Making #Badgelife: The BSidesDFW 9 Badge

BSIDES DFW 9

Presented by @alt_bier

HARDWARE HACKING VILLAGE

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

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 9 badge MadHat @unspecific worked on the concept and the art work which I used to make the badge.

He started with the Cat theme as a play on 9 lives and this being the 9th BSidesDFW conference. Other concepts were added in such as the Texas Star shape and the “Last One Digit Con” tag line.

For the electronics we decided on a LAN Tap function that worked like Mike Ossmann’s Throwing Star LAN Tap.

The result was a great looking badge with a simple reliable design that would teach its owners about basic circuits, soldering, and using LAN Tap’s for viewing traffic on a wire.

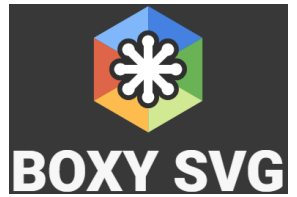
Choose a Program to Design With

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

- Fritzing (**http://fritzing.org**)
- 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/**)

Choose a Program to Design With

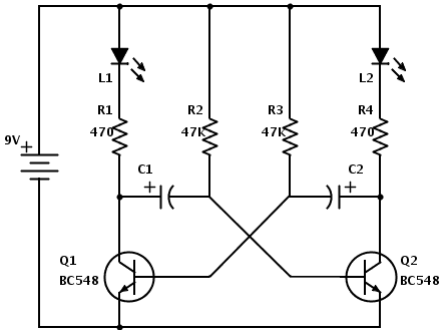
For the BSidesDFW 9 badge, I decided to use KiCad for the PCB design.

This made the most sense since we decided to use a similar electronics design as Mike Ossmann's Throwing Star LAN Tap. His Tap was designed in KiCad and he made his code available via github.

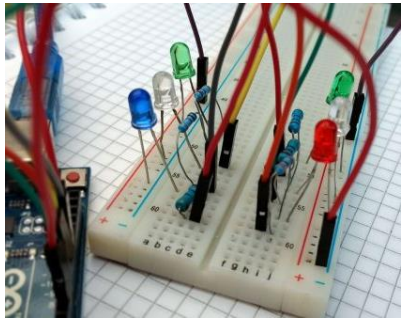
As for vector graphics, I have used Adobe Illustrator for many years making it my preferred choice for graphics. This is also the program that MadHat @unspecific used for the artwork that I imported into the badge.

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.



