Topics	Links	Workflow and Notes
Summary & Intro	https://github.com/JohnGENZ/ZL3TILProject	A series of short talks covering: 1) Using CAD for PCB design and fabrication 2) PCB making methods and using a fabrication service 3) SMD components and soldering 4) Making a case with 3D printing and folding 5) Programming an ATTINY using the Arduino IDE. We will be using a small function generator as an illustrative project. The support files and notes are on GitHub. The talks will focus is on work flow not software. If you have used Word, Excel, Photoshop, Inkscape or programmed your phone then you can use PCB CAD and you will find it empowering. SMD is not hard, it makes life easier – and solder paste is magic!
Choice of CAD package	https://www.adafruit.com/product/931 https://www.sparkfun.com/products/10124 https://youtu.be/1AXwjZoyNno https://youtu.be/CCTs0mNXY24 https://youtu.be/oId-h6AeXXE https://forum.sparkfun.com/viewtopic.php?t=20087	Eagle, KiCAD, Altrium, OrCAD, PADs …… Eagle well supported by Open Source Community – Adafruit, SparkFun make Eagle drawings available Supported by the major parts suppliers e.g. Mouser, Digikey. Good online resources – a search for “Eagle Tutorial” gives 53 million plus hits. My curated collection is here – old but covers the basics – maybe you only need the first one. When looking for help it pays to be specific – e.g. merge Eagle library with Eagle library.

The illustrative project – a Tiny Function Generator	http://www.technoblogy.com/show?2FCL http://www.technoblogy.com/show?22HF http://www.technoblogy.com/show?20W6	A useful audio function generator. All the design work done including some clever code. Has a through hole PCB already – we are going to make an SMD board. Easy to modify or add waveforms. Consider the "Virtual GND" output. Don't look too closely at the I2C bus. The original design is a small double-sided board designed to mount behind a front panel. We will design and build a single sided SMD board with no vias that mounts in a box and connects to the display and encoder by cables.
Finding Eagle and Installing it	https://www.autodesk.com/products/eagle/free-download https://www.autodesk.com/products/eagle/compare	2 layer Eagle is free – board size and other limitations apply – but probably good enough for us. Fabrication services also have very low prices for "small" boards – so it costs a lot more if you need to go larger. 2 layer boards can also be a good choice for prototyping because it is easier to get probes to all parts of the circuit.
Workflow comments		We will select components where we can, create the drawing, layout the PCB, export a BOM, make and assemble the board. The main difference will be the component selection. It is preferable to nail this at the start. SMD sizes – we will mainly use 0805 passives.

		The process is slow to begin with – it gets much faster as you establish a set of standard components. If you have not tried 2 screens, now is a great time to start.
Component Selection (1)	https://www.adafruit.com/product/931 https://www.aliexpress.com/item/5pcs-0-91-inch-128x32-I2C-IIC-Serial-Blue-OLED-LCD-Display-Module-0-91-12832/32808724036.html	Workflow: 1) Open Mouser 2) Find Component 3) Download datasheet 4) Download model 5) Save in a project basket 6) Import model into Eagle library for project 7) You can use generic components for standard passives, but if you are going to buy the components then you may want to check supply at design time.
Component Selection (2) Finding a capacitor	www.mouser.com http://mouser.componentsearchengine.com/pcb-libraries.php https://nz.mouser.com/Cart/	Open Mouser Type in "Capacitor" – approx. 488056 to choose from – 1st WTF moment – which one do we want? We will try to find a .1uF capacitor (C1) 1) Choose Ceramic Capacitors, only 279200 now 2) Choose the value – 6056 left 3) Choose package (0805) 722 4) Choose Voltage (50VDC) – 258 remaining – that's better 5) Apply filters 6) Sort by ascending price 7) Choose your preferred brand **that is in stock** in numbers big enough to indicate it is well used. Mouser part 885012207098

		8) Put some in your basket 9) The following are notes for later – I have done this bit already for this project. See ToneGenerator_Parts.lbr 10) Download the 3D model and save the file 11) Install the library loader 12) Import the model into your library When done the basket might look like Basket_Mar30_0551PM.xls . Finally, export it to Excel – this captures the information you will need to build a BOM later.
Using CAD software to draw and lay out a PCB	https://learn.sparkfun.com/tutorials/using-eagle-schematic/all	1) Start Eagle – 2nd WTF moment when you start eagle – you want to start a new project. 2) Right click is good - After creating the project, create a new schematic. 3) First job – place the components on the drawing. 4) Download the parts libraries from GitHub. 5) Use the library manager to browse to the downloaded files and install them (USE) 6) Add parts to the schematic a. V1 – empty b. V1i – all parts – note that connectors used even if at the end all we want are the solder pads. c. V1ii – lay out parts d. V1iii – join the dots e. V1iiii – all joined up f. V1iv – prettified g. V1v – values added, ready for laying out a board.

