



# Making #Badgeline: BSidesDFW AKIRA Badge

# How do you make a Badge?

- Determine Concept – Shape/Art/Function/Etc.
- Choose a Program to Design With
- Work Out Electronics via Schematic/Breadboard
- Layout Electronics/Components on PCB Design
- Layer PCB Shape/Cutouts/Silkscreen-Art
- Output Gerber Files for Manufacturer
- Choose a Manufacturer to Make Your Badge
- Assemble Your Badge

# Determine Concept

Figure out what you want your badge to look like and what it will do. Then consider:

- PCB Shape and Cutouts and Color
- Electronics / Components Required
- Silkscreen Artwork and Text and Color
- Power Source and Attachment to PCB
- Overall Wearability / Usability

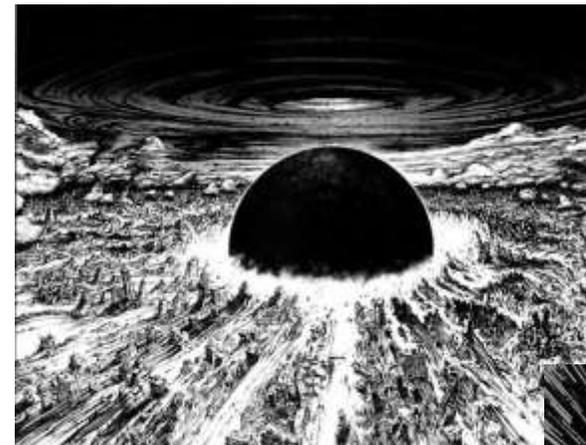


## Determine Concept

For the BSidesDFW AKIRA Badge @0isac0 came up with the AKIRA theme, I came up with the concept of the bike in front of the nova, and @unspecific helped me with some of the artwork.

For the electronics I wanted to use this badge as a platform to showcase the CH552G microcontroller, my first surface mount badge, and my first badge with a crypto challenge. The circuit lights LED's placed around the edges of both bike tires.

The result is a great looking badge that uses light to give the illusion of motion.



# Choose a Program to Design With

Figure out what program you will use to design your badge. There are many options each with pros and cons. Here are a few:

- KiCad EDA (<http://kicad-pcb.org>)
- Eagle PCB (<https://www.autodesk.com/products/eagle/>)
- DesignSpark (<https://www.rs-online.com/designspark/>)



# Choose a Program to Design With

Figure out what program you will use to work on the ART for your badge. Most PCB design programs require vector graphics for importing art to the silkscreen layers. There are many options each with pros and cons. Here are a few:

- Adobe Illustrator (<https://www.adobe.com/products/illustrator.html>)
- Inkscape (<https://inkscape.org/>)
- BoxySVG (<https://boxy-svg.com/>)



INKSCAPE



BOXY SVG

## Choose a Program to Design With

For the BSidesDFW AKIRA Badge, I decided to use KiCad for the PCB design.



This made the most sense since I had been using this platform for other badges and PCB designs and had quite a bit of experience with it.

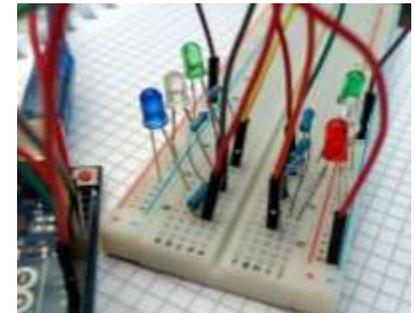
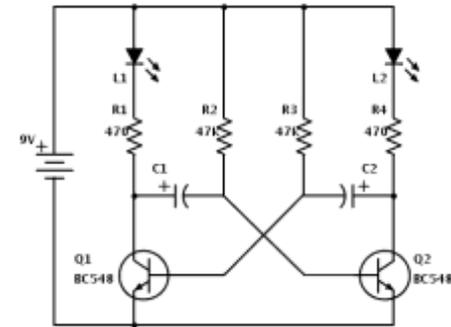
As for vector graphics, I have used Adobe Illustrator for many years making it my preferred choice for graphics.