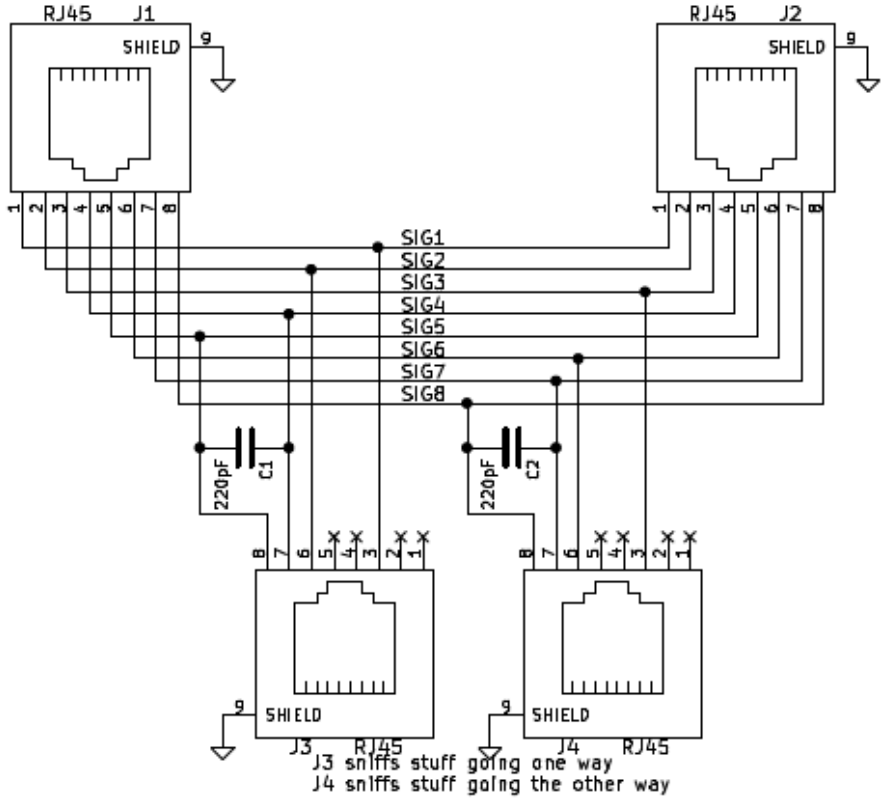
Work Out Electronics via Schematic

For the BSidesDFW 9 badge I used a similar electronics design as the Throwing Star LAN Tap.

This circuit connects four RJ45 connectors and two capacitors as a simple passive LAN Tap.

Two of the RJ45 connectors are set up for inline traffic between two devices. The other two receive a copy of the traffic between the inline devices to be monitored.

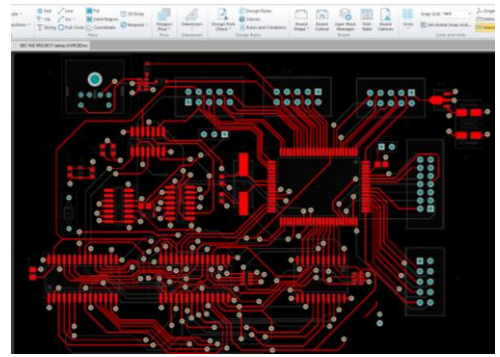
Since this is a passive Tap design it does not require a power source and is limited to viewing 100M Ethernet traffic. The capacitors trick the endpoints to auto negotiate a speed less than 1G.

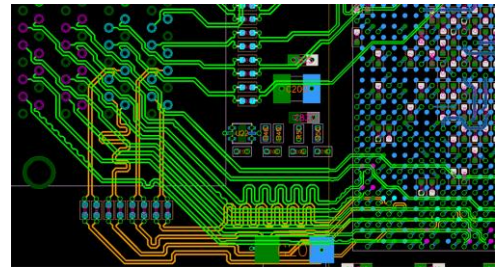


Layout Electronics on PCB Design

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



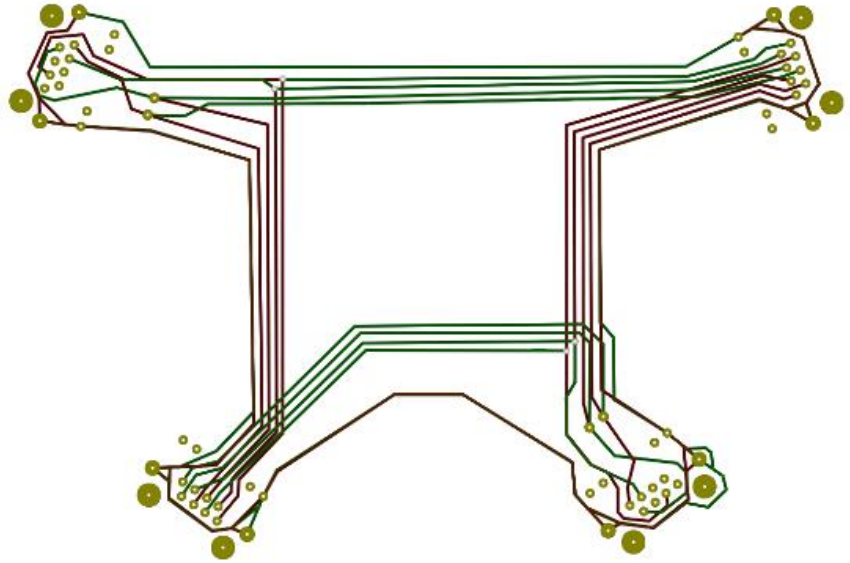


Layout Electronics on PCB Design

For the BSidesDFW 9 badge I knew that I was going for a Texas Star shape with the RJ45 connectors mounted in four of the star points.

