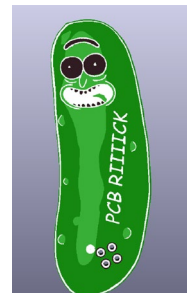
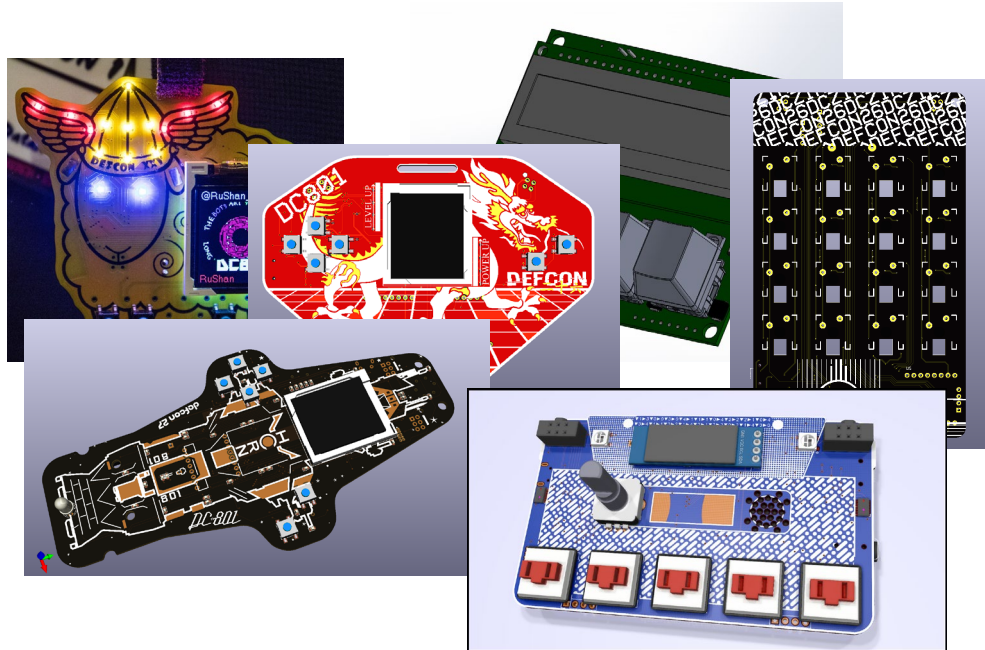


Getting Started with KiCad

Designing the less shitty Shitty Add On

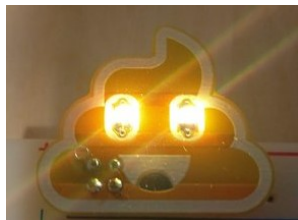
Intro

- Who am I and how did I get in here?



SAOs

- SAO – Shitty Add On
- Designed by Brian Benchoff on a whim for DC26
- Small boards that plug into a badge for power



DC26 Shitty Addon Connector!

VCC SDA
GND SCL

Basically, it's an I2C bus, 3V3 and GND. We'll figure out I2C addresses and commands shortly. Is it good engineering? No, it's a shitty add-on.

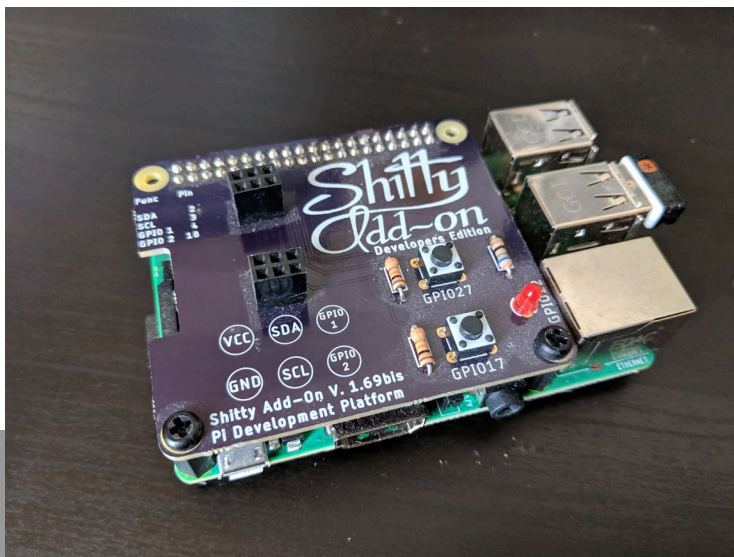
Circle indicates VCC

'Master' badges use female headers, shitty addons use male pins. 0.1". Don't know metric /shrug