# Work Out Electronics via Schematic

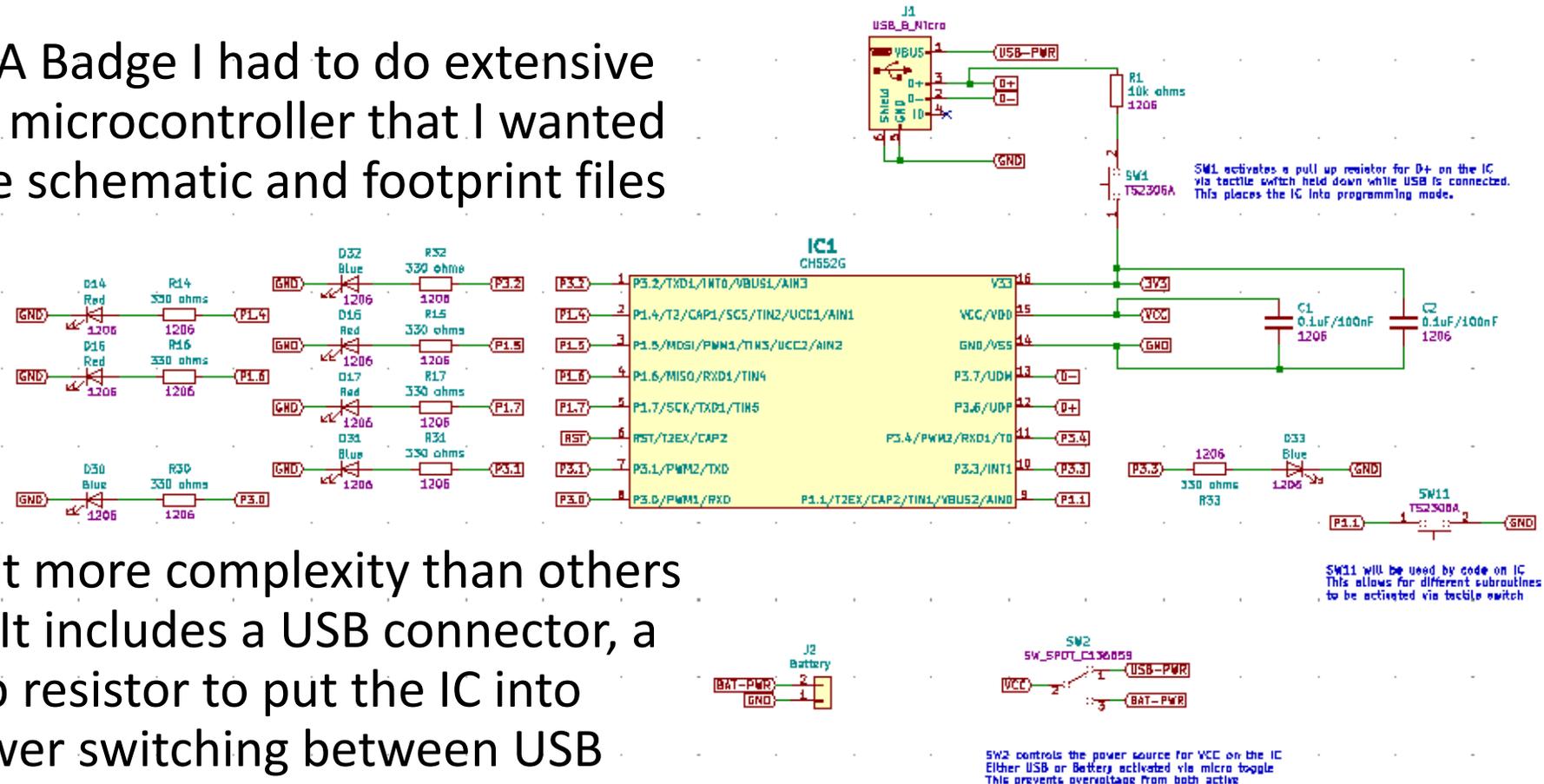
Figure out how the electronics should work for whatever it is your badge will do.

- It is best to start with a circuit schematic.
- If possible, you should test your electronics prior to committing them to a PCB. This can be accomplished by building your circuit schematic on a breadboard or prototyping board.
- Having a “known good” working electronics design will prevent issues with your final manufactured badge.



## Work Out Electronics via Schematic

For the BSidesDFW AKIRA Badge I had to do extensive research on the CH552G microcontroller that I wanted to use and then build the schematic and footprint files for it from scratch.

