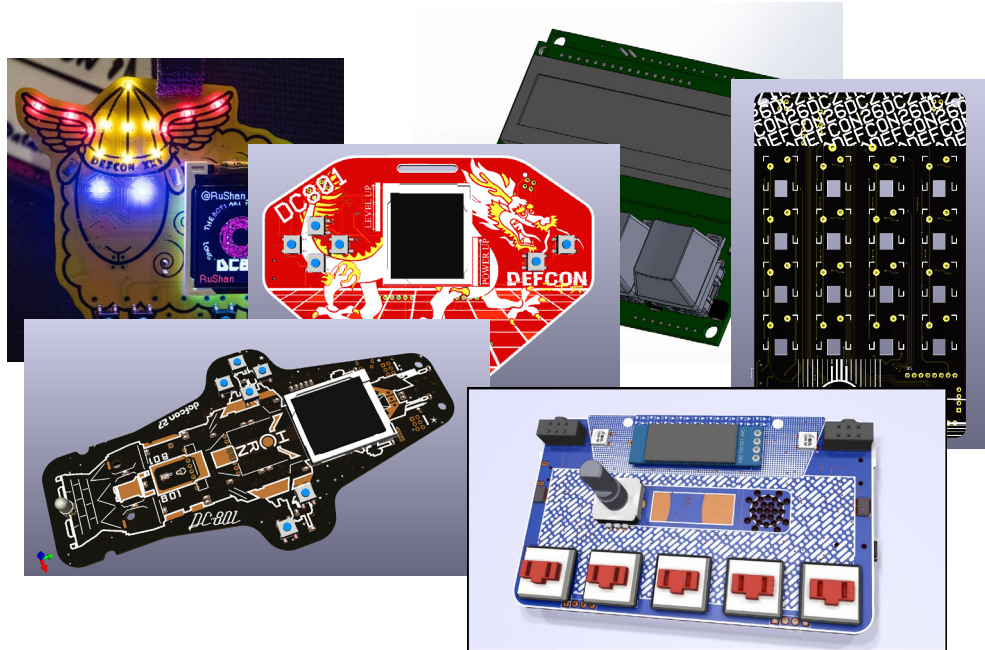# Getting Started with KiCad

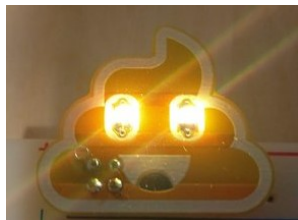Designing the less shitty Shitty Add On

# Intro

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

# SAOs

- SAO – Shitty Add On

- Spec by Brian Benchoff on a whim for DC26

- Small boards that plug into a badge for power





DC26 Shitty Addon Connector!

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.
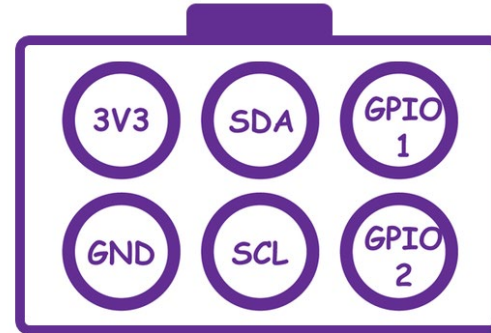
Circle indicates VCC

