# Making #Badgelife:
# The DFW Hacker Badge

Presented by @alt_bier

# 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 DFW Hacker badge I had very little time to work on concept.

We had talked about a beer bottle badge in the past so I decided to use that shape.

For the electronics I needed to keep it a simple design I knew would work. So, I used a simple Arduino RGB LED design that I had used in various training classes in the past.

The result was a simple reliable design that would teach its owners about basic circuits, soldering, and Arduino programming.

# 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 DFW Hacker badge, given the time constraints, I had to use what I knew.

Even though I had never designed a PCB before I had used the Fritzing software to design circuits for training classes I gave. I knew it was capable of at least simple PCB design as well.

As for vector graphics, I have used Adobe Illustrator for many years making it my preferred choice for graphics.
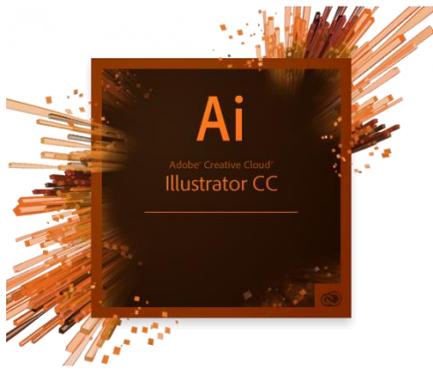
# Work Out Electronics via Schematic

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.
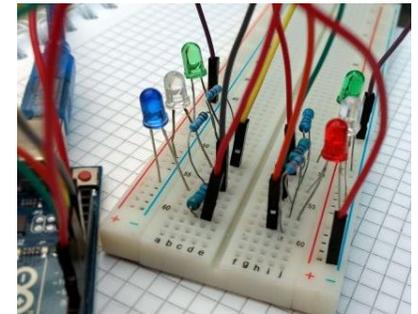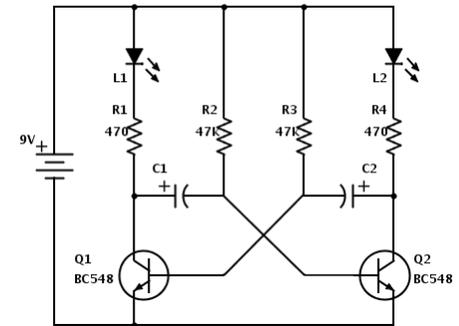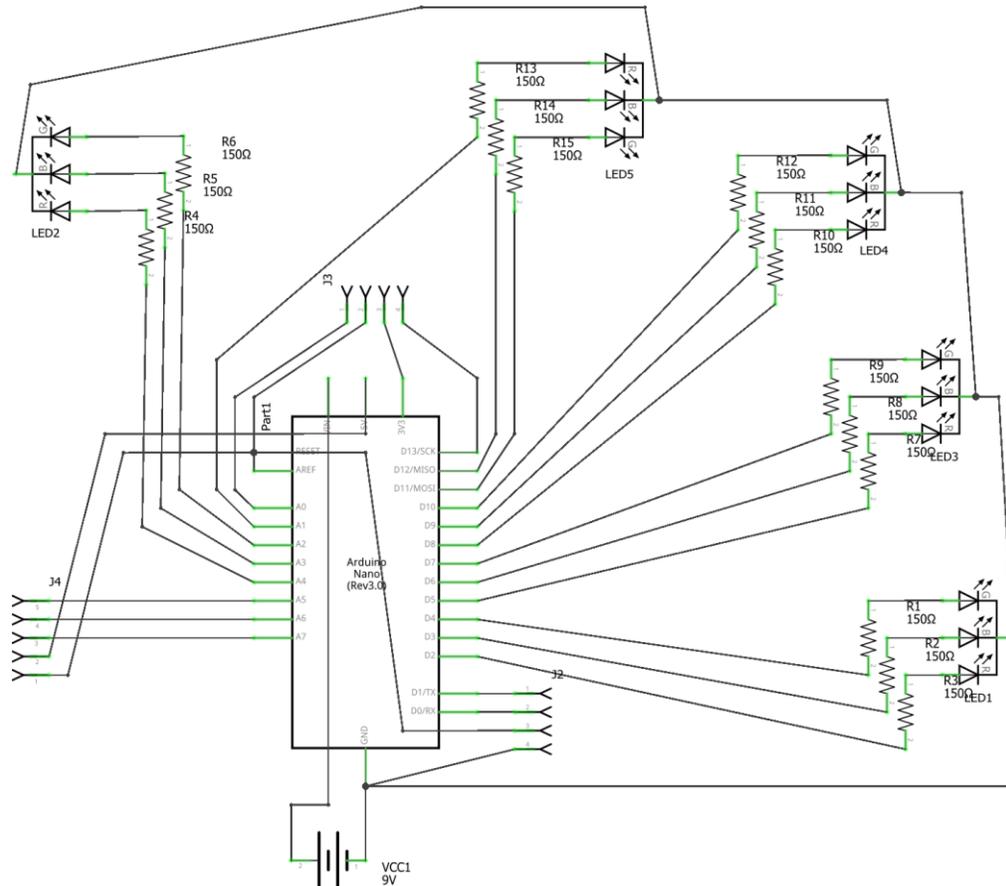
# Work Out Electronics via Schematic

For the DFW Hacker badge I used a modified Arduino circuit that I had used before.