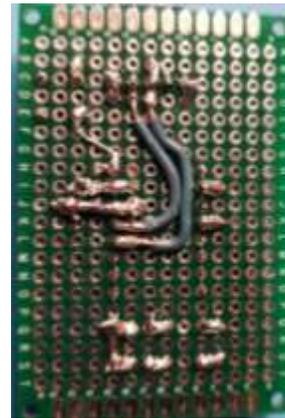
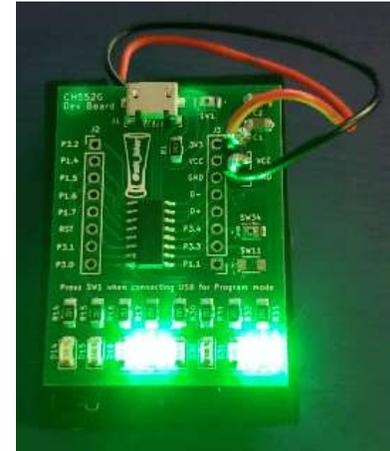
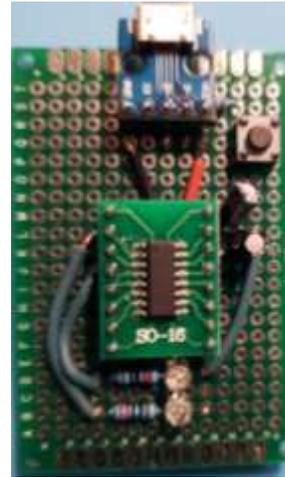


This circuit has quite a bit more complexity than others I have done in the past. It includes a USB connector, a button controlled pull up resistor to put the IC into programming mode, power switching between USB and battery, a button to control LED modes, etc.

## Work Out Electronics via Schematic

For the BSidesDFW AKIRA Badge I also had to do extensive testing of the new chip and circuit before committing to do it on a badge.

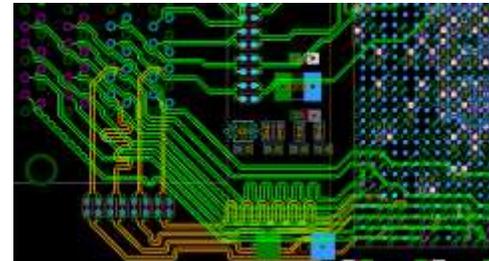
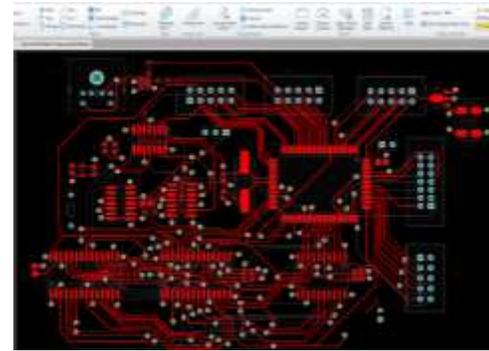
I started with a fairly ugly development prototype and progressed to a dedicated PCB for testing the chip and circuit functionality and finally the LED colors that might look best.



# Layout Electronics on PCB Design

Convert your electronics schematic into a PCB design by placing components and trace wires between them.

- Most PCB design software will auto populate the PCB design with components from a schematic.
- Move the components around to where you want them placed on your PCB
  - For Through Hole components make sure the spacing between the holes matches your component specifications
- Connect the component leads with lines that represent the copper traces that will be added to the PCB.
  - Place and connect VIA (vertical interconnect access) points on multi-layer boards to connect the layers where needed