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

# SAOs

- New for Defcon 27 – SAO v1.69bis

- Backwards compatible, still as shitty

- Now with more pins, better retention

- 2 new GPIO

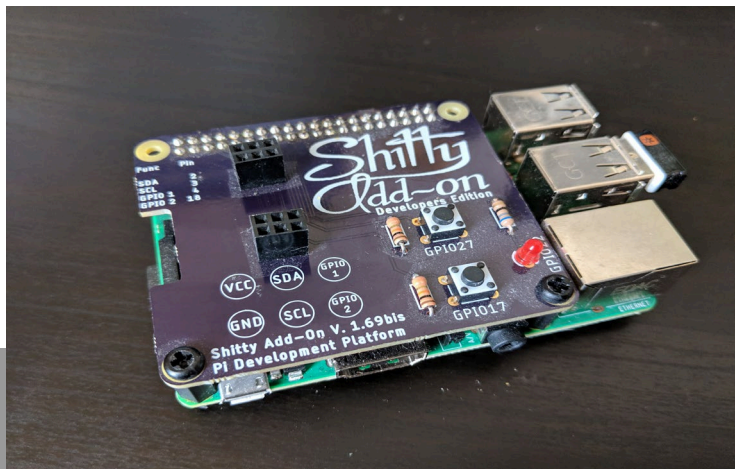## Shitty Add-on V. 1.69bis Pinout

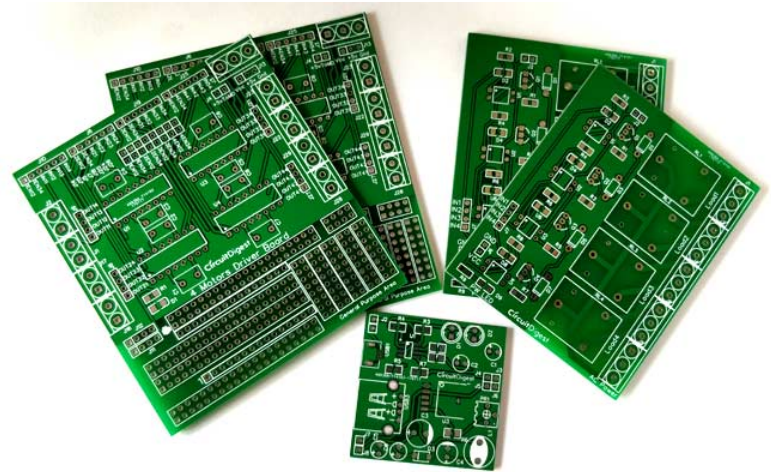| 3V3 | SDA | GPIO 1 |
| GND | SCL | GPIO 2 |

**Badge (Top View)**

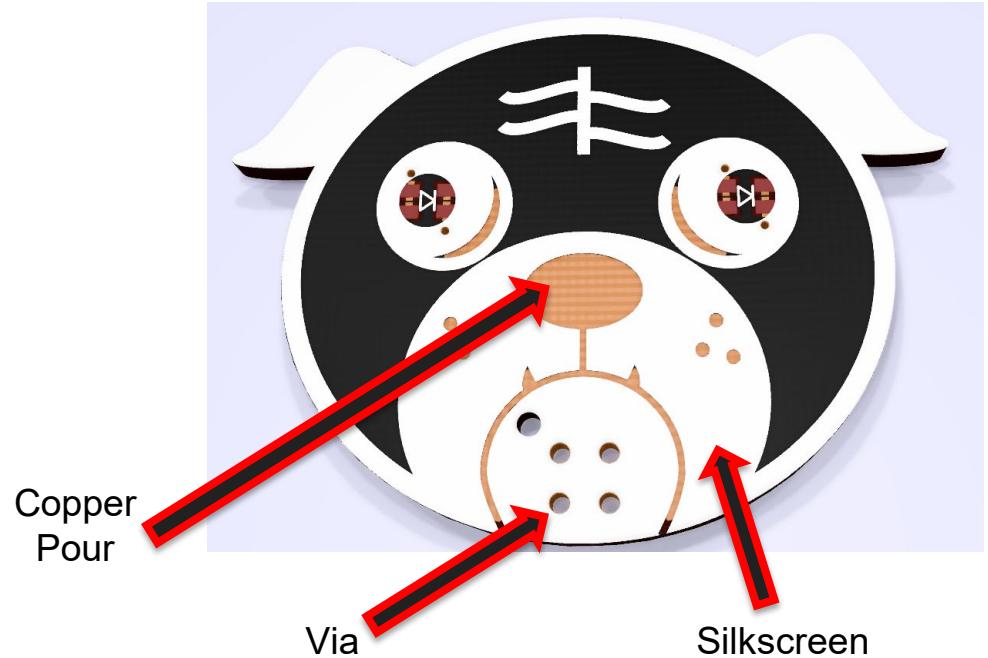**SAO (Bottom View)**

**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
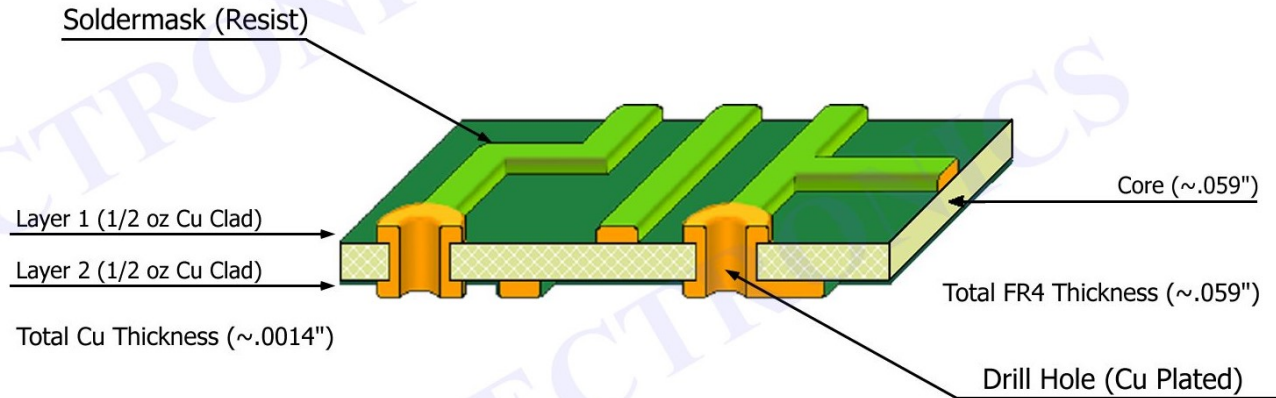  mils (0.001 inches).  Yay units.