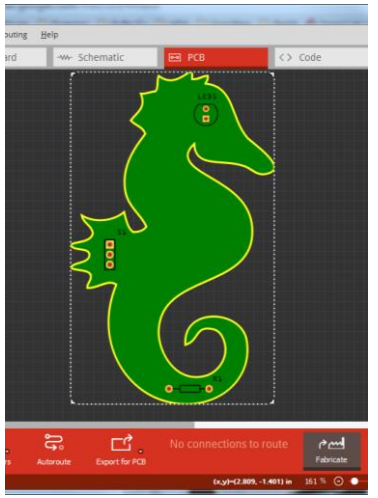
So I estimated these positions for the initial layout. This would later be adjusted when the board cutouts and silkscreen layers were added.

I had to verify and adjust each components size in the software so that the through holes and corresponding solder pads were 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.

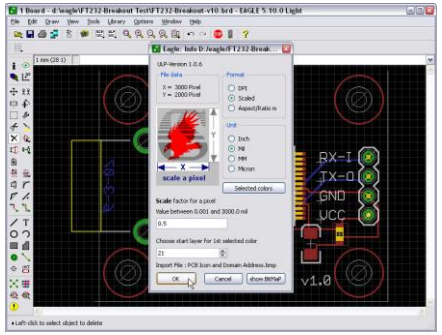


Layer PCB Shape/Cutouts/Art

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 9 badge I created three images using Illustrator that I converted to bitmap files for importing.

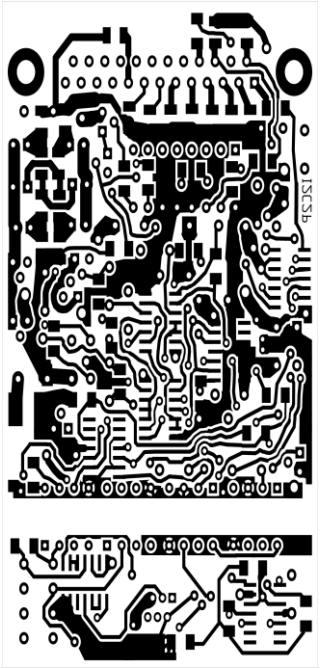
The first image was a filled shape of a Texas Star with voids for the cutouts. The second contained the top layer silkscreen art. The third contained the bottom layer silkscreen art.

I imported 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 PCB editor. I imported these module files into the layers Edge.Cuts F.SilkS and B.SilkS.

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.

