

ZAM Badge 2024

Überblick

Bei der Planung des VULCA-Treffens kam die Idee eines Badge als Konferenzpräsent und Lötübung auf. Die Idee besteht darin, die folgende Grafik zu nehmen und diese in eine Leiterplatte mit blinkenden LEDs hinter dem Auge und den Buchstaben umzuwandeln.



Projekt-Metadaten

[ZAM-Badge-24 GitHub Repository](#)

Jochen Hunger jochen.hunger@betreiberverein.de

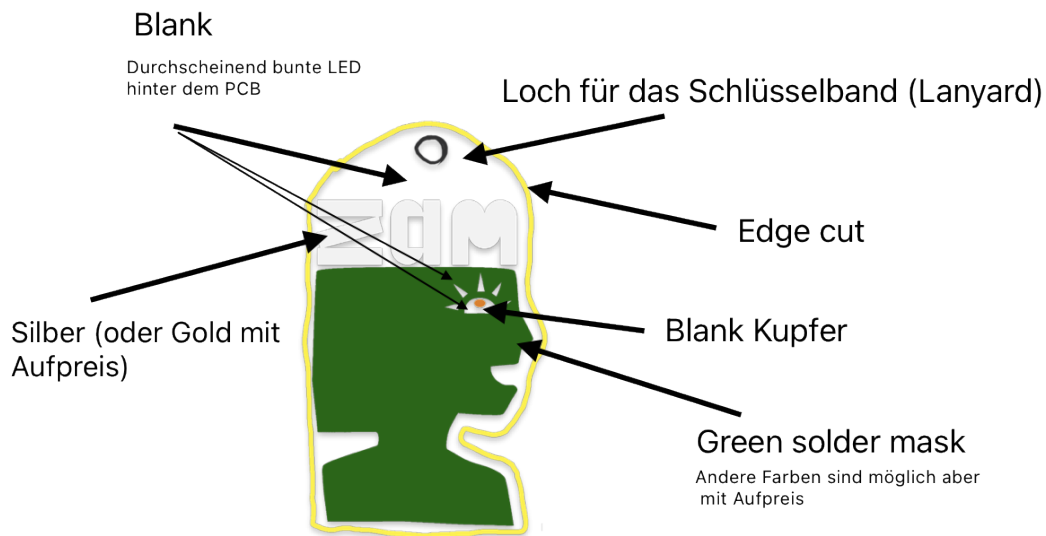
Steve Mayze smayze@yahoo.com

Logbuch / Schritte

Tag 1 Zusammenarbeit

Ich habe schon früher Grafiken in PCBs umgewandelt. Ich wollte dieses Projekt unbedingt in Angriff nehmen, da ich einige Wochen zuvor an der KiCon Europe 2024 teilgenommen hatte und den Vortrag „[Zusammenarbeit mit Künstlern, um schöne Platinen zu entwerfen](#)“ sehr genossen hatte.

Als Erstes musste ich mir das Konzept ausdenken. Auf Grundlage der Originalgrafik skizzierte ich eine mögliche Anordnung der PCB-Schichten.