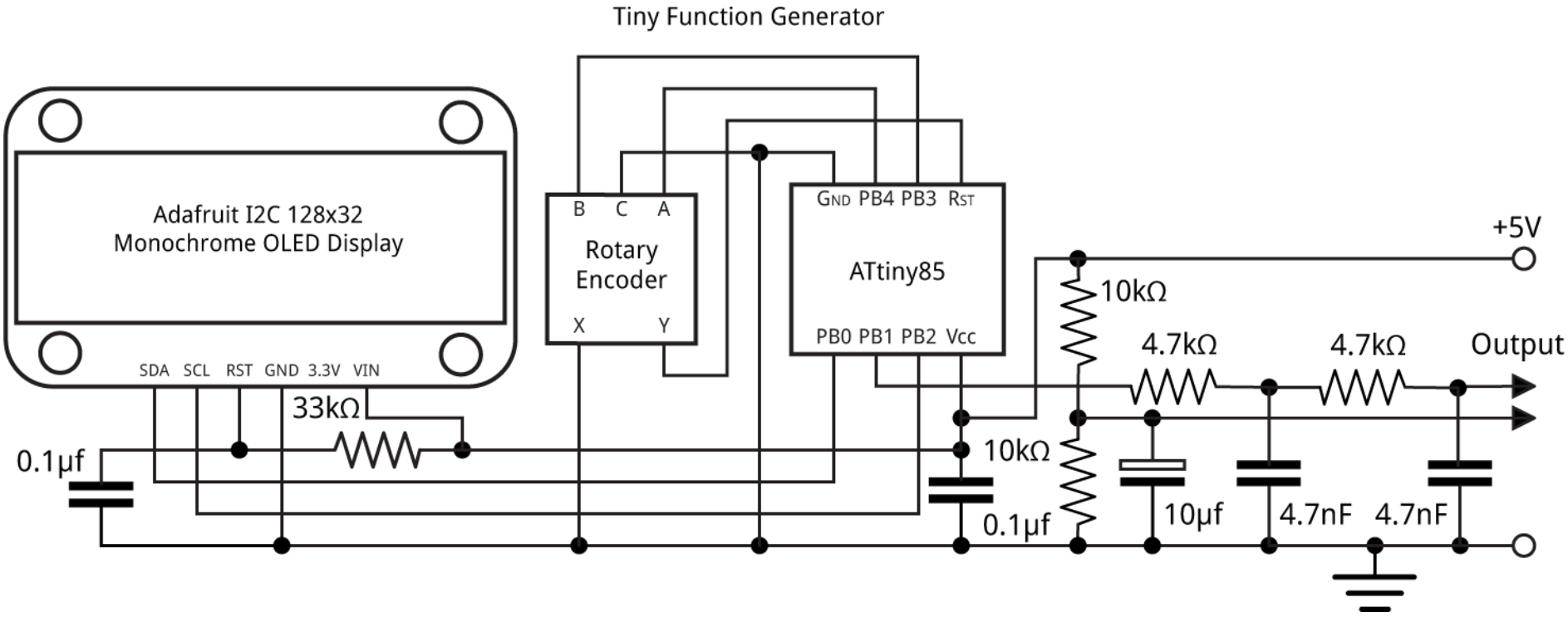
Design Rules	http://support.seeedstudio.com/knowledgebase/articles/447362-fusion-pcb-specification	1) These are the rules that govern how the board can be made. 2) Most typically they relate to the capability of the fabricator or the fabrication process. The answer is different if you want to mill a board, or hand drill it, or it may depend on how you apply the resist. They may also relate to the electrical properties of the circuit – maybe for current carrying or rf performance. 3) Don't use minimum values unless you really must. 6mil spacing and 10 mil traces are very small! 4) default JGE Seeed V6 10mm Trace free.dru is a modified version of the Eagle default files for use with the free version of Eagle (2 layers).
PCB Fabrication		We are going to generate a single sided board with no holes – this is simple to make at home. If I get a board made by a fabricator the cost is a function of area and 2 sides cost the same as single sided. The holes are included as well. Assembly cost vs component cost varies – but I have found it sensible to make a 2 sided board and get the fabricator to place the SMD components. 1) Click SCH/BRD to generate a board. 2) Set the size of the board – note that pin spacing is usually .1 of an inch so the grid is good at 0.05 of an inch. Our board is 1400mil by 1600mil (35.5mm by 40.64mm) 3) Put the holes on the board!!! 2.5mm = 98.425 mil 4) Load the DRU file!!