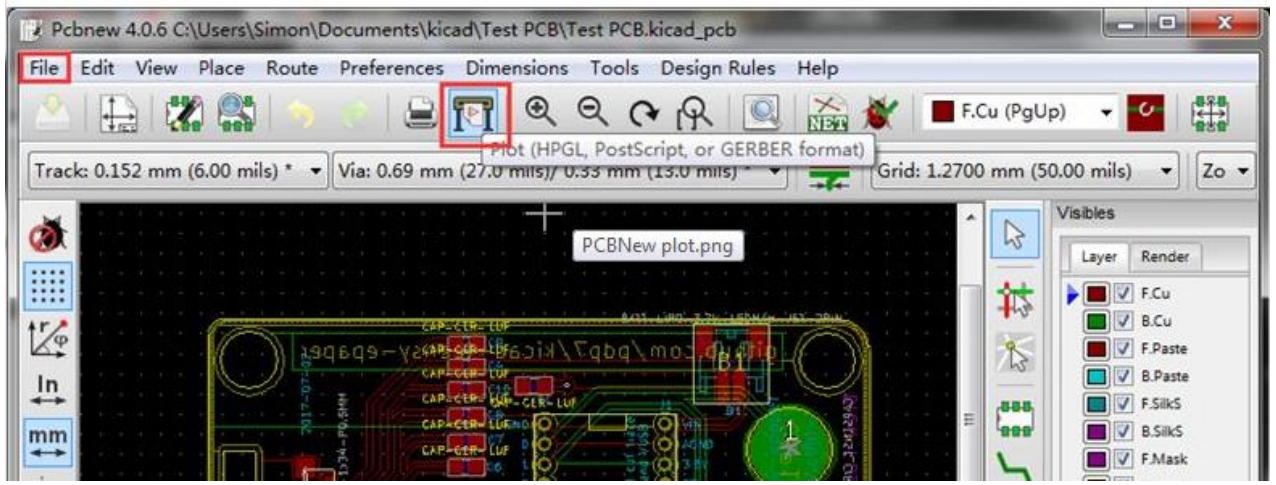


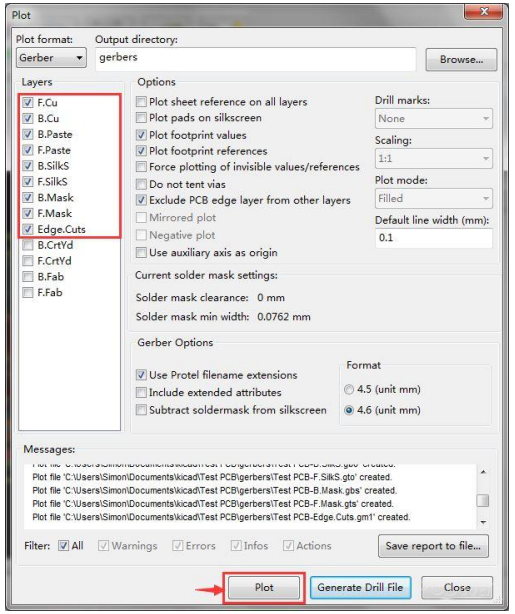
- The Gerber format is an open ASCII vector format that has been the de facto standard used by the PCB industry for over 30 years.
- The Gerber files describe the printed circuit board images: copper layers, solder mask, legend, etc. The Mfr. uses these files to fabricate the PCB
- Most PCB design software 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)

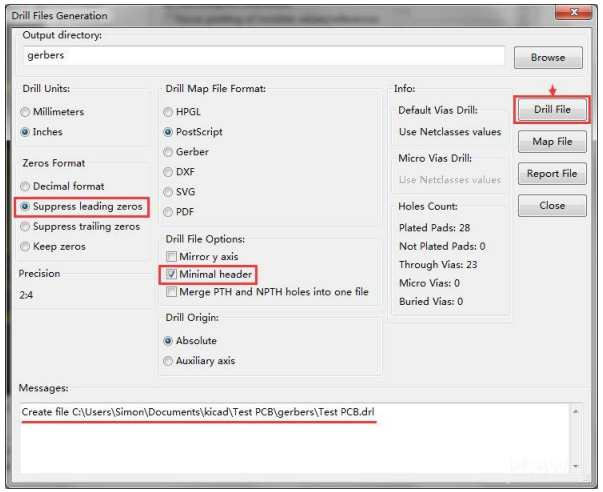
Output Gerber Files for Mfr.

For the BSidesDFW 9 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://oshpark.com/** ← A bit pricy but beautiful and made in USA

Choose a Mfr. to Make Your Badge

For the BSidesDFW 9 badge I wanted to use the same manufacturer that I had for the DFW Hacker badge JLCPCB.com since they did a good job.

I knew about the color limitations they had with PCB and solder mask options and that they only use white silkscreen color.

So, I made sure we used the colors they support to allow us to use them since they are inexpensive and fast. I was able to choose these colors from the web ordering interface and did not need to modify the Gerber files.

- **https://jlcpcb.com/** ← Used This Manufacturer

Success! Questions?

We hope everyone at the BSidesDFW 9 conference enjoys this badge as much as those of us who worked on it do.

The badge concept and artwork was done by MadHat @unspecific and the badge design was done by @alt_bier with input from many others. The electronics design was borrowed from the LAN Tap by Mike Ossmann.

https://github.com/gowenrw/BSidesDFW_2018_Badge ←Badge Code & Info