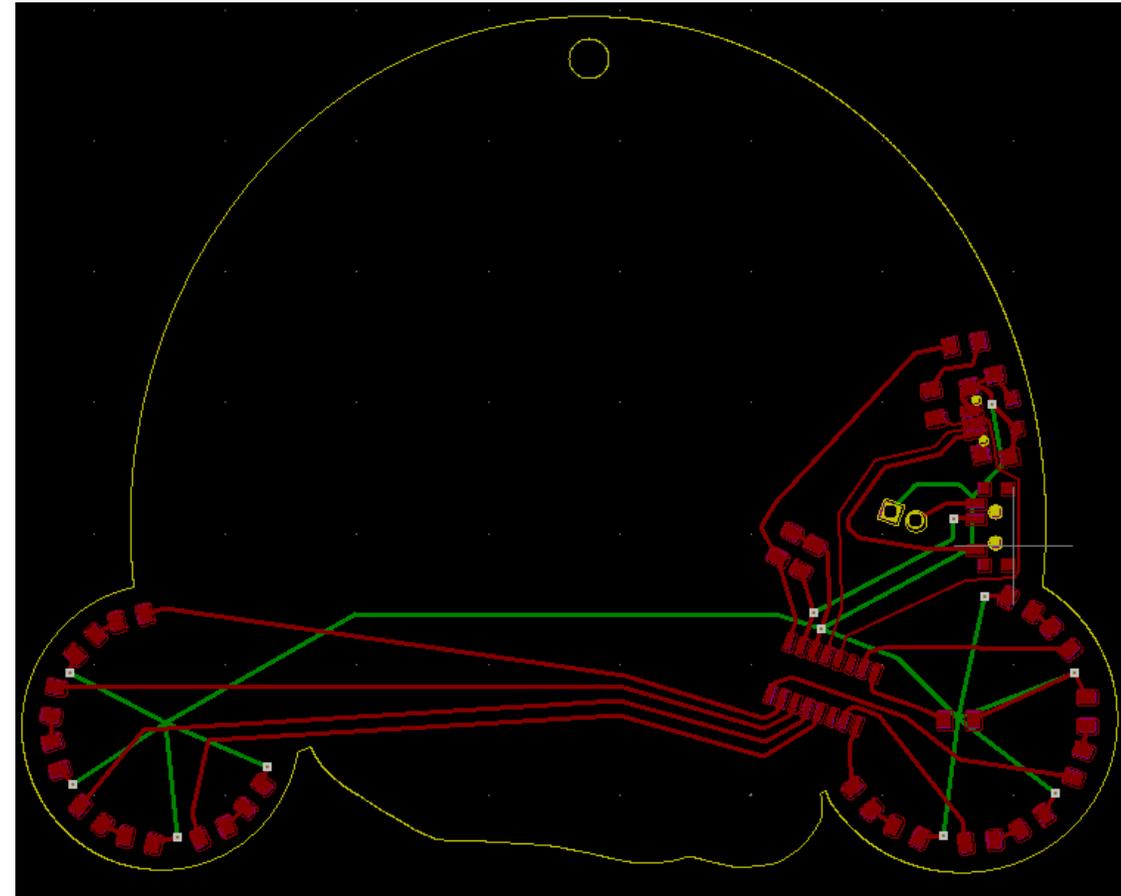
## Layout Electronics on PCB Design

For the BSidesDFW AKIRA Badge I knew that the design included a motorcycle that I wanted to light the tires on. In addition I wanted to hide the other components into the design of the bike.

So I estimated the component positions for the initial layout. This would later be adjusted when the board cutouts and silkscreen layers were added.

I had to verify and adjust each components size in the software so that the components and their solder pads would be placed in the correct positions. This was done by downloading the specification sheets for the components I was going to use which detailed the spacing requirements.

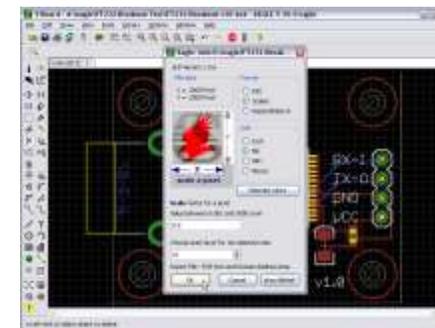
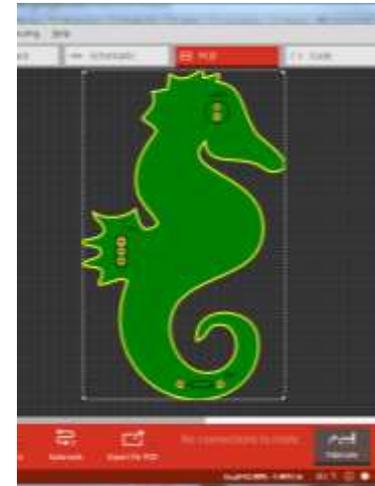
For many components footprint files existed but for some I had to create them. Regardless if a footprint existed or not I verified that each footprint I chose matched the specifications I needed.



# Layer PCB Shape/Cutouts/Art

Once your basic PCB design is complete layer in other things like PCB shape and cutouts and Silkscreen Art.

- The outer shape of the PCB, the internal cutouts, and the silkscreen art are all separate layers in the PCB design.
- Add each of these to the design (usually by importing a vector graphics file) and adjust your component layout and traces and vias accordingly.