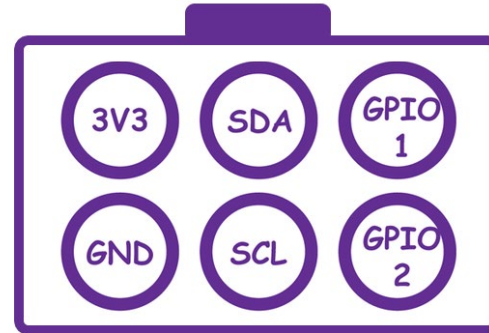


SAOs

- New for Defcon 27 – SAO v1.69bis
- Backwards compatible, still as shitty
- Now with more pins, better retention



Shitty Add-on V. 1.69bis Pinout



Badge
(Top View)



SAO
(Bottom View)

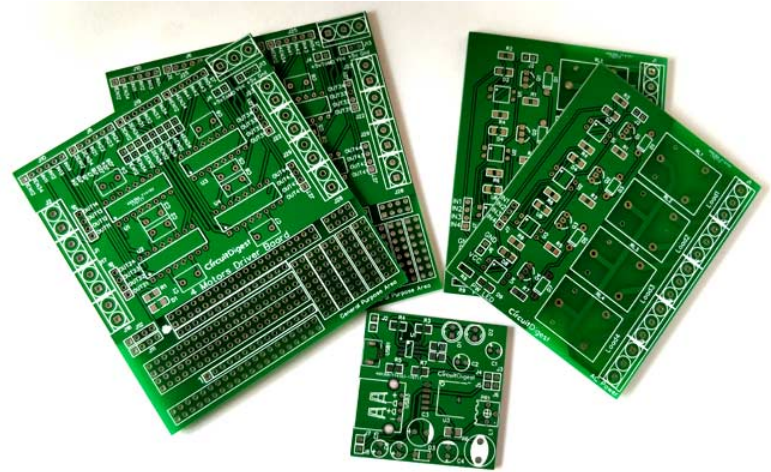


Red Circle is 3V3



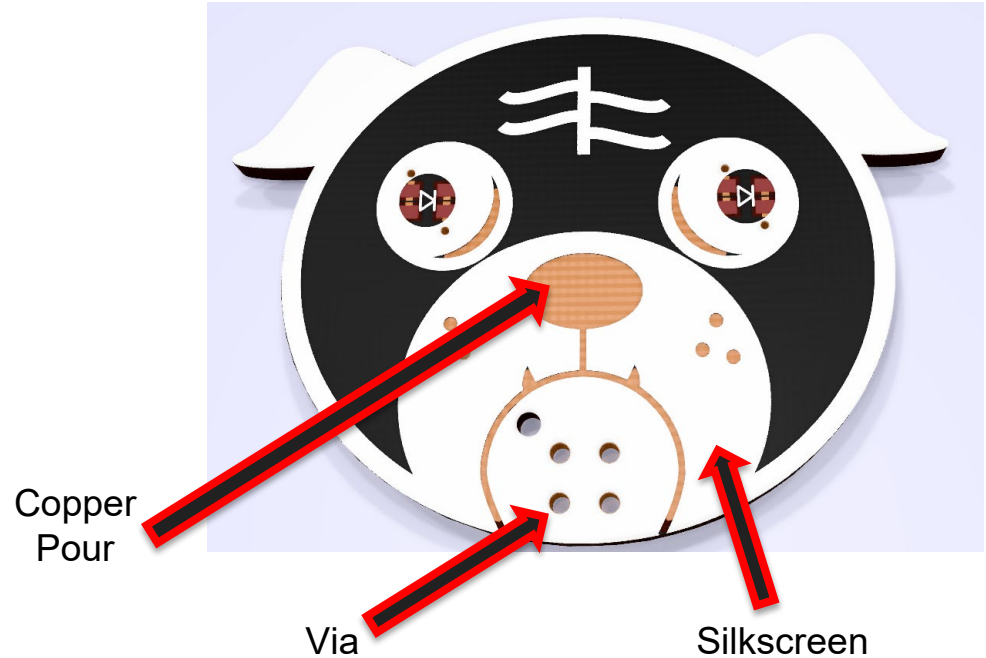
Printed Circuit Boards

- Printed Circuit Board – PCB
- Consists of layers of insulation, copper, and printed graphics
- Board houses like to specify board dimensions in millimeters, but trace widths in mils (0.001 inches)



