



SO YOU WANT TO BUILD A BADGE!?! ↻

Building badges for fun and no profit





We are the Badge Pirates



WHY SHOULD YOU LISTEN TO US?

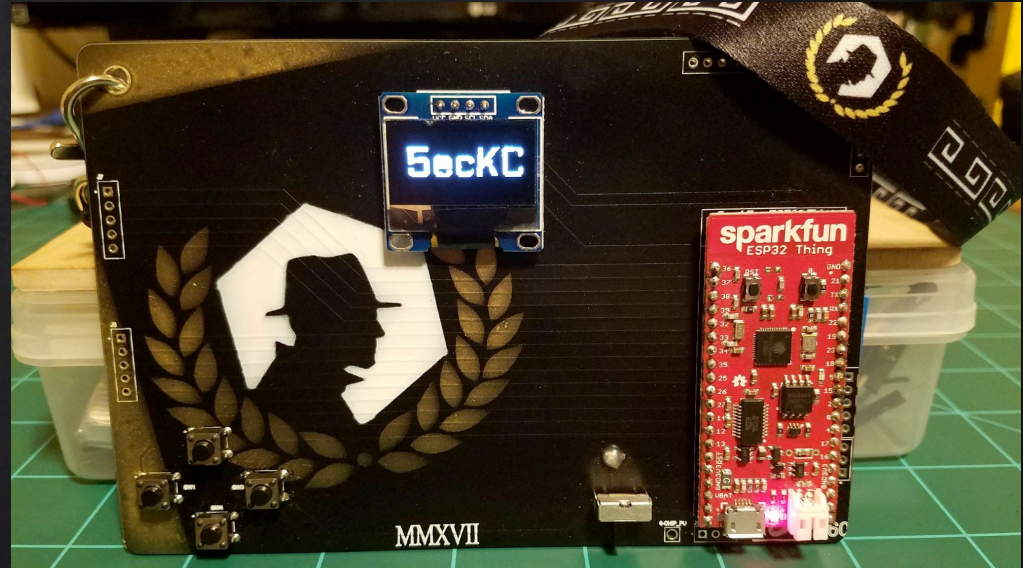
- 1) You shouldn't...
- 2) We have been making badges since 2017
- 3) We (or better put our board manufacturers) have produced 20K+ PCBs for different projects we have designed
- 4) First electronic badge and custom learn to solder project for Maker Faire KC (2019)
- 5) You really shouldn't...



OUR FIRST BADGE...

From concept to delivery in under 45 days

- Artwork was airbrushed using a laser cut stencil
- Powered by a ESP32 Thing Dev board
- There is hot glue on the front to keep the battery from shorting out
- I told you not to listen to us...

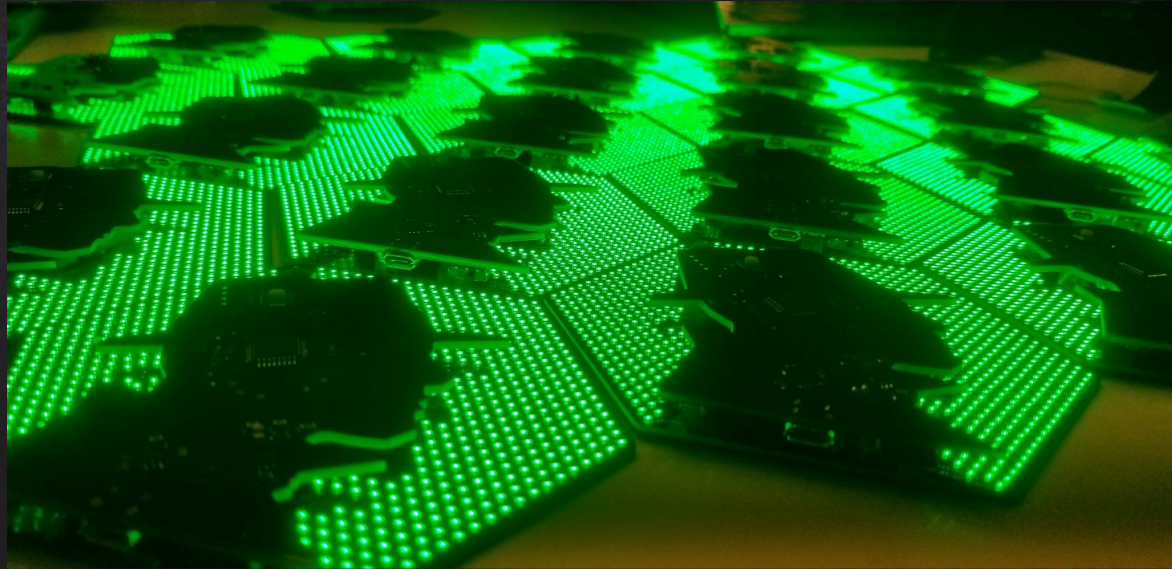


OUR MOST RECENT BADGE

SecKC DC27 unofficial Defcon badge:

