PCB

Board size: 63.46x34.39 mm (2.5x1.35 inches)

- This is the size of the rectangle that contains the board
- Thickness: 1.6 mm (63 mils)
- Material: FR4
- Finish: HAL
- Layers: 2
- Copper thickness: 35 µm

Solder mask: TOP / BOTTOM

- Color: Green

Silk screen: TOP / BOTTOM

- Color: White

Important sizes

Clearance: 0.2 mm (8 mils)

Track width: 0.4 mm (16 mils)

- By design rules: 0.0 mm (0 mils)

Drill: 1.1 mm (43 mils)

- Vias: N/A mm (N/A mils) [Design: 0.4 mm (16 mils)]
- Pads: 1.1 mm (43 mils)
- The above values are real drill sizes, they add 0.1 mm (4 mils) to plated holes (PTH)

Via: N/A/N/A mm (N/A/N/A mils)

- By design rules: 0.5/0.3 mm (20/12 mils)
- Micro via: yes [0.2/0.1 mm (8/4 mils)]
- Buried/blind via: yes
- Total: 0 (thru: 0 buried/blind: 0 micro: 0)

Outer Annular Ring: 0.3 mm (12 mils)

- By design rules: 0.3 mm (12 mils)

Eurocircuits class: 4A - Using min drill 1.1 mm for an OAR of 0.3 mm

General stats

Components count: (SMD/THT)

- Top: 25/1 (SMD + THT)

- Bottom: 0/0 (NONE)

Defined tracks:

Used tracks:

- 0.4 mm (16 mils) (156) defined: no

Defined vias:

Used vias:

Holes (excluding vias):

- 1.0 mm (39 mils) (2)

Oval holes:

Drill tools (including vias and computing adjusts and rounding):

- 1.1 mm (43 mils) (2)

Solder paste stats:

Using a paste with 87.75 % alloy, that has an specific gravity for the alloy of 7.4 g/cm³ and 1.0 g/cm³ for the flux. This paste has an specific gravity of 4.15 g/cm³.

The stencil thickness is 0.12 mm.

Side	Pads with paste	Area [mm²]	Paste [g]
Total	64	109.18	0.54

Note: this is just an approximation to the theoretical value. Margins of the solder mask and waste aren't computed.