This circuit connects five common cathode RGB LEDs to an Arduino Nano device that controls how they light up.

The original power source I planned on using was a 9V battery. But I later found a 6V coin cell holder that ended up being a better fit. Although the circuit will work with either since the VCC on the Arduino will step the input voltage down to 5V.



fritzing

# 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
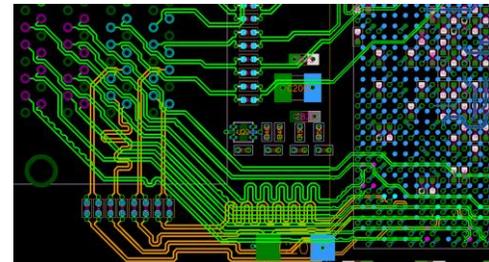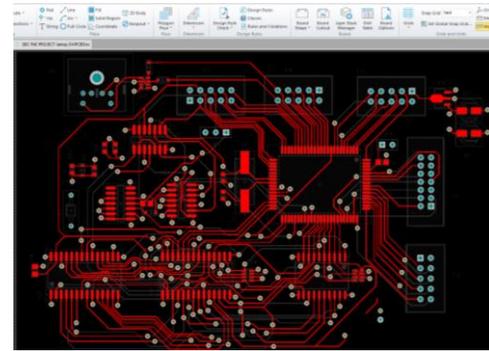
# Layout Electronics on PCB Design

For the DFW Hacker badge I knew that I was going for a beer bottle shape with the Arduino Nano mounted vertically at the bottom and the LEDs mounted in a star shape in the middle.

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.
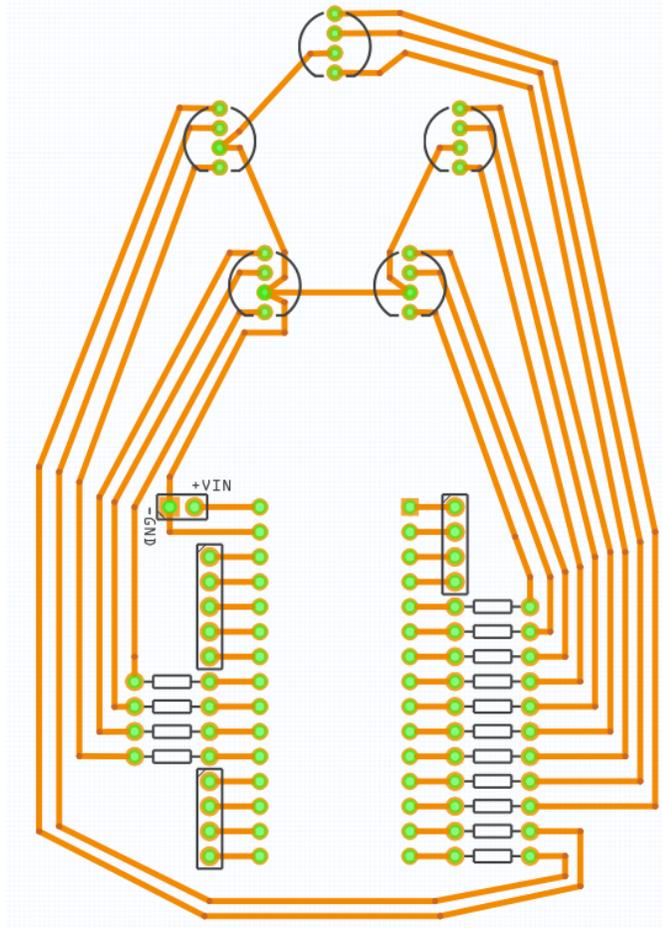
# Layer PCB Shape/Cutouts/Art