# Printed Circuit Boards

- Vias – holes, may be through plated

- Traces – thin copper lines, 'wires'

- Pours – Large areas of copper
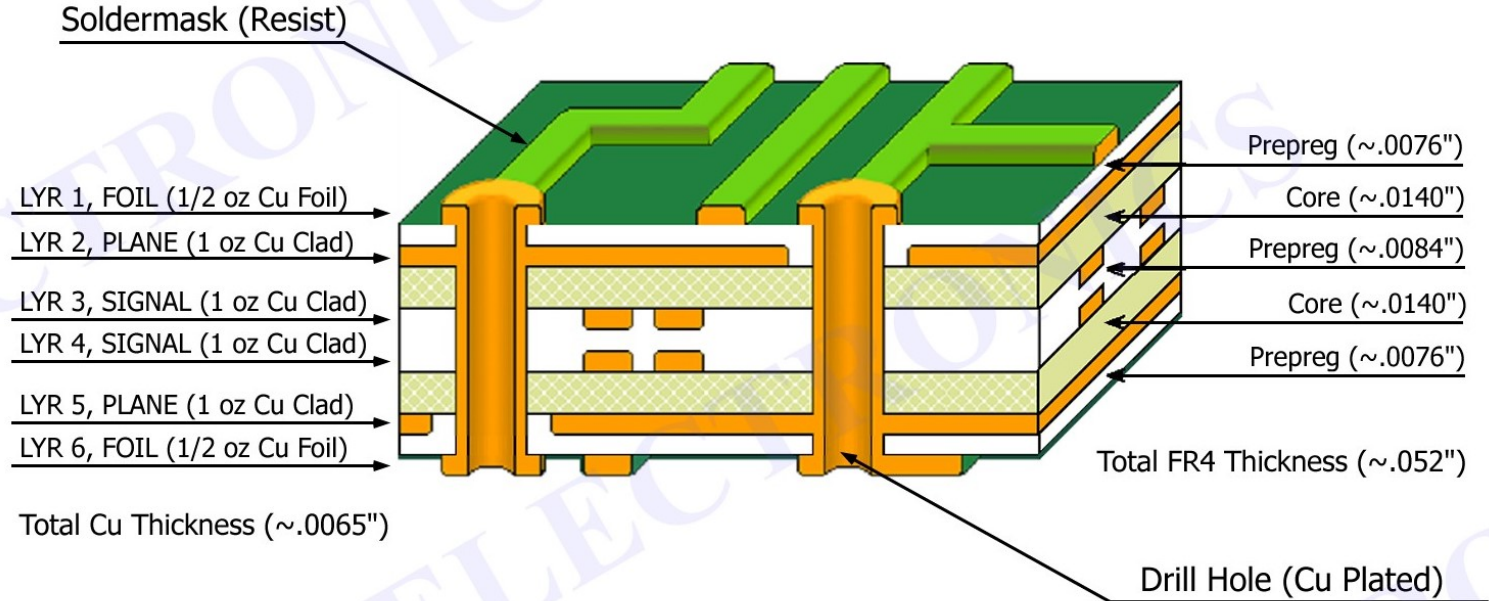
- Silkscreen – Artwork or labels

Copper Pour

Via

Silkscreen

# Printed Circuit Boards

- Typical 2 layer stackup

Soldermask (Resist)

Core (~.059")

Layer 1 (1/2 oz Cu Clad)

Layer 2 (1/2 oz Cu Clad)

Total FR4 Thickness (~.059")

Total Cu Thickness (~.0014")

Drill Hole (Cu Plated)

# Printed Circuit Boards

- Keep adding layers!



Soldermask (Resist)

LYR 1, FOIL (1/2 oz Cu Foil)
LYR 2, PLANE (1 oz Cu Clad)
LYR 3, SIGNAL (1 oz Cu Clad)
LYR 4, SIGNAL (1 oz Cu Clad)
LYR 5, PLANE (1 oz Cu Clad)
LYR 6, FOIL (1/2 oz Cu Foil)

Total Cu Thickness (~.0065")

Prepreg (~.0076")
Core (~.0140")
Prepreg (~.0084")
Core (~.0140")
Prepreg (~.0076")

Total FR4 Thickness (~.052")

