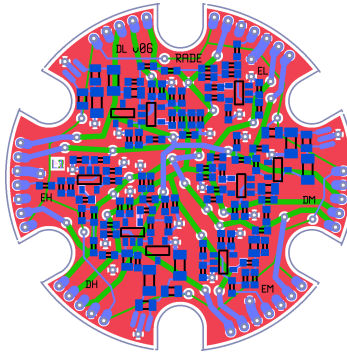


MSL RAD Front-End-Electronics

RADE PCB layout

Stephan I. Böttcher

v06s: June 25, 2007



The RADE board supports the readout of the neutron channel scintillator of the RAD instrument.

The board will be mounted below the neutron channel, with the backside facing the scintillator. The board shape is circular with a diameter of 46 mm, with six cutouts to accommodate the inner preload spring assembly. The board shall be staked to the upper spring plate, with a thin insulating layer of polyimide glass between the board and the plate.

Three neutron channel detectors will be connected with small flex strips, which are glued and wire-bonded to the detector. The other end of the strips will be staked to the RADE board and connected with short solder wires.

The power supply of the RADE electronics is routed via Teflon insulated wires (17-strand, AWG 32) from the RADF board through the anticoincidence scintillator to solder pads on the perimeter of the RADE board. A further set of solder pads supports the power supply wires to the RADD boards.

The electronics on the RADE board include three *Charge Sensitive Amplifiers* (CSA) and three *Fast Shapers* (FSH).

1 Fabrication

The board shall be made according to IPC 6012 class III standards.

1.1 Material

The base material is polyimide glass, Total board thickness $1\text{ mm} \pm 10\%$, with $35\text{ }\mu\text{m}$ copper thickness, surface finish HAL PbSn, with solder mask and silk on the front side only.

1.2 Design Rules

The minimum trace width is 0.2 mm (8 mil), clearance 0.2 mm (8 mil), the clearance on top surfaces to copper carrying -70 V shall be 0.3 mm (12 mil).

The minimum via hole diameter is 0.4 mm (16 mil), with annular rings of at least 0.25 mm (10 mil) width.

The edge clearance of copper traces and pads is 0.5 mm (20 mil) minimum.

1.3 Layers

The board has four copper layers, with SMD components on one side. The backside is filled with a ground plane.

The inner layers are a ground plane and a power routing layer. The layer order is front, ground, power, back. The ground plane has cutouts under the inverting input nodes of the current feedback amplifiers (AD 8005).

Layer order	Gerber file
1 front layer copper	v06s.group0.gbr
2 ground layer copper	v06s.group3.gbr
3 power layer copper	v06s.group4.gbr
4 back layer copper	v06s.group1.gbr
board shape	v06s.group2.gbr
front solder mask	v06s.frontmask.gbr
front silk screen	v06s.frontsilk.gbr
drill file	v06s.plated-drill.cnc
back solder mask	(v06s.backmask.gbr)
front solder paste	(v06s.frontpaste.gbr)
fab, please ignore	(v06s.fab.gbr)

The files in parenthesis are PCB tool output not relevant to this board's fabrication.

2 Design

2.1 Tools

The design is done with GNU EDA tools. (<http://www.geda.seul.org/>)
The layout tool is PCB version 20060822. GAF version is 20060824.

2.2 Schematics

The schematics for the CSA and FSH were derived from master schematics (Fig. 2 and 3) by a script which prefixes net names and reference designators with an instance prefix, L1_, L2_, and L3_ for the CSAs, and S1_, S2_, and S3_ for the shapers. The RADE schematics (Fig. 1) defines connections between the amplifier instances, and to the solder pads.

2.3 Layout

Fig. 4 to 7 show the copper layer layouts, Fig. 8 shows the reference designators of all parts.

Each layer is shown at scale 1:1 and expanded to text width.