		a. V1v – Start point, with components available 5) Check out the layers Notes on placing components – at this stage the board is a "rats nest" with components connected by "air wires". Try and place the components so there are as few cross overs as possible. b. V1vi – Components placed – needs ground pour. Ground pour is a polygon with isolate set to 12mil. c. V1vii – Components prettified – ground pour added. Notes on the auto router – it should usually complete more than 80% - if not try a different component placing. Sometimes the autorouter does not like the ground pour on a single sided board – try removing it (ripping it up) if the autorouter is not working properly. Manual routing and connecting up the ground pours is usually obvious – but may not be pretty. The autorouter is limited in the number of vias and corners a route can have. d. V1viii – routing complete
Check what's in the layers now?		1) Check there is nothing on the layers we shouldn't have – e.g. nothing on the bottom layer. 2) Check layer 19 – there should be nothing unrouted. 3) Check the names layers (25,26)

		4) Check the values layers(27,28) 5) The stop layers (29,30) show where the solder mask is not going. 6) The cream layers (31,31) are very similar – they show where the solder paste is going – important when we make solder masks. 7) Drills, holes and milling (44,45,46) can be very important but not in this example because we are making a single layer board. 8) Outlines on layer 51.
Creating Cam Files and using a PCB Fabrication Service		1) Use the template from system examples, third party, Seeed Fusion, 2 layer 2) Check the layers in the Gerber file (the NC instructions for the machines that make it) 3) Export as a zip file (check box at the top) See ToneGeneratorSMD_V1viii_2019-03-31 CAM.zip in the project files.
Prepare the BOM		1) Select the schematic and choose ULP 2) Run bom – probably the first option 3) Save values and .csv to disk 4) Import .csv in to Excell (does not work if you just open it) 5) Example files are ToneGeneratorSMD_V1viii BOM.csv for the export from Eagle, and Eagle BOM 31 March 19.xlsx for the resulting .xls file. 6) Build the BOM for Seeed. See the template Seeed BOM Template 2018.xlsx 7) Review the resources you have: a. Exported Mouser basket (Basket_Mar30_0551PM.xls) b. Template Seeed BOM (Seeed BOM Template 2018.xlsx)

		c. Bom exported from your drawing (Eagle BOM 31 March 19.xlsx) 8) You will use the Seeed template to build a BOM for Seed. You need the component designator and quantity from the bom exported from your drawing (Eagle BOM 31 March19.xlsx) and the Manufacturer's part number and link from the Mouser basket. The link is found by using the Mouser part number to search the Mouser web site. If you used generic components in the diagram you need to find the specific part at this stage. 9) The result will be a Seed BOM that looks like Seeed BOM 31 March 2019i.xlsx
Solder Masks or Stencils	https://www.pololu.com/product/446 http://support.seeedstudio.com/knowledgebase/articles/466664-fusion-pcb-stencil-specification https://www.autodesk.com/products/eagle/blog/getting-started-solder-paste-stencils/ https://community.glowforge.com/t/how-to-create-a-pcb-stencil/17447	1) Most fabricators make them. See Seeed and Polou for examples. 2) Expensive, especially in stainless steel 3) Can make them in mylar with a laser cutter 4) Method is based on exporting the cream layer from Eagle and using it to make an SVG or gerber file for the laser cutter. 5) Details require knowledge of the target laser cutter. See the Glowforge example 6) There is a sample solder mask on GitHub. It is a standard vector graphics file (.svg) ToneGeneratorSMD_V1_SolderMaskplain.svg
Using Surface Mount Components (SMDs) - options and tools		

3D printing - Making the box	https://www.autodesk.com/campaigns/fusion-360-for-hobbyists https://www.youtube.com/watch?v=VVmOtM60VWw https://youtu.be/lBK0UBjVrYM	
Folding the box		
Software overview		
Using the Arduino IDE		
Programming the ATTINY		


Tiny Function Generator
Adafruit I2C 128x32
Monochrome OLED Display
SDA SCL RST GND 3.3V VIN
B C A
Rotary
Encoder
X Y
GND PB4 PB3 RST
ATtiny85
PB0 PB1 PB2 Vcc
+5V
10kΩ
4.7kΩ
4.7kΩ
Output
33kΩ
0.1µf
10kΩ
0.1µf
10µf
4.7nF
4.7nF