Drill Hole (Cu Plated)
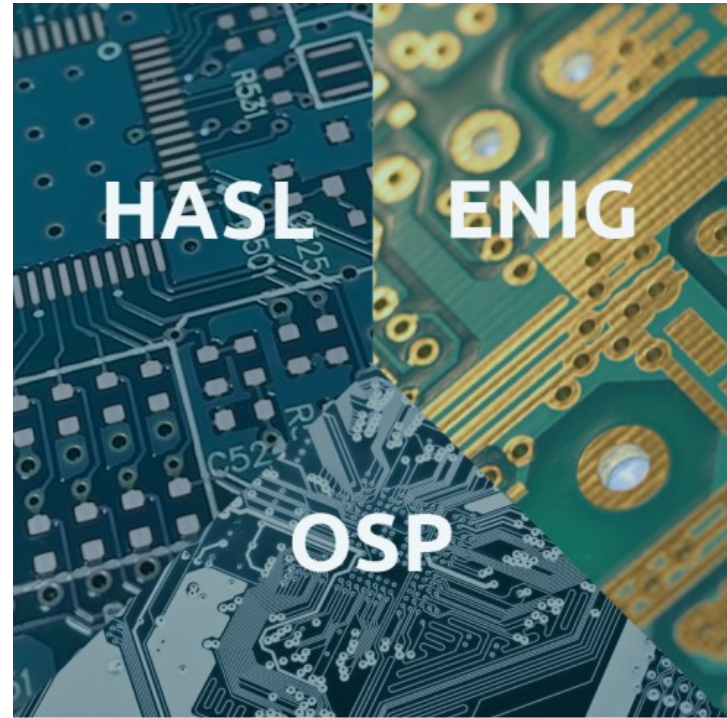
# Printed Circuit Boards

- 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 – resists staining via handling