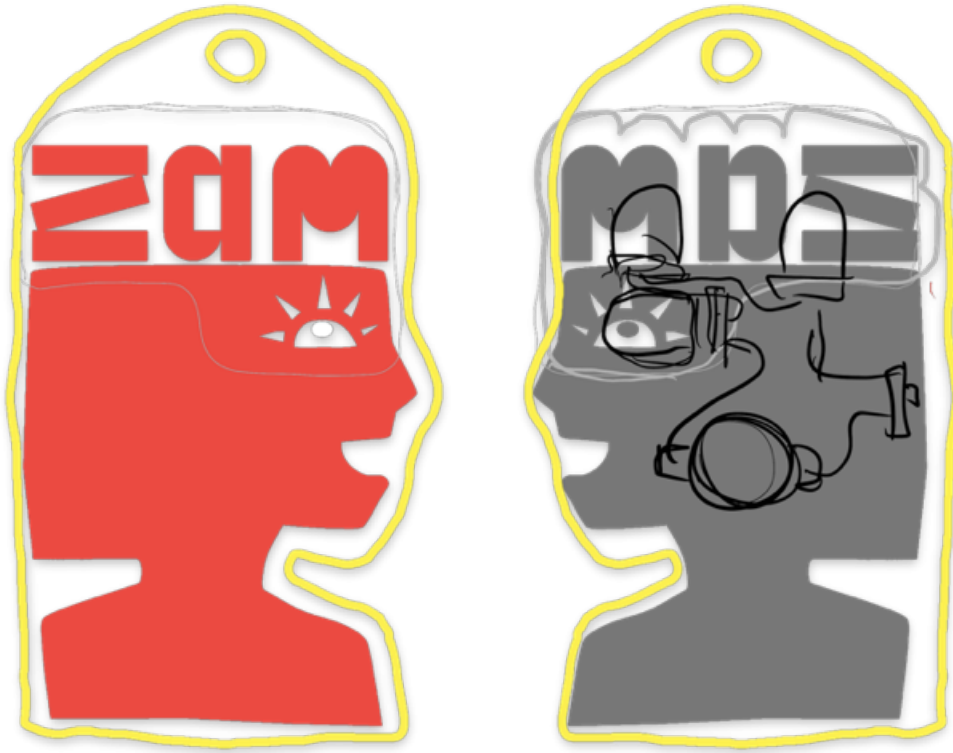


Basierend auf dieser Skizze hat Iochen dann seine Vorstellungen dargelegt. Daraus ließe sich . Die Idee ist, Kopf und Buchstaben alle in der gleichen len Bereich hinter den Augen und um die Buchstaben LEDs hindurchscheinen.



Ich habe dies noch etwas weiter ausgeschmückt und mir

Gedanken darüber gemacht, wo die Komponenten platziert werden könnten.

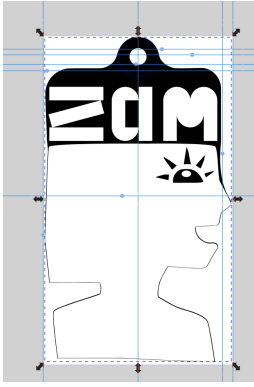


Tag 2. Erstellen des Designs

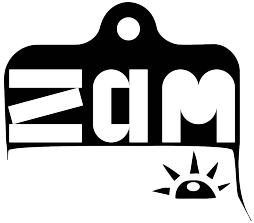
Once I had KiCad available I proceeded to turn the SVG of the graphic into a PCB. To achieve this, the layer feature of Inkscape was invaluable.



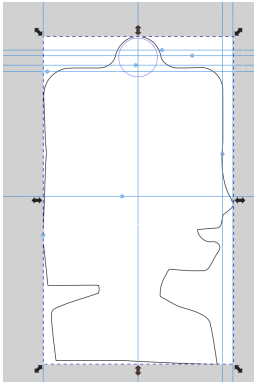
The original image was exported as Front Copper (F.Cu). This will ensure that the section of the board that represents the *head* and letters will be opaque and not let any light through.



The next layer to export as the solder mask. For the entire image was exported but this has to be converted to a negative since this defines where the solder mask *should not go*.