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.
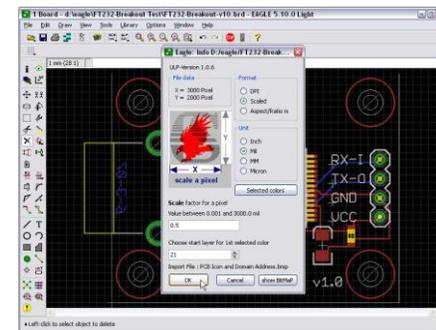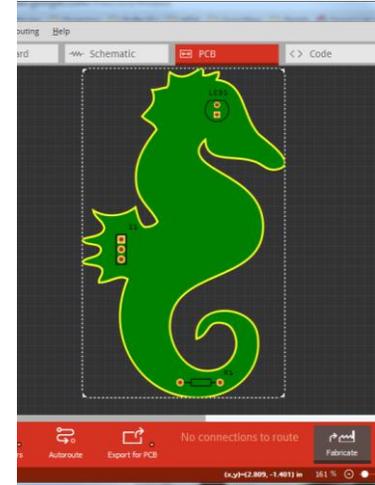
# Layer PCB Shape/Cutouts/Art

For the DFW Hacker badge I created two images using Illustrator that I then combined into a single image with two separate group IDs when exported to SVG.

The first image / group ID was a filled shape of a beer bottle with a void at the top for the lanyard cutout. The second image / group ID contained all of the top layer silkscreen art.

I then imported this SVG file into Fritzing.

This is where I ran into a problem. Fritzing requires width and height be specified. This is regardless of any viewBox that may be present. Illustrator exports with a viewBox only, so I needed to manually add the width and height matching the viewBox values.

With both of these layers imported I just needed to adjust the component placement to fit into the shape and art layers where I wanted them.

# 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.
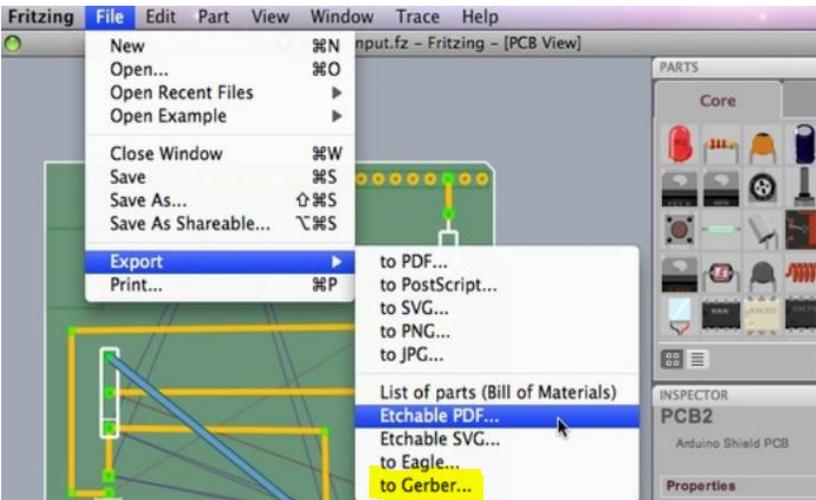
- The Gerber format is an open ASCII vector format that has been the de facto standard used by the PCB industry for over 30 years.

- 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)
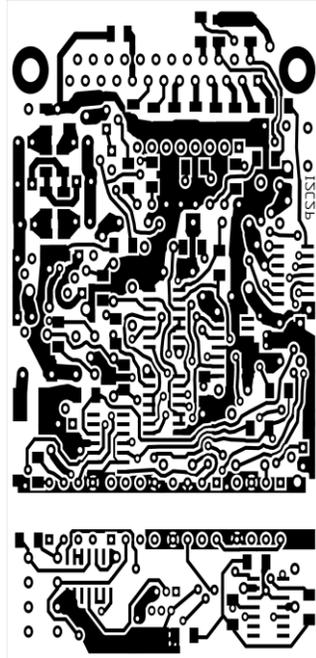
# Output Gerber Files for Mfr.

For the DFW Hacker badge I used the Fritzing export function to export the Gerber files required to produce the PCBs.

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 DFW Hacker badge I had no idea what manufacturer to use since this was my first ever badge.

I started with PCBShopper.com to get an idea of pricing and timeframes.  I ended up using JLCPCB.com since they were inexpensive and fast.

There were some limitations with this manufacturer such as them not supporting brown as a PCB or solder mask color (the original plan was for the badge to be a brown bottle) and they only use white silkscreen color.

But, given the tight timeframe and budget for the project this was the best manufacturer.  So, I changed the plan and made it a white on green design like a Rolling Rock or Heineken bottle.  I was able to choose these colors on their ordering website and did not have to modify the design files.

- **https://jlcpcb.com/**

← Used This Manufacturer

# Success!        Questions?

This badge was designed in six days and sent to manufacturing for a prototype run on day seven.  Five days after that I had badges in hand.  Twelve days total from idea to badge in hand and they turned out awesome!  DFWHackerBadge.com ←Code & Info