- 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*%
```
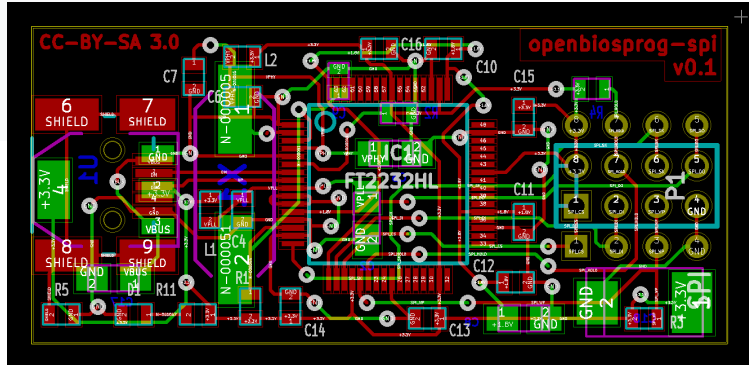
Not baby food



Gerber.

# Artwork for PCBs

**Bad**

- Design with PCB limitations in mind

- Silkscreen, Soldermask, Copper and bare PCB are your color palette

- Tiny details will get lost

**Good**

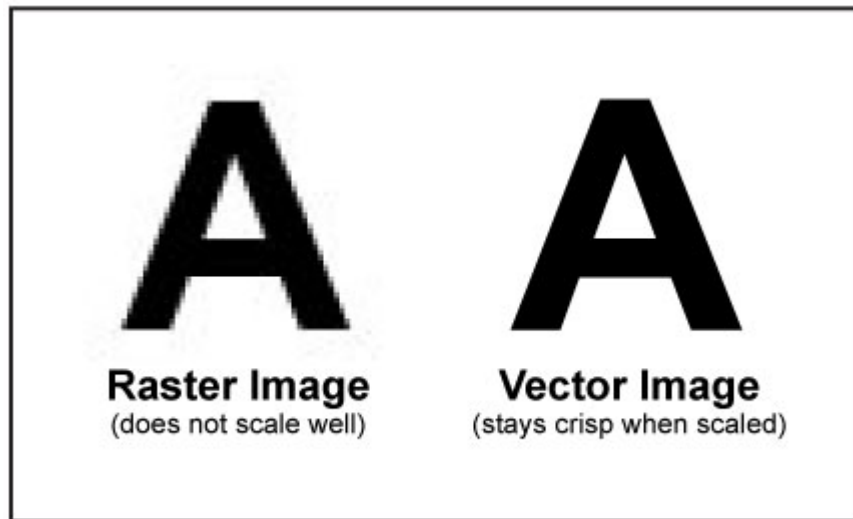- No gradients – try using halftones

OMG Halftones!

# Artwork for PCBs

- Vector vs Bitmap
  Try to design with vector

- Suggested Software
  Inkscape, Illustrator



**Raster Image**
(does not scale well)

**Vector Image**
(stays crisp when scaled)

# Halftones

- Halftones use dots of increasing size to approximate a gradient

- Can add a lot of detail for 'cheap'
  Rendering the 3D in KiCad is gonna suck...
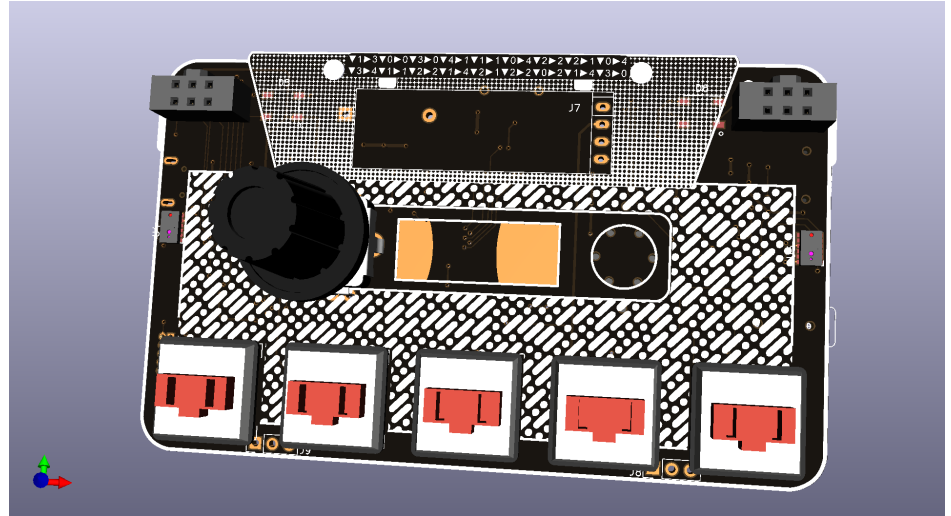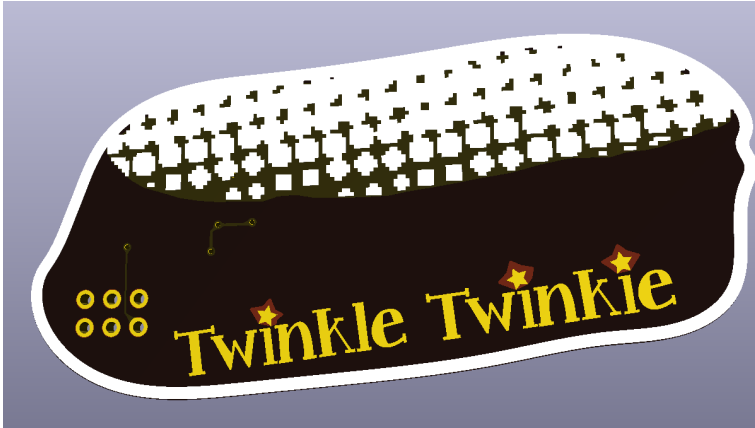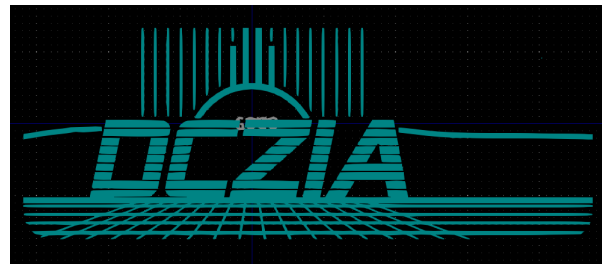
# Artwork for PCBs

- Scaling and exporting

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

  Export your image at high DPI – I use 1200DPI

  KiCad's bitmap importer looses details at low DPI

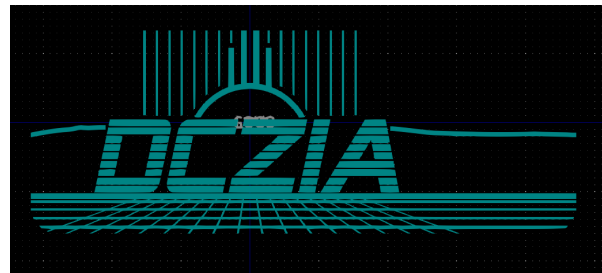- Upcoming KiCad modules might be able to import SVGs natively