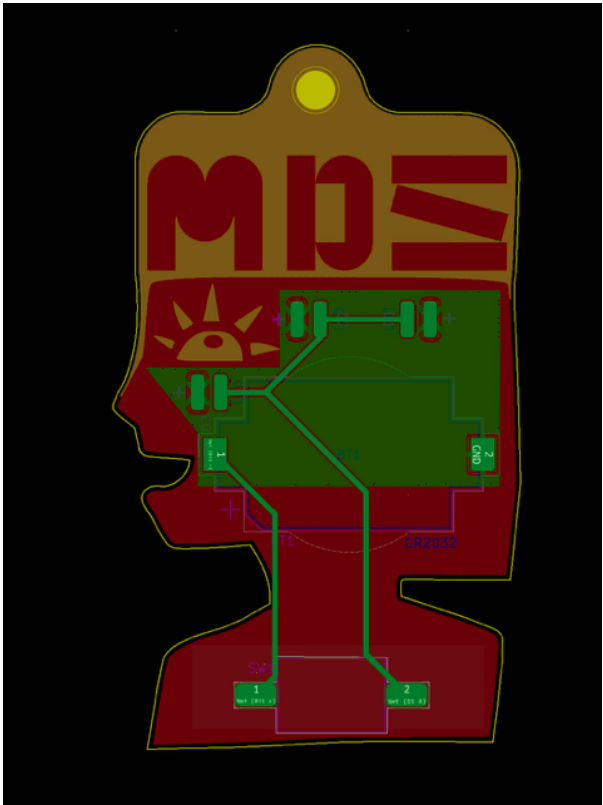
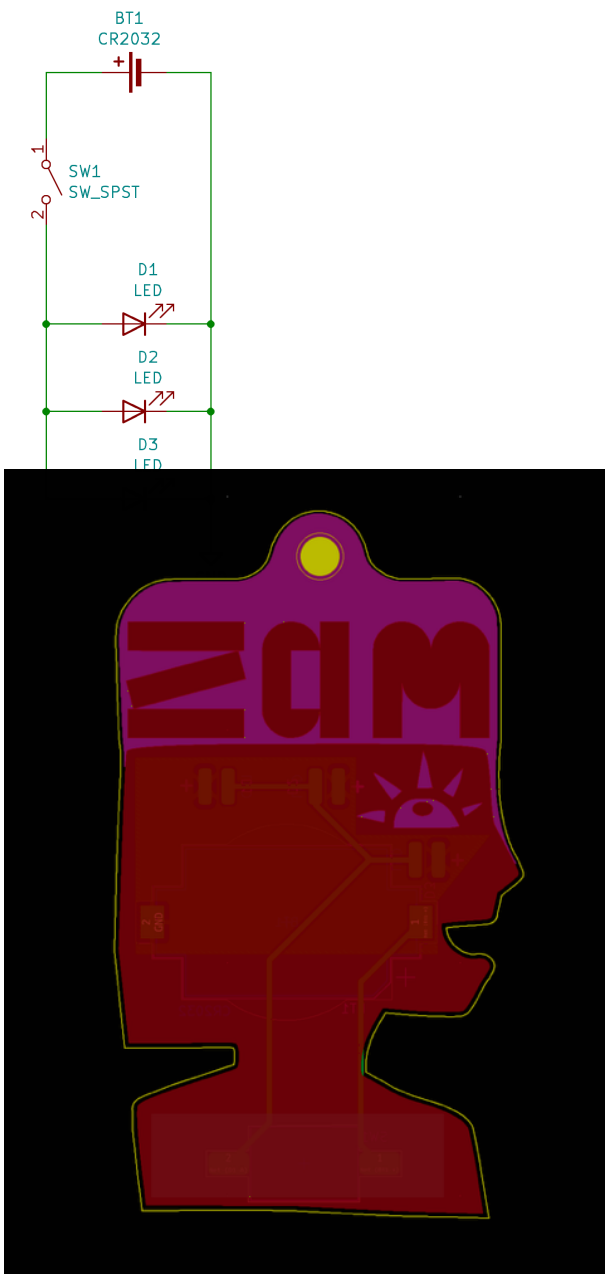


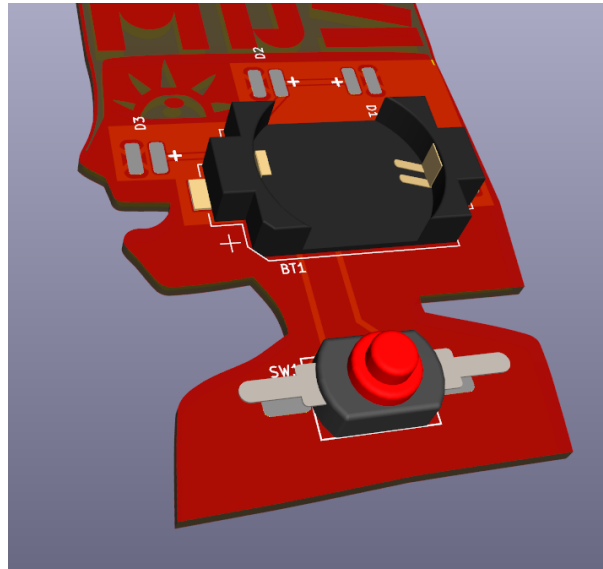
This image is then taken into a graphics program such as GIMP or Krita and trimmed down as the entire outline for this design.