- Badge is comprised of two separate boards
- Most LEDs ever on a Defcon badge (648)
- Took over 100K leds to make all the badges

Ok, maybe we've figured out a couple things..



1.

HOW ARE PCBs MADE?





“PCBs have layers. Ogres have
layers”

–Shrek (circa 2001)

PCBs 101

Printed Circuit Boards (PCBs) are built up in layers

Substrate/Core (commonly FR4)

- Copper
- Solder Mask
- Silkscreen

Most common number of layers is 2

- 1 layer each front/back

PCB Outline is referred to as the Edge Cut Layer



PCB CREATION PROCESS

- 1) Design the artwork
- 2) Import into PCB Electronics Design Automation (EDA) tool
- 3) Add any electronic circuit design if required (next talk)
- 4) Export project from EDA tool to Gerber files
- 5) Upload Gerbers to PCB manufacturer
- 6) Pay \$\$\$
- 7) PCBs arrive at your doorstep
- 8) No profit...

PCB TOOLS

TOOL CHAIN



Inkscape



Svg2shenzen
(Inkscape extension)



PCB Electronics Design
Automation (EDA)

STEP 1

Design your artwork

- Stick to vector formats
- It is possible to import vectors directly into Kicad as modules from DXF files

Things to note in PCB artwork design:

- The soldermask layer is a 'negative' layer and by default covers the entire PCB face
 - Anything placed on this layer will 'remove' the soldermask from that location
 - Soldermask layer that works this way
 - All other layers (silkscreen, copper) are 'inclusive' layers and will only be included if something is placed on that layer

If you place something on a back layer you MUST reverse it if you want it to look correct (unless it is a cutout)

FREQUENTLY USED PCB LAYERS (INKSCAPE / KICAD)

F.Cu (Front Copper)

F.Silk (Front Silkscreen)

F.Mask (Front Mask)

B.Cu (Back Copper)

B.Silk (Back Silkscreen)

B.Mask (Back Mask)

Edge.Cuts (Exterior Board outline)