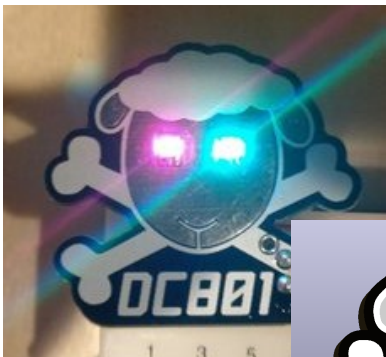

Source
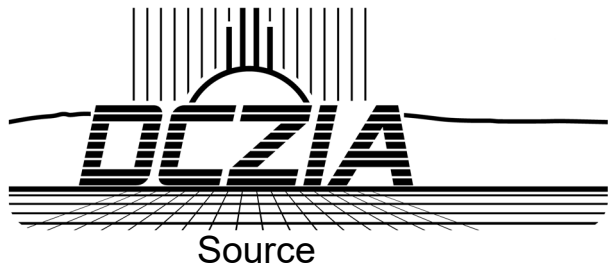

72DPI
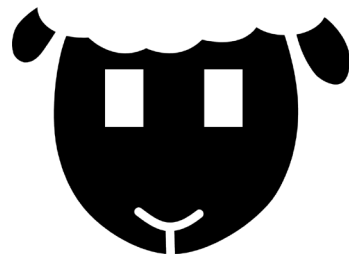

300DPI


1200DPI

# Artwork for PCBs

- Simplfy Layers

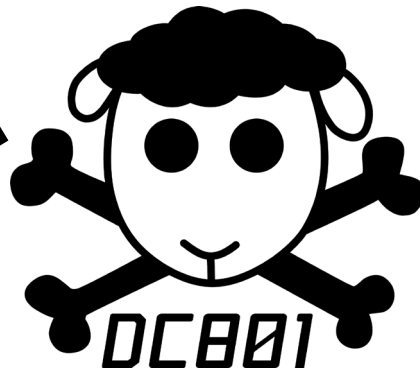  Export each layer as a different image

  Black and white only







Design File



Bare Copper Layer
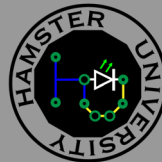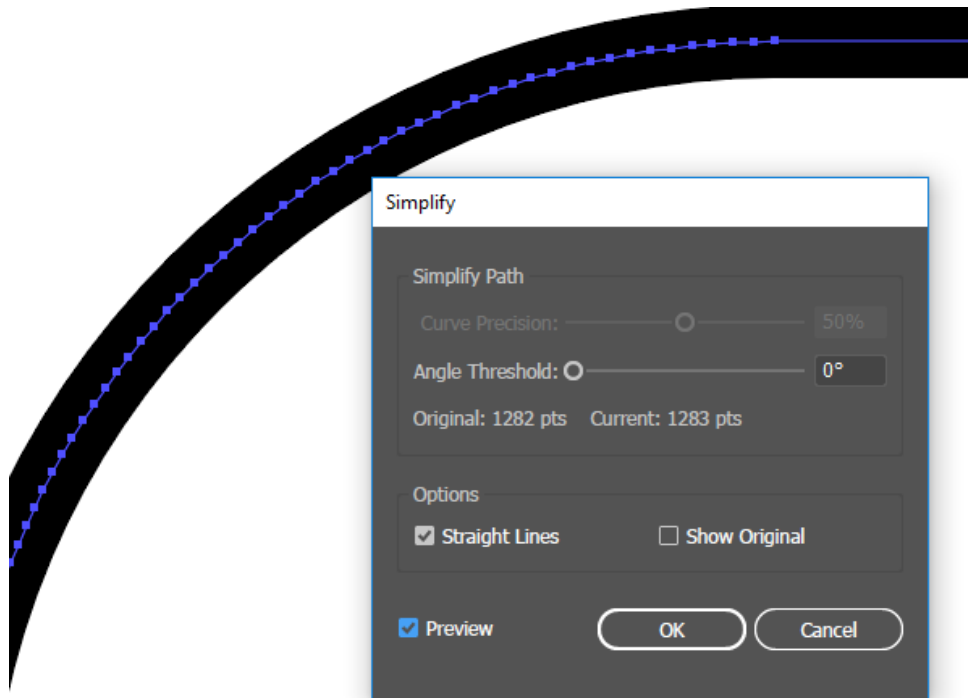


Silkscreen Layer

# 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 simplfy paths to straight lines to approximate a curve



**Simplify**

Simplify Path

Curve Precision: 50%

Angle Threshold: 0°

Original: 1282 pts    Current: 1283 pts

Options

☑ Straight Lines        ☐ Show Original

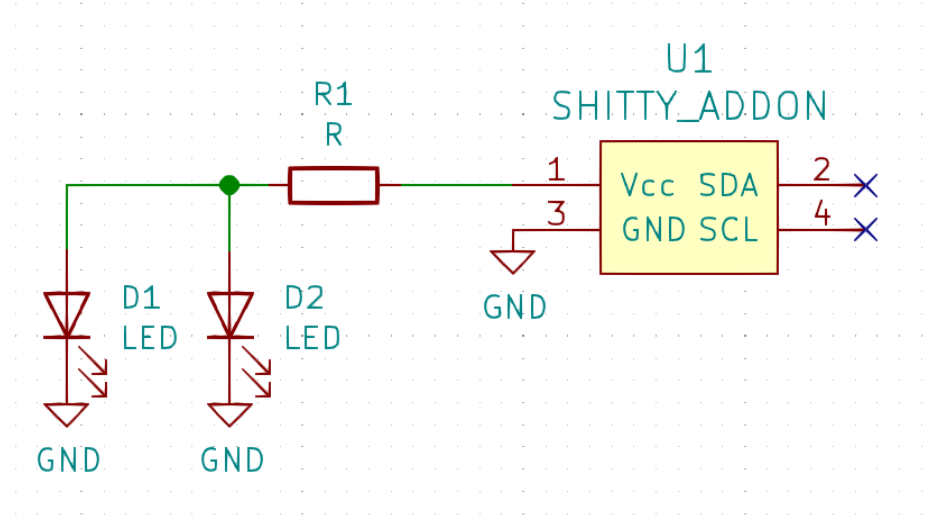☑ Preview        OK        Cancel

# KiCad

- What is KiCad?

- Started in 1992

- Open source, lots of development

- KiCad's first con was held in Chicago earlier this year

- KiCad covers both Schematic Capture and Board Layout

- 3D preview is great for instant visual feedback

# KiCad - Schematic

- Schematic capture