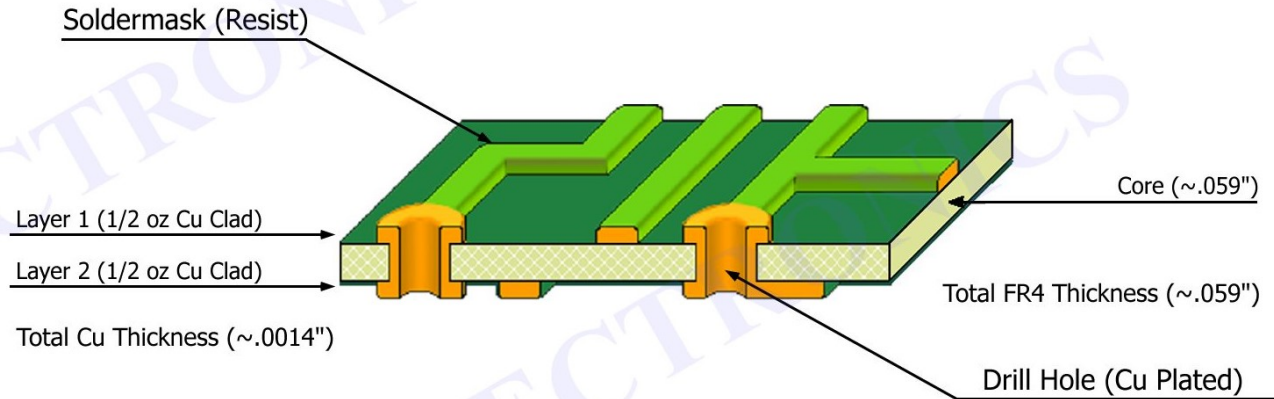
Printed Circuit Boards

- Vias – holes, may be plated
- Traces – thin copper lines, ‘wires’
- Pours – Large areas of copper
- Silkscreen – Artwork or labels