EXAMPLES

...assuming the demo gods are not looking kindly on us

Inkscape w/svg2shenzen



KiCad PCB Viewer

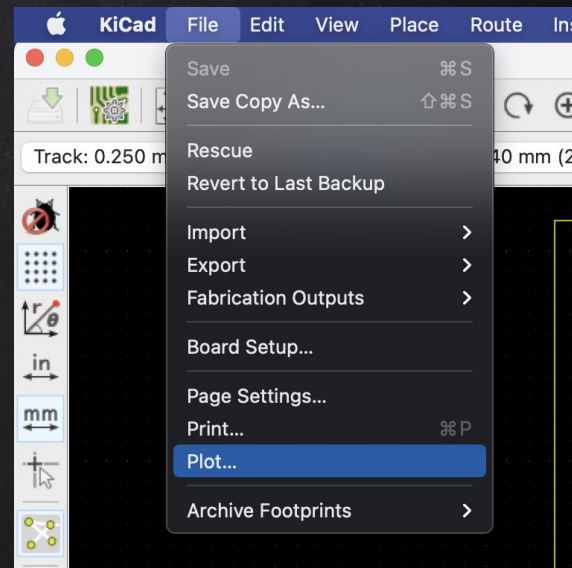
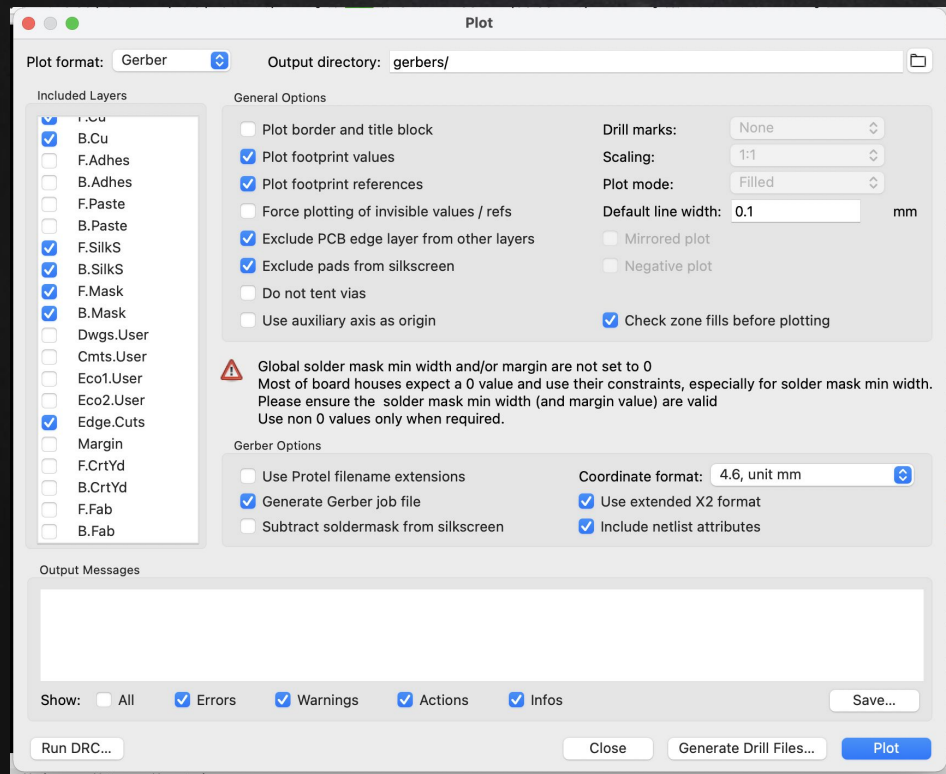


KiCad PCB Viewer (3D Render)



STEP 2

Export the Gerber Files from KiCad



(Design Rule Check)

STEP 3

Place the order:

- Pick a board manufacturer (PCBWay, OSHPark, JLCPCB, etc)
- Upload Gerbers
- Select Soldermask, Silkscreen & Copper elements
- Smash the buy button like its \$1 BTC (or ETH if your into that)



ORDERING PCBs

Substrate

PCB thickness

Solder Mask

Silkscreen

Copper Finish

PCB Specification Selection [How it works \(3 steps\)](#) [Quick-order PCB >>](#)

Board type: ☒ Single pieces ☐ Panel by Customer ☐ Panel by PCBWay

Different Design in Panel: 1 2 3 4 5 6

* Size (single): X mm

* Quantity (single): pcs

Layers: 1 Layer ☒ 2 Layers ☐ 4 Layers ☐ 6 Layers ☐ 8 Layers ☐ 10 Layers ☐ 12 Layers ☐ 14 Layers

Material: ☒ FR-4 ☐ Aluminum ☐ Rogers ☐ HDI(Buried/blind vias) ☐ Copper Base

*Material model can be remarked below. HDI is available for 4-layer or more.

FR4-TG: TG 130-140 ☒ TG 150-160 ☐ TG 170-180 ☐ S1000H TG150

Thickness:

* Unit: mm

Min Track/Spacing: 3/3mil 4/4mil 5/5mil ☒ 6/6mil ☐ 8/8mil

Min Hole Size: 0.15mm 0.2mm 0.25mm ☒ 0.3mm ☐ 0.8mm ☐ 1.0mm ☐ No Drill

Solder Mask: ☒ Green ☐ Red ☐ Yellow ☐ Blue ☐ White ☒ Black

☐ Purple ☐ Matte black ☐ Matte green ☐ None

Silkscreen: ☒ White ☐ Black ☐ None

Edge connector: Yes ☒ No

Surface Finish: ☒ HASL with lead ☐ HASL lead free ☐ Immersion gold(ENIG) ☐ OSP ☐ Hard gold ☐ Immersion silver(Ag)

☐ ENEPIG ☐ None(Plain copper)

WHAT'S NEXT?

So you're going to design your own PCB now...right?

Need help? Have questions? Are you on SecKC discord?

You are on SecKC discord right?

#badgelife

THANKS FOR YOUR TIME!



@carlfugate
@badgepirates



@networkgeek



LINKS TO TOOLS

Inkscape: www.inkscape.org

Svg2shenzen: <https://github.com/badgeek/svg2shenzhen>

KiCad: www.kicad.org

PCB Manufacturer:



www.pcbway.com