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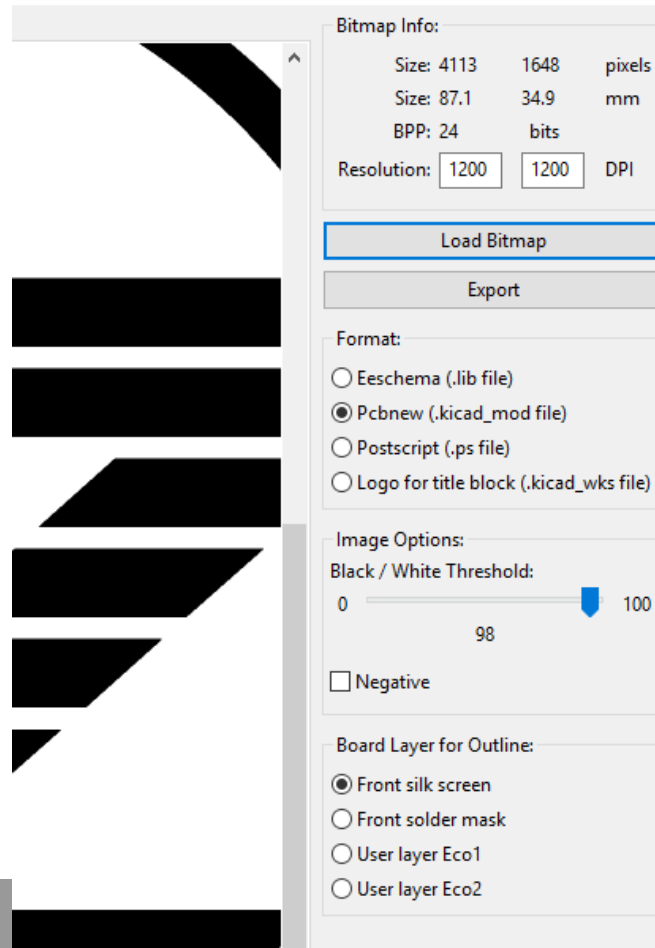
- Keep it neat for readability

# KiCad - Schematic

- Schematic footprint association

  For each part, assign a footprint

- Save, then apply changes in the board editor

# KiCad - Import

- Import 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

**Bitmap Info:**

| | | | |
|---|---|---|---|
| Size: | 4113 | 1648 | pixels |
| Size: | 87.1 | 34.9 | mm |
| BPP: | 24 | | bits |

Resolution: 1200 | 1200 | DPI

Load Bitmap

Export

**Format:**
- ○ Eeschema (.lib file)
- ● Pcbnew (.kicad_mod file)
- ○ Postscript (.ps file)
- ○ Logo for title block (.kicad_wks file)

**Image Options:**
Black / White Threshold:
0 —————————— 100
98

☐ Negative

**Board Layer for Outline:**
- ● Front silk screen
- ○ Front solder mask
- ○ User layer Eco1
- ○ User layer Eco2

# 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
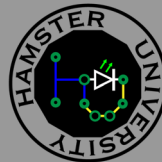  Soldermask layer is a negative layer!
  Just copy and paste in place the copper polygon you want to expose and move to soldermask



Without soldermask Hole



With soldermask Hole

# KiCad - PCB

- Create a new board layout and import your parts