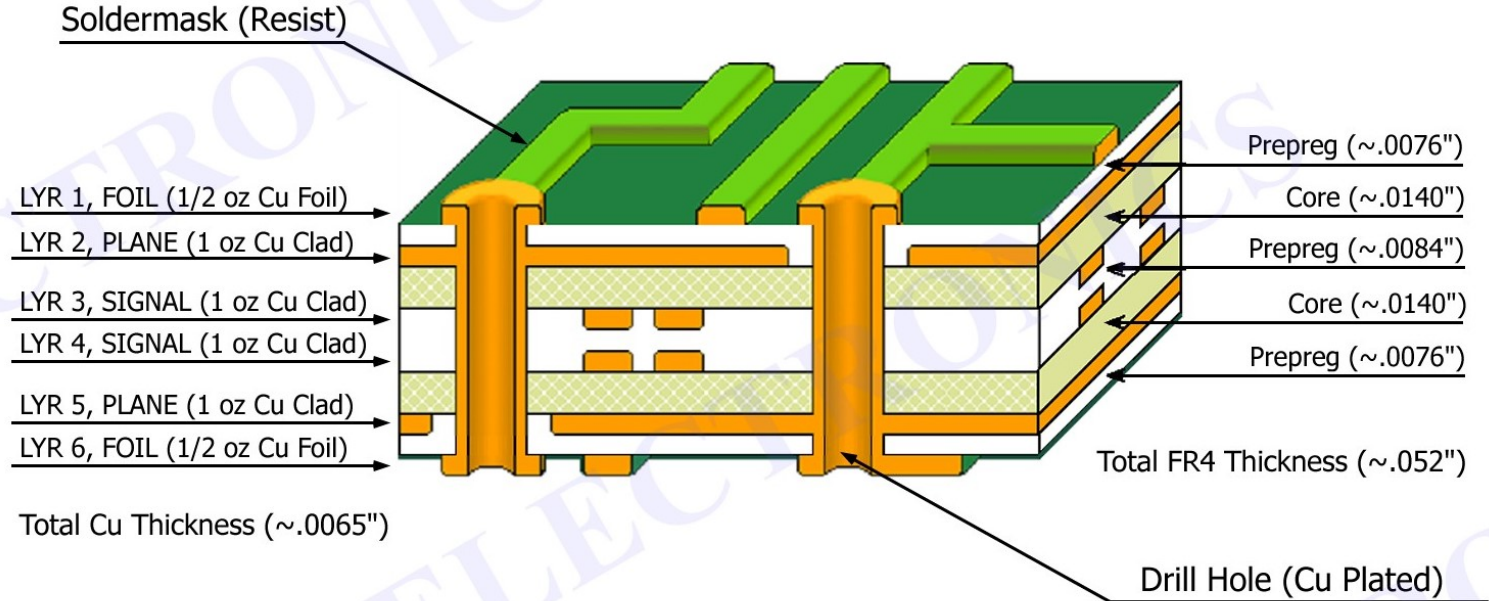
Printed Circuit Boards

- Typical 2 layer stackup



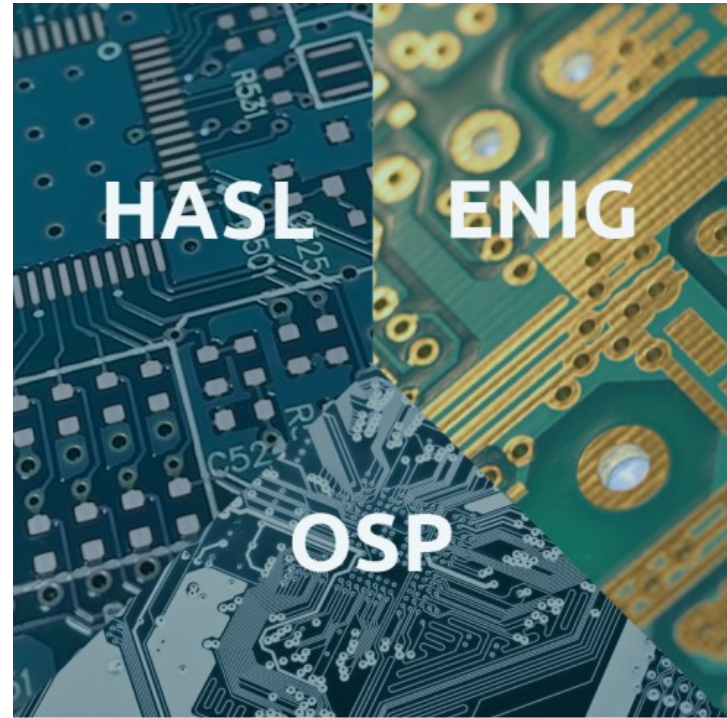
Printed Circuit Boards

- Keep adding layers!



Printed Circuit Boards

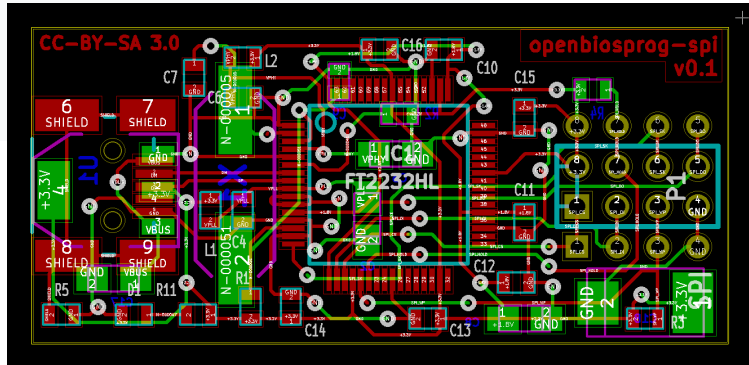
- Copper Finish - OSP vs HASL vs ENIG
- Organic Solderability Preservative
Usually a gold color – handling will stain it
- Hot Air Solder Leveling
Silver finish – it's solder dipped
- Electroless Nickel Immersion Gold
Gold color – handles well
- HASL usually default – other options are more \$\$



Printed Circuit Boards

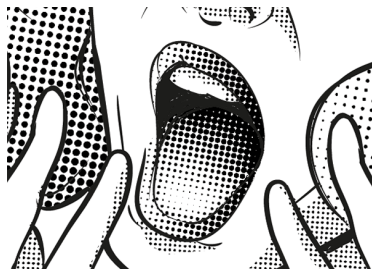
- Gerbers
ASCII format for describing board layout
Released in 1980!

```
G04 Short version a file taken from the Example Job 1, created by Filip Vermeire, Ucamco*
%TF.FileFunction,Copper,Bot,L4*%
%TF.FilePolarity,Positive*%
%TF.Part,Single*%
%FSLAX36Y36*%
%MOMM*%
%TA.AperFunction,Conductor*%
%ADD10C,0.15000*%
%TA.AperFunction,ViaPad*%
%ADD11C,0.75000*%
%TA.AperFunction,ComponentPad*%
%ADD12C,1.60000*%
%ADD13C,1.70000*%
```



Artwork for PCBs

- Design with PCB limitations in mind
- Silkscreen, Soldermask, Copper and bare PCB are your color palette
- Tiny details will get lost
- No gradients – try using halftones



OMG Halftones!

