires.
    b. Remember that a trace can pass between two holes from a component (passing over the lines from the silkscreen layer).
    c. Remember that Eagle will show you a certain arrangement of connections but you can change them following the rules of electrical nodes and circuitry. Ex. If all negative pins of components are connected to the negative battery terminal, then any one of the pins can be connected to each other and only one needs to touch the battery.
3. When everything else is done, use the Design Rule Check (DRC) to make sure your board is ready to be printed.
4. Save your file in a safe location.
5. Get rid of all your connections with the ripup tool and try again using the autoroute button only for the top layer. Save this new file using a different name so that you can compare the two, what differences do you notice?

_____

_____

Next steps

Next week you will be given a PCB with this circuit, made by us, and all of the components to allow you to practice through-hole soldering. Compare our design with yours and let us know if and how yours is better☺

Additional Resources

- Here is a similar tutorial for Eagle that you might find useful, https://learn.sparkfun.com/tutorials/using-eagle-schematic
- To practice your PCB design skills and for project ideas you can look for some here: https://circuitdigest.com/diy-pcb-projects.