**Printed circuit board (PCB) design**

There are several free tools for PCB design – I have found EasyEDA (<https://easyeda.com/>) to be most straightforward but some features are more advanced in Eagle (free with .edu email <https://www.autodesk.com/products/eagle>). In EasyEDA, the general steps for board construction are

1. Click ‘New PCB’ and set the board size you want (this can be edited later but it is a little tricky)
2. Add ‘footprints’ of the components that will go on the circuit board (ie microcontroller, d1 mini, etc). These are found in Place>Footprints, and are mostly user-contributed so double check that the order of pins and board shape match your components. The grey circles/squares will be the holes in the PCB that you can put pins through. These footprints can also be edited but it’s easiest to find one that already matches what you need. Drop onto the board wherever you think it will fit; you will likely end up moving components around several times if you are trying to design a compact circuit board. EasyEDA assumes you would like to place multiple footprints of any design you use; press escape after dropping what you want onto the board to get your normal cursor back.
3. Use the PCB Tools>‘connect pad to pad’ tool to virtually wire your components together (e.g. click on the 3.3V output of the microcontroller, and then click the ‘vin’ or ‘vcc’ header of a sensor). This will add blue lines between all of the connected pads; these are just keeping track of what should connect so don’t worry about where they are (it’s ok for them to cross, go over other components, etc).
4. Once all components are connected, click Route>Auto Router to lay out the traces on the board. This tool is a bit finnicky and sometimes doesn’t run. Try refreshing the page or coming back later. It also sometimes has trouble with some footprints. It is always possible to manually add those traces using the PCB Tools>Track tool, but make sure that you do not accidently cross or connect with any other traces that should be separate. You can add traces on the top or bottom of the PCB; switch by clicking the colored square next to the layer you want to edit in the ‘Layers and Objects’ window. It is ok for traces in the top layer to cross traces in the bottom layer, but don’t cross traces within the same layer. I find it hard to keep track of the different connections and have ended up printing out copies of the top and bottom layers to check all connections manually.
5. Using the ‘Layers and Objects’ window you can add text/boxes/etc to be printed on the board. Most footprints already include some amount of printed text which you can edit. To do this click on the yellow box next to the ‘TopSilkLayer’ in the menu. Note that text in grey on the circuit boards will not get printed, only the yellow text/lines in the silk layer.
6. When done, you can download a Gerber file that can be used to print PCBs by going to Fabrication>PCB Fabrication File(Gerber). From this window you can automatically export to JLCPCB directly for printing or save the file (a zipped folder with information about each PCB layer) to print later.