Bad

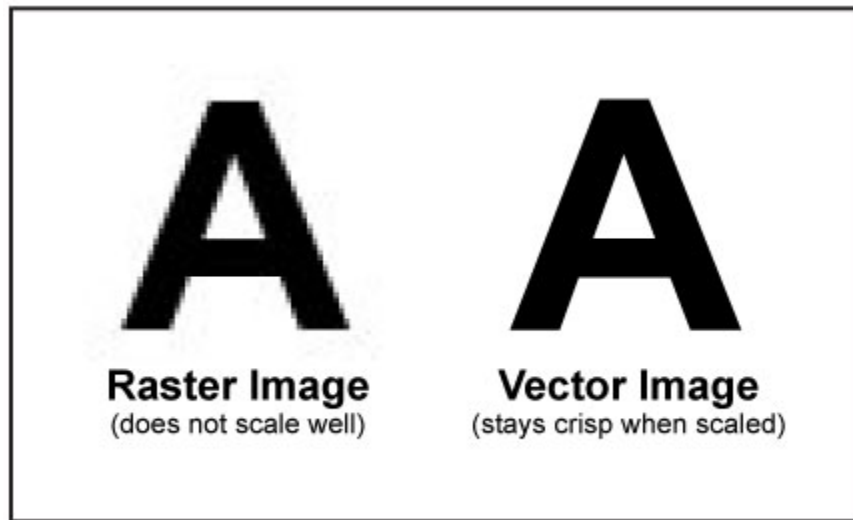


Good



Artwork for PCBs

- Vector vs Bitmap
Try to design with vector
- Suggested Software
Inkscape, Illustrator

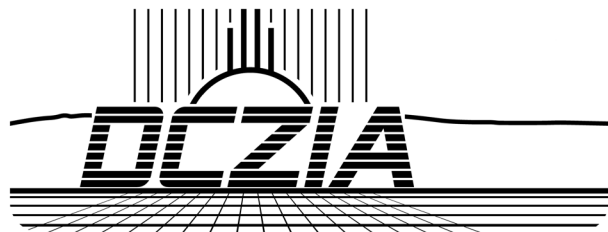


Artwork for PCBs

- Scaling and exporting

Design your art at 1:1 scale to the real deal

Export your image at high DPI – I use 1200DPI



Source

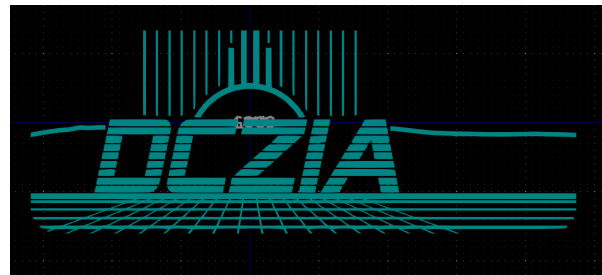
72DPI



300DPI



1200DPI

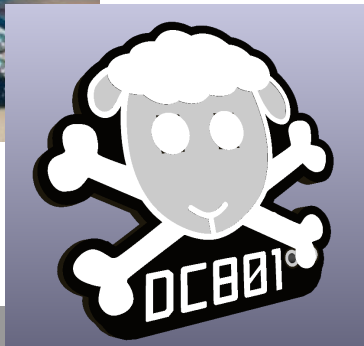
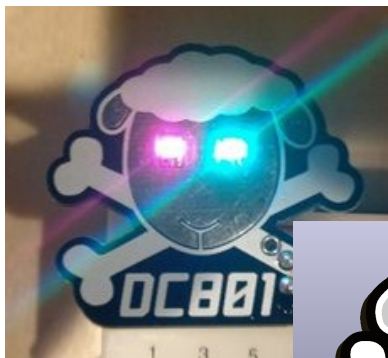