## Layer PCB Shape/Cutouts/Art

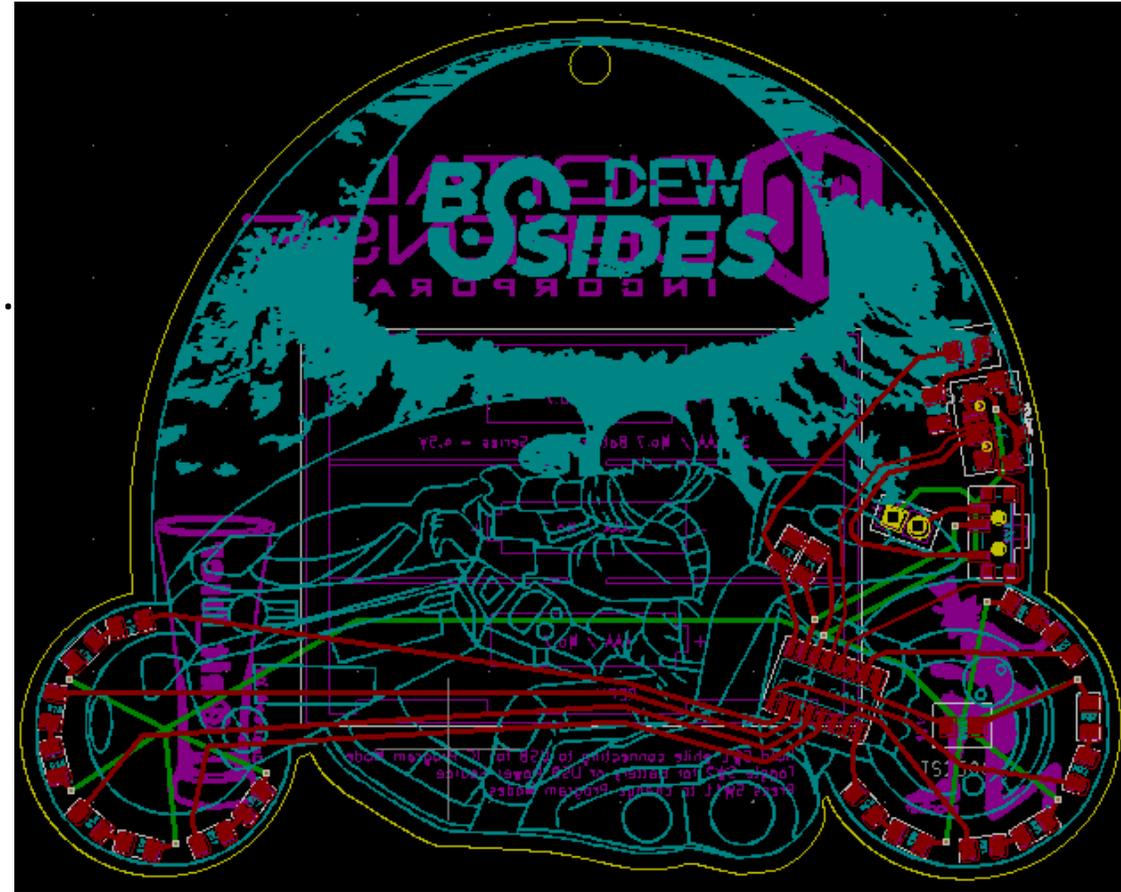
For the BSidesDFW AKIRA Badge I created several images using Illustrator that I imported into KiCad.

One image was the edge cuts layer that had the badge shape. Another contained the top layer silkscreen art. Several more contained logos for the bottom layer silkscreen.

I imported some of these files into the KiCad using its Bitmap2Component tool which turned them into Kicad Module files (.kicad\_mod) that I could import within Kicad's PCBnew editor. This worked for all the images in the F.Silks and B.Silks layers.

However, the Edge Cuts layer is special and required the image to be exported to DXF format in order to be imported into the Edge.Cuts layer properly.

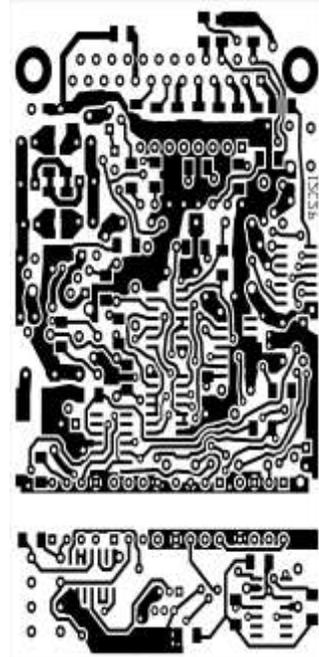
With all of these layers imported I just needed to adjust the component placement to fit into the shape.