Schematic

Schematic in SVG format

PCB Layers

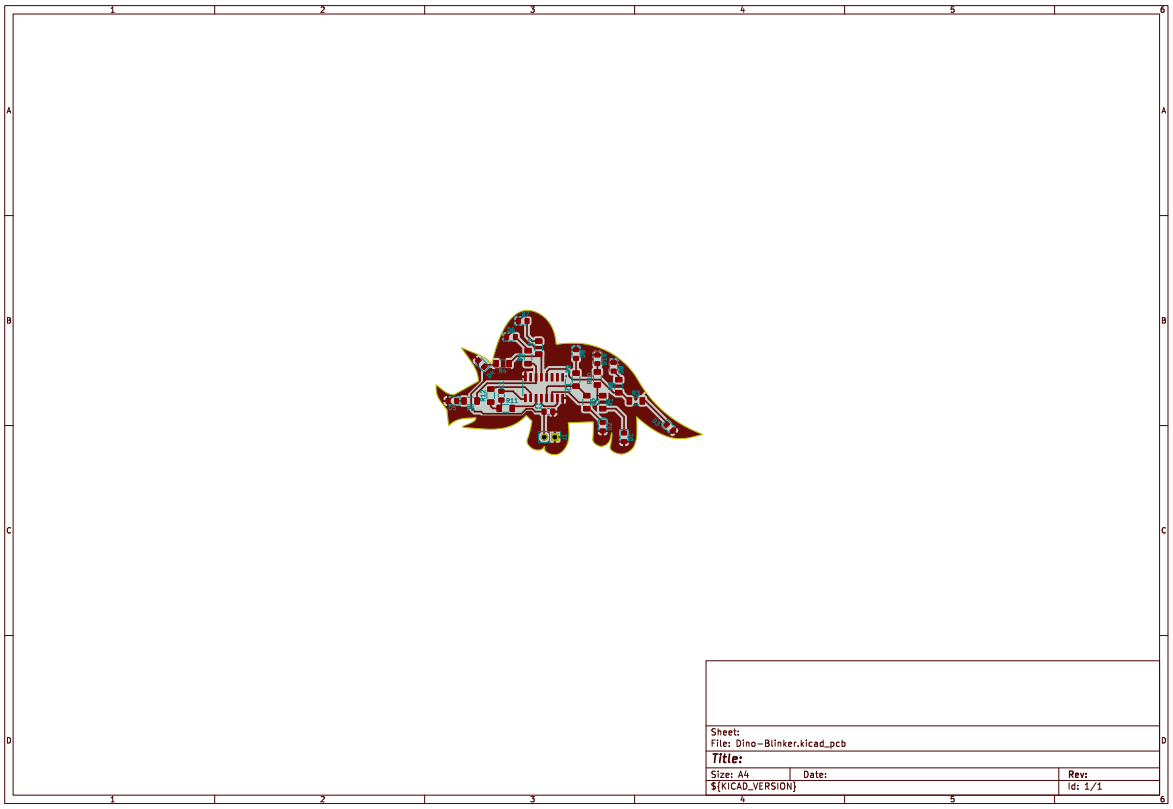

Figure 1: PCB Front copper

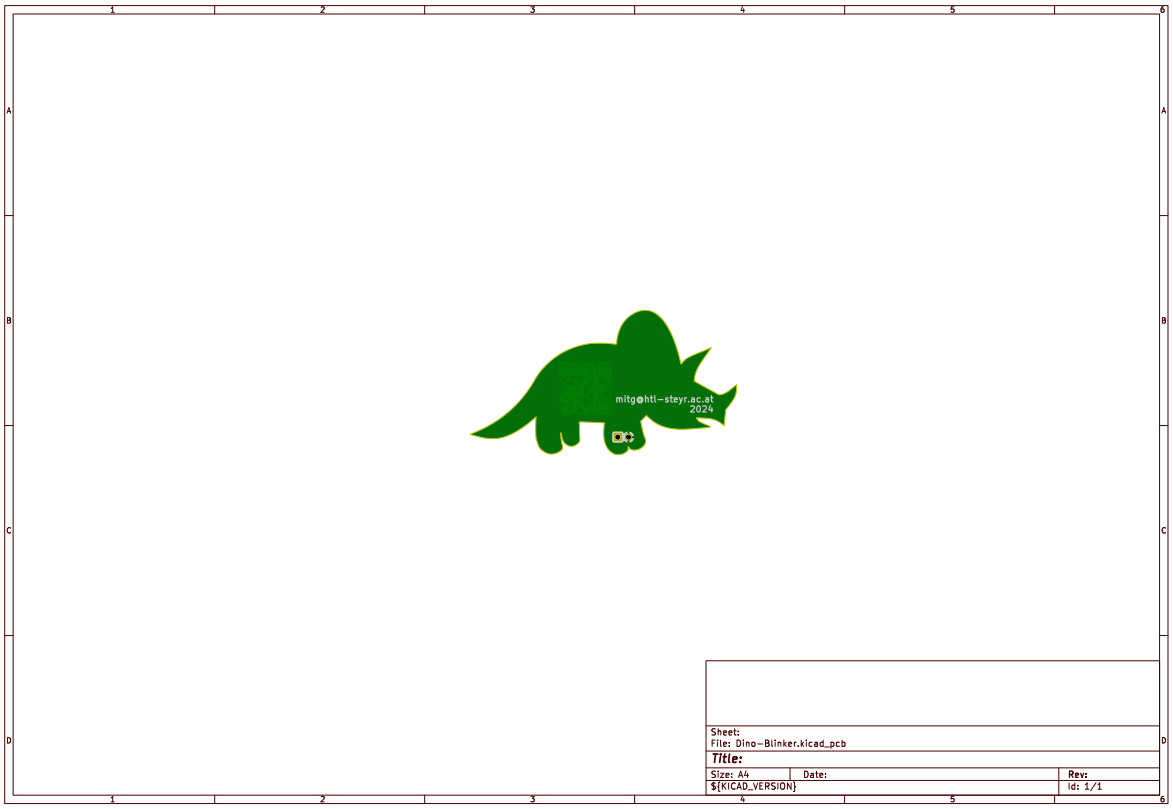