Artwork for PCBs

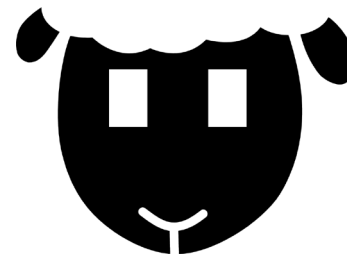
- Simplify Layers

Export each layer as a different image

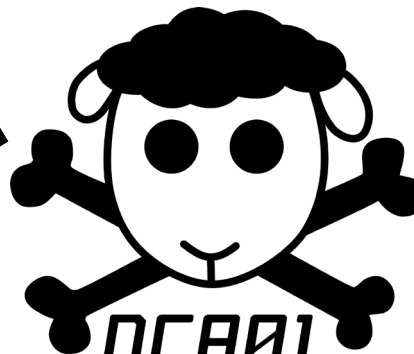
Black and white only



Design File



Bare Copper Layer



Silkscreen Layer



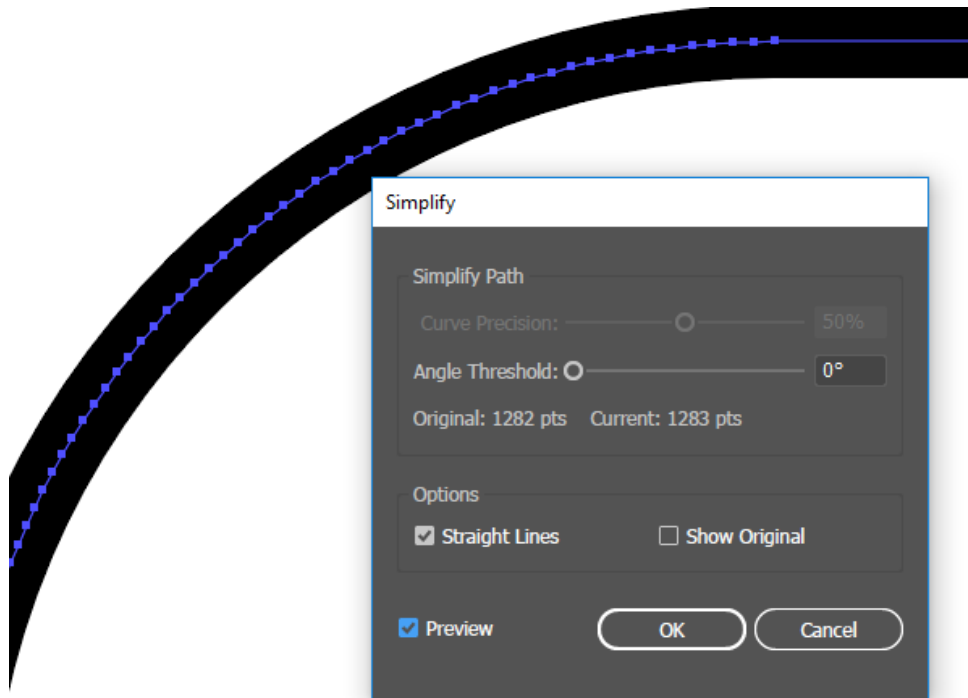
Artwork for PCBs

- Edge cuts and outlines

Designing at 1:1 means you can export your outline as a DXF

KiCad DXF import cannot import curved lines!

Add anchor points, then simplify paths to straight lines to approximate a curve



Artwork for PCBs

- Hands on!



KiCad

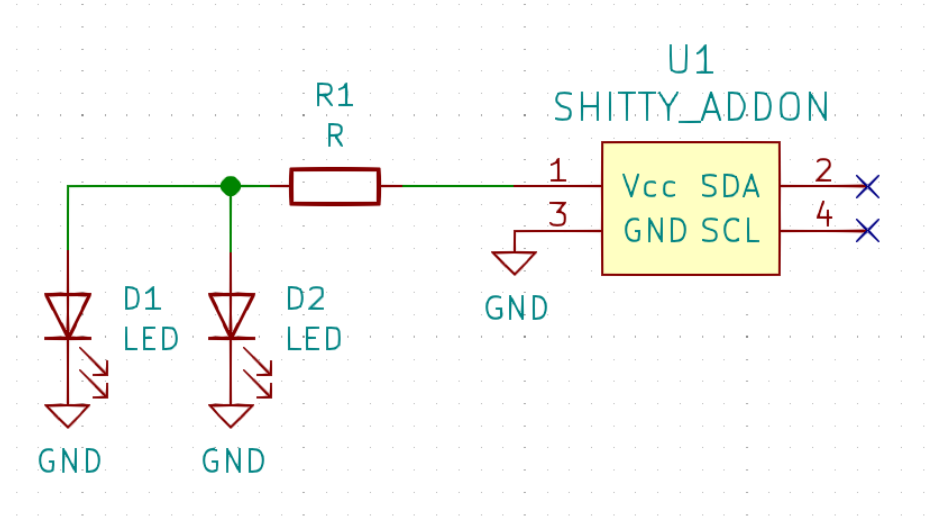


- What is KiCad?
- Started in 1992
- Open source, lots of development
- KiCad's first con was held in Chicago last month
- KiCad covers both Schematic Capture and Board Layout



KiCad - Schematic

- Schematic capture
- Start here to design your circuit
- Keep it neat for readability



KiCad - Schematic

- Schematic footprint association
For each part, assign a footprint
- Save, then generate a netlist