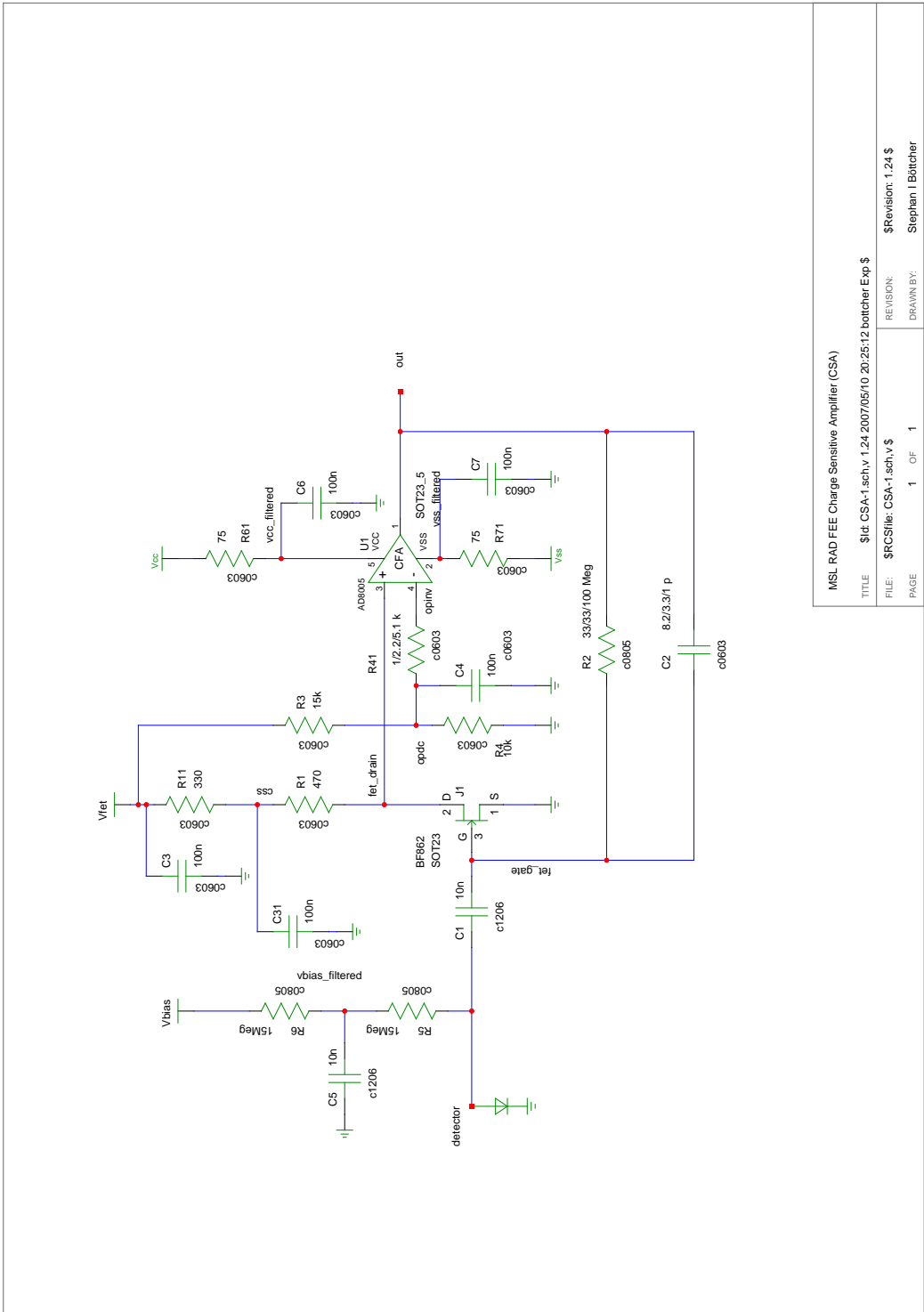


Figure 2: Charge sensitive amplifier schematics: CSA-1.sch