Figure 2: PCB Bottom copper

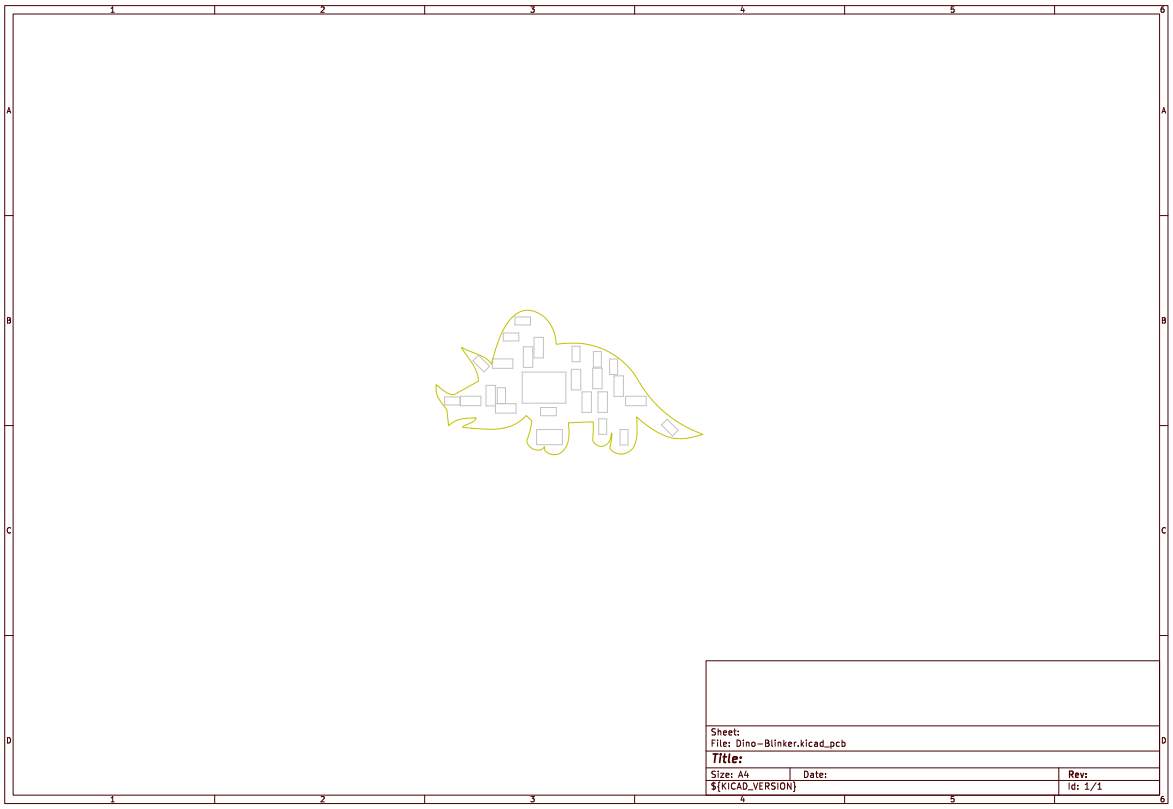

Figure 3: PCB Front courtyard area

Figure 4: PCB Front documentation