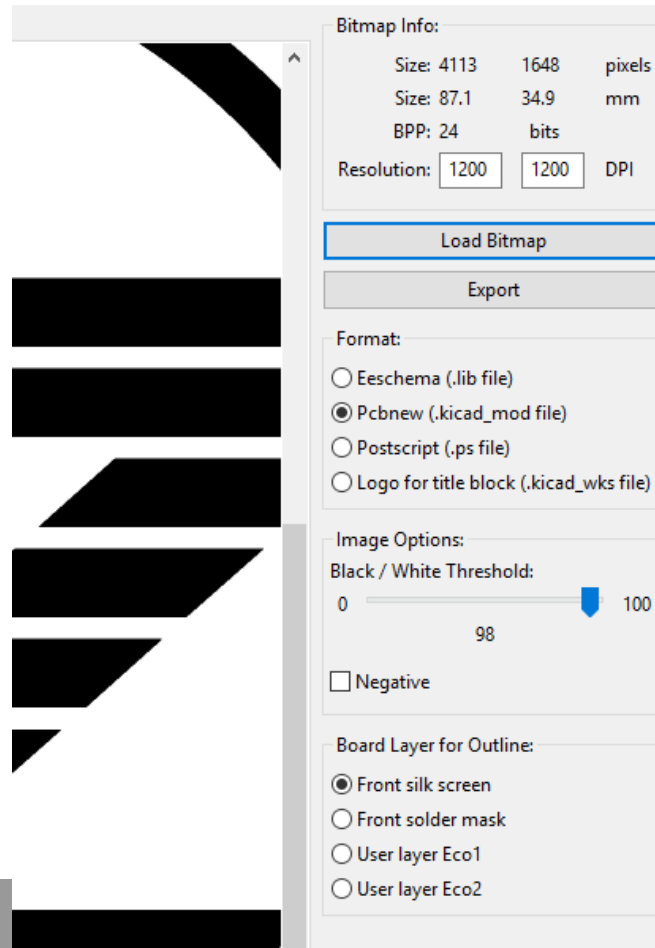
KiCad - Schematic

- Hands on!



KiCad - Import

- Importing artwork with Bitmap to Component Convertor
- If you designed to scale, this is an easy step
- The convertor converts a bitmap to a vector that KiCad can use
- We can move the layers around later



KiCad - Import

- After conversion, you can use the footprint editor to move layers around
- Bare copper is copper without a soldermask.

In KiCad, this means adding a shape to the soldermask layer

Soldermask layer is a negative layer!

Just copy and paste in place the copper polygon you want to expose and move to soldermask



Without soldermask Hole



With soldermask Hole



KiCad - Import

- Hands on!



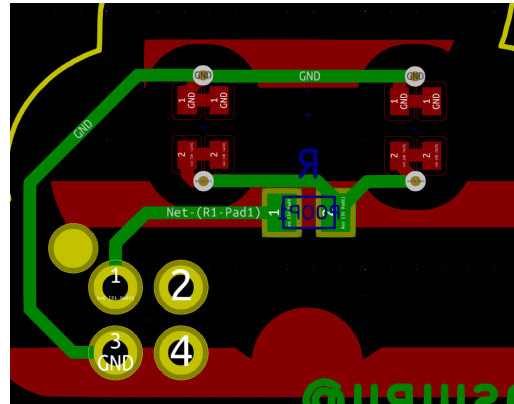
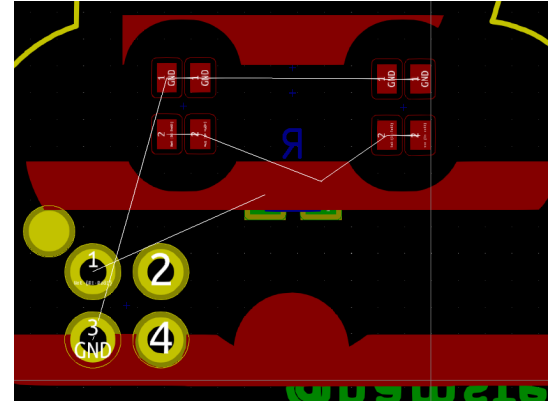
KiCad - PCB

- Create a new board layout and open your netlist
- Place your imported footprints
- Import your outline
- Place your component footprints



KiCad - PCB

- Solve the rats by laying down traces
- Note that most board houses have a limit of 6mils trace/space unless you want to pay more