# Output Gerber Files for Mfr.

When your PCB design is complete you will want to export Gerber files for the PCB manufacturer. The Gerber file format is a standard that most PCB manufacturers use.

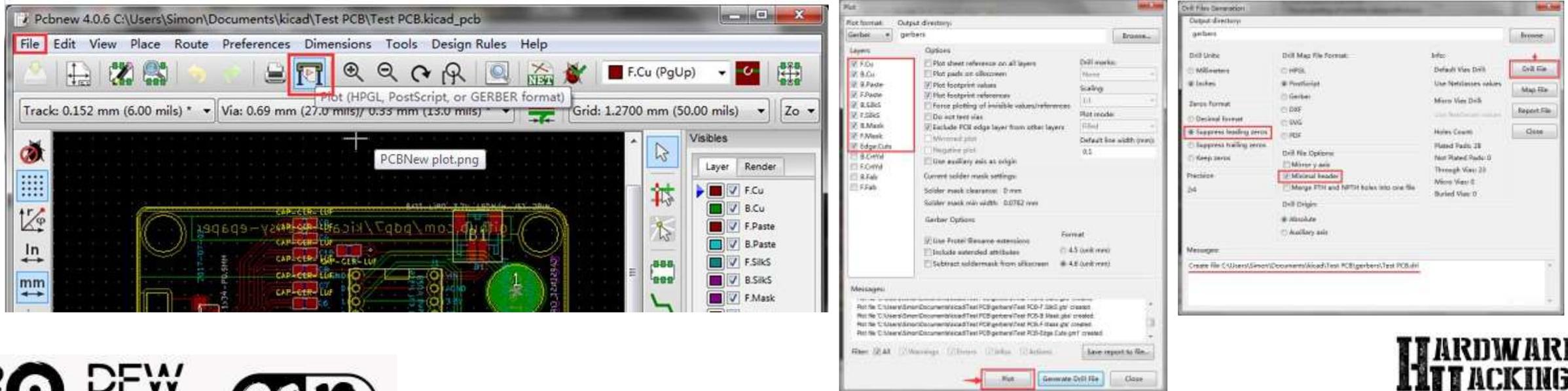
- The Gerber format is an open ASCII vector format that has been the de facto standard used by the PCB industry for over 30 years.
- The Gerber files describe the printed circuit board images: copper layers, solder mask, legend, etc. The Mfr. uses these files to fabricate the PCB
- Most PCB design software provide a method for exporting the Gerber files that the Manufacturer will need.
- Most PCB Mfrs. will tell you how they want to receive these files. (e.g. Zip file via Web upload)



## Output Gerber Files for Mfr.

For the BSidesDFW AKIRA Badge I used the KiCad export function to export the Gerber files required to produce the PCBs.

This was done by choosing the “Plot” setting within the PCBNew tool. Using the menus I chose all the layers I wanted plotted to Gerber files and also created a drill file. This created a total of nine files that I then added to a single ZIP file to send to the manufacturer.



# Choose a Mfr. to Make Your Badge

There are several factors to consider when choosing a PCB manufacturer. Cost is probably the top factor. But, time and location and quality are all also important. The cheapest PCB Mfr. might take months to deliver or might have poor quality. Look at customer reviews and do some small prototype runs to confirm they meet your needs. There are many PCB Mfrs. to choose from. Here are a few:

- <https://pcbshopper.com/> ← Travelocity like site for PCB Mfrs.
- <https://jlcpcb.com/> ← Inexpensive but with some limitations.
- <https://pcbway.com/> ← Inexpensive but with some limitations.
- <https://oshpark.com/> ← A bit pricy but beautiful and made in USA

## Choose a Mfr. to Make Your Badge

For the BSidesDFW AKIRA Badge I did a bake off between JLCPCB.com and PCBWAY.com with prototype runs.

We ended up liking the look of the colors on the JLCPCB run better than PCBWAY. The matte black looked better and the white silkscreen was brighter.

The kicker was that JLCPCB was also a bit cheaper.

• <https://jlcpcb.com/>

← Used This Manufacturer

## Success! Questions?

We hope everyone at the BSidesDFW 2019 conference enjoys this badge as much as those of us who worked on it do.

<https://akirabadge.com> ← Badge Code & Info