The last layer is based on the entire image and converted to an outline. This is then expanded using the *Outset tool*. This will ensure that milling tool following the edge cut will not conflict with the actual image outline.

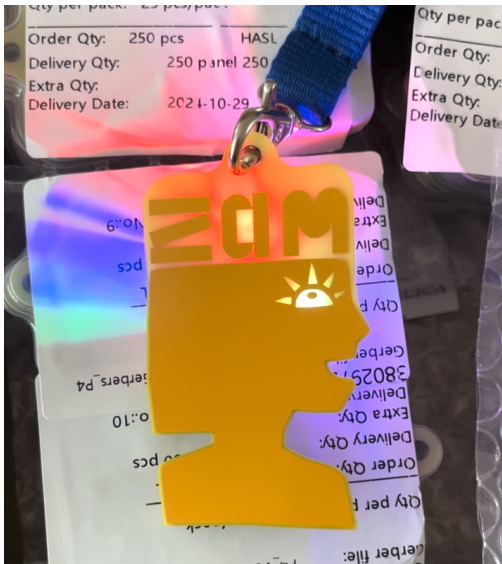
Once the layers are exported and verified, the layout of the board can begin. First of all, a simple circuit is created to defined the required parts. We don't have a lot of time for this project to try things out. So I opted to have space for three LEDs though I believe only two will be required.





Tag 3. Montage und Prüfung

A week or so after uploading the design files to the fabricator, a panel arrived in the post. I had to assemble to see if there were any issues.



The assembly is quite simple. We start by soldering the LEDs first. The battery holder is large and the LED pads are quite close. There is a risk of damaging the battery holder if the soldering iron should touch it while soldering the LEDs.

The leads of the LEDs should be bent into place and cut to length. Make sure to take note of the anode and cathode of the LED. These leads need to connect to the appropriate pad. If they are connected the wrong way, you will not damage the LED. It just won't glow. The anode is the longer lead and should be placed on the pad marked with the plus (+) sign.

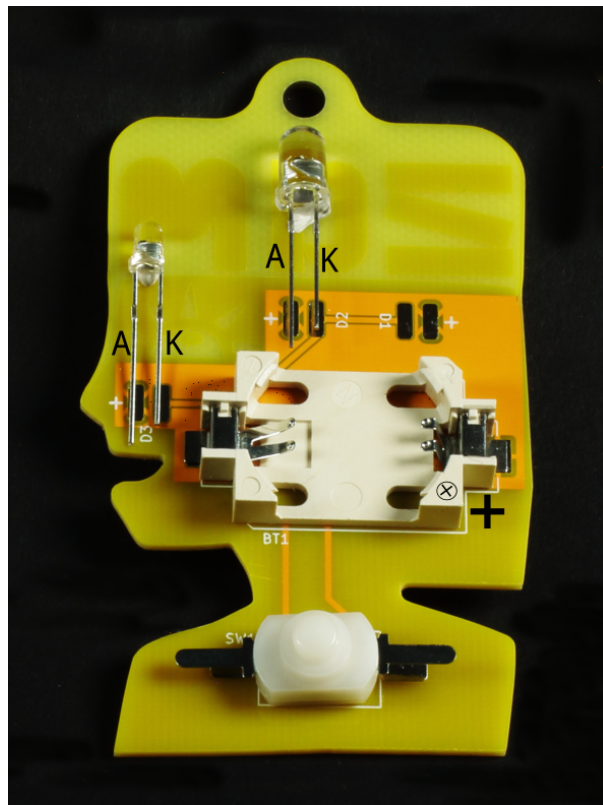
After the LEDs are soldered in place, then the battery holder and switch can be placed on the board. Like the LEDs take note of the orientation of the battery holder. Refer to the graphic below to see the correct orientation. The ⊕ marking on the battery holder should be bottom right as shown in the image.

The switch has no special orientation. The leads will need to be bent into shape so that they neatly touch their pads.

Note that for the battery holder and the switch you might need to apply more heat to get a good connection.

Bauteile

- 3mm LED - eye



Revision #9

Created 1 November 2024 09:31:42 by Steve Mayze

Updated 4 November 2024 08:58:09 by Steve Mayze