KiCad - PCB

- RUN DRC
- Seriously
- Do it
- Design Rules Check



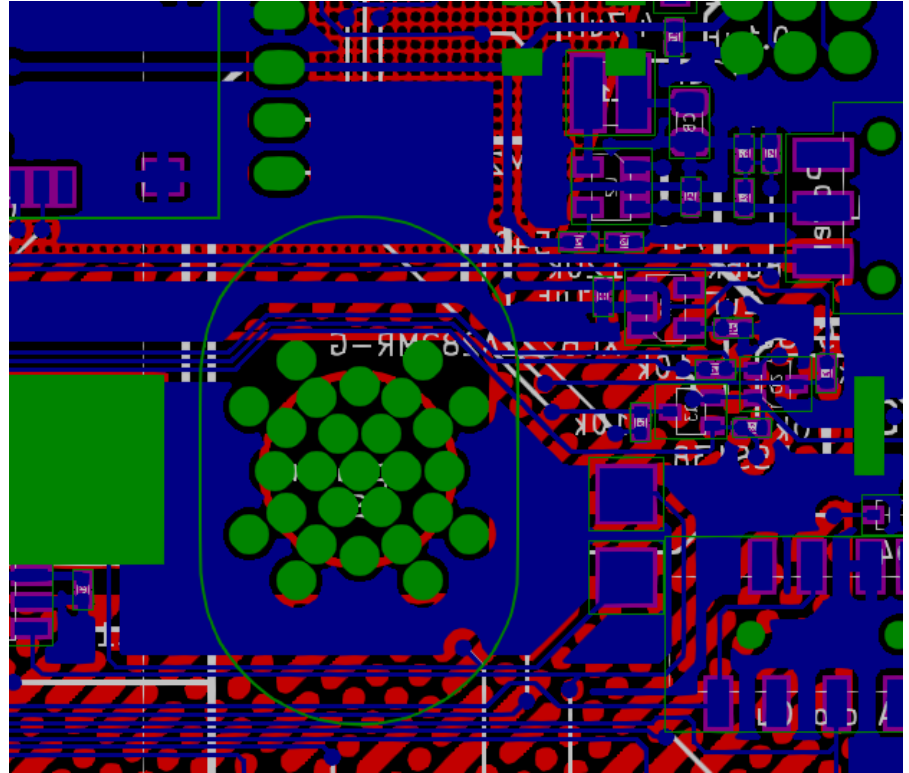
KiCad - PCB

- Hands on!



Bringing it all together

- Export Gerbers
- At the very least, you need top and bottom copper, silk, and soldermask
- Check with your board house for specific requirements
- Check the result in gerbv or other Gerber viewer before sending out



Bringing it all together

- Hands on!



Ordering boards

- Typical options
- Less than 100x100mm is cheap
- Play with the options
- Sometimes an extra 100 boards is just a few bucks more

Base Material	<input checked="" type="radio"/> FR-4 TG130	<input type="radio"/> Aluminum	<input type="radio"/> Flexible Boards							
No. of Layers	<input type="radio"/> 1 layer	<input checked="" type="radio"/> 2 layers	<input type="radio"/> 4 layers	<input type="radio"/> 6 layers						
PCB Dimensions	<input type="text" value="100"/>	*	<input type="text" value="100"/>	* Units in mm						
PCB Quantity	<input type="text" value="10"/>	▼								
No. of Different Designs	<input checked="" type="radio"/> 1	<input type="radio"/> 2	<input type="radio"/> 3	<input type="radio"/> 4	<input type="radio"/> 5	<input type="radio"/> 6	<input type="radio"/> 7	<input type="radio"/> 8	<input type="radio"/> 9	Example
PCB Thickness	<input type="radio"/> 0.60	<input type="radio"/> 0.80	<input type="radio"/> 1.00	<input type="radio"/> 1.20	<input checked="" type="radio"/> 1.60	<input type="radio"/> 2.00	<input type="radio"/> 2.50	<input type="radio"/> 3.00	* Units in mm	
PCB Color	<input checked="" type="radio"/> Green	<input type="radio"/> Red	<input type="radio"/> Yellow	<input type="radio"/> Blue	<input type="radio"/> White	<input type="radio"/> Black				
Surface Finish	<input checked="" type="radio"/> HASL	<input type="radio"/> HASL Lead Free	<input type="radio"/> ENIG	<input type="radio"/> Hard Gold						
Minimum Solder Mask Dam	<input type="radio"/> 0.1mm↑	<input checked="" type="radio"/> 0.4mm↑								
Copper Weight	<input checked="" type="radio"/> 1oz.	<input type="radio"/> 2oz.	<input type="radio"/> 3oz.							
Minimum Drill Hole Size	<input type="radio"/> 0.2mm	<input type="radio"/> 0.25mm	<input checked="" type="radio"/> 0.3mm							
Trace Width / Spacing	<input type="radio"/> 4/4 mil	<input type="radio"/> 5/5 mil	<input checked="" type="radio"/> 6/6 mil							
Blind or Buried Vias	<input type="radio"/> Yes	<input checked="" type="radio"/> No								



Ordering boards

- China board houses are cheap, but you have to wait for shipping
Quality is pretty darn good
Delay one day! – due to time difference, simple questions can cause delays
- OSH Park is a good option in the States for a reasonable price
But I hope you like purple...
- Fab Assembly can get pricey – consider doing it by hand for simple designs



Sourcing Parts

- Aliexpress is a good option if you have time and are willing to risk it
- Choose Amazon if you want the same aliexpress stuff with a markup and Prime shipping
- Digikey, Mouser are tried and true
- Arrow is a strange option, they usually have free overnight shipping and most parts
- If the fab is going to assemble it, let them source parts where possible



Questions?

Snide Comments?

@hamster