- 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

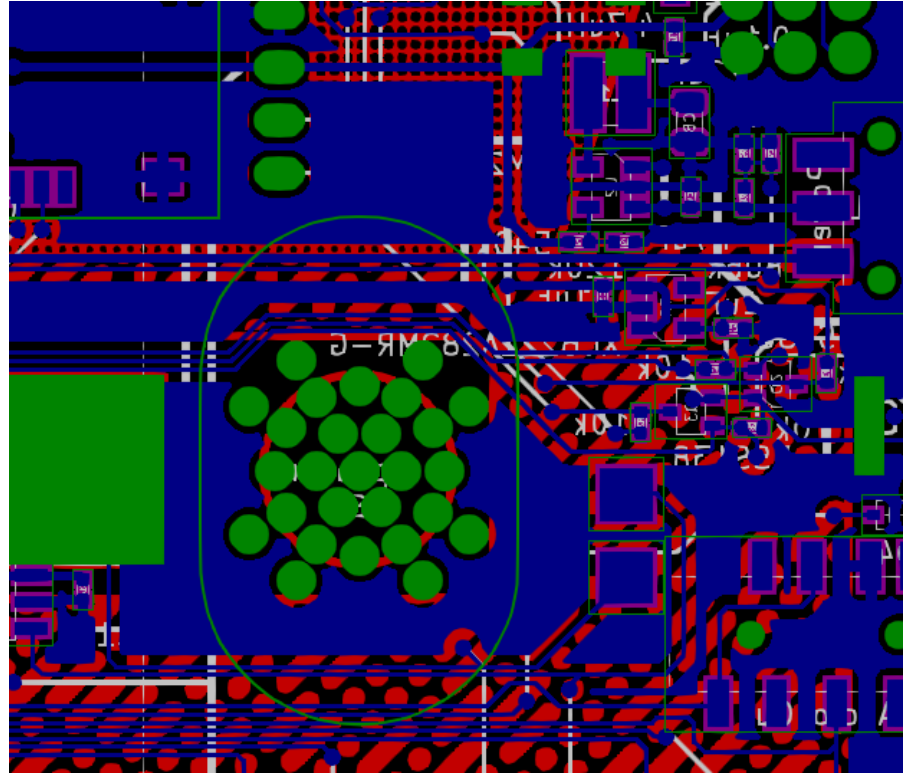- Beware that copper pours in your imported art won't be detected by DRC – don't short things out!
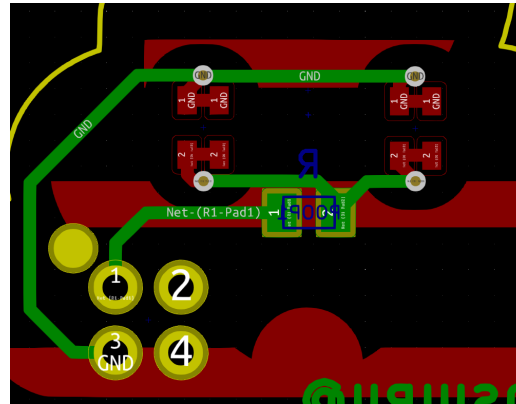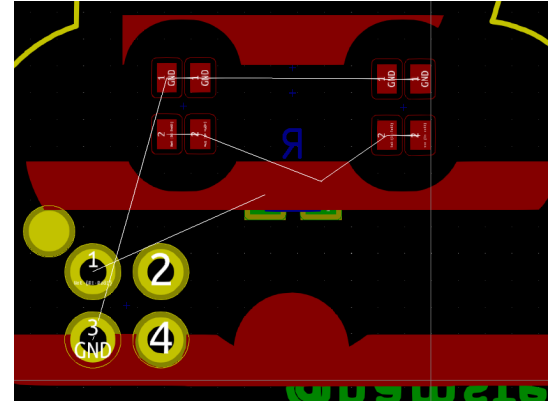
# KiCad - PCB

- RUN DRC

- Seriously

- Do it

- Design Rules Check

# Bringing it all together

- Export Gerbers

- Check with your board house for specific requirements, many have howtos

- Check the result in gerbv or other Gerber viewer before sending out

# Panels!

- Open source tools exist to place multiple boards into a single panel

- Can be much cheaper to run

- If you stencil solderpaste, you can stencil a whole panel at once
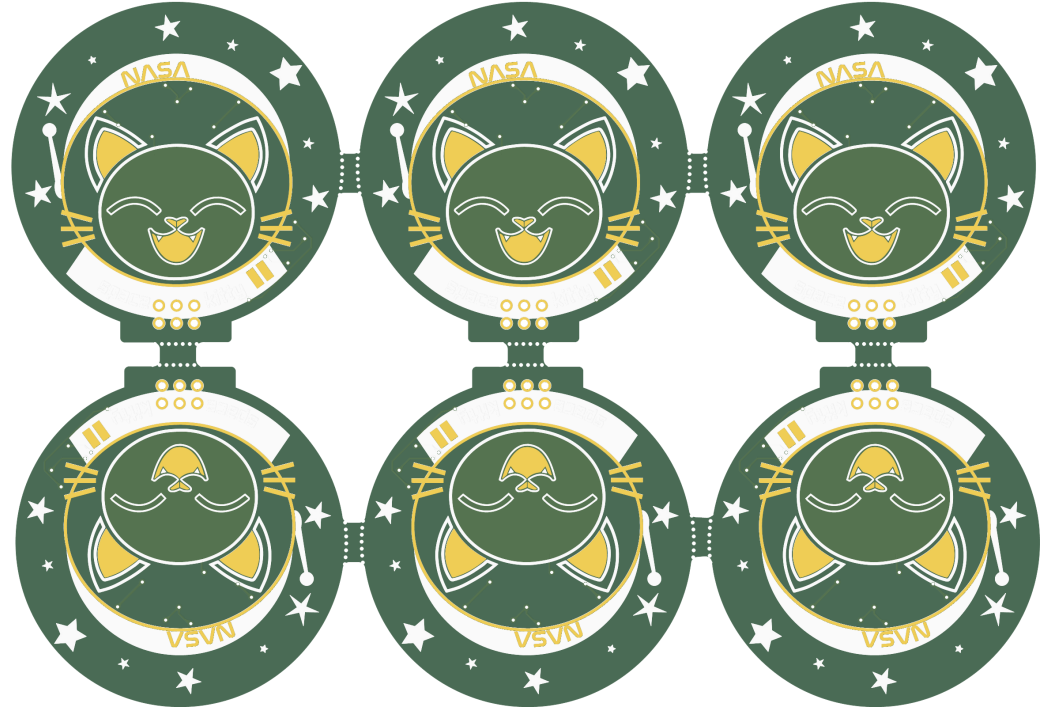
# Ordering boards

- Typical options

- Less than 100x100mm is cheap

- Play with the options

- Sometimes an extra 100 boards is just a few bucks more

# 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, mostly

- If the fab is going to assemble it, let them source parts where possible Importing parts into China is… fun?

Questions?

Snide Comments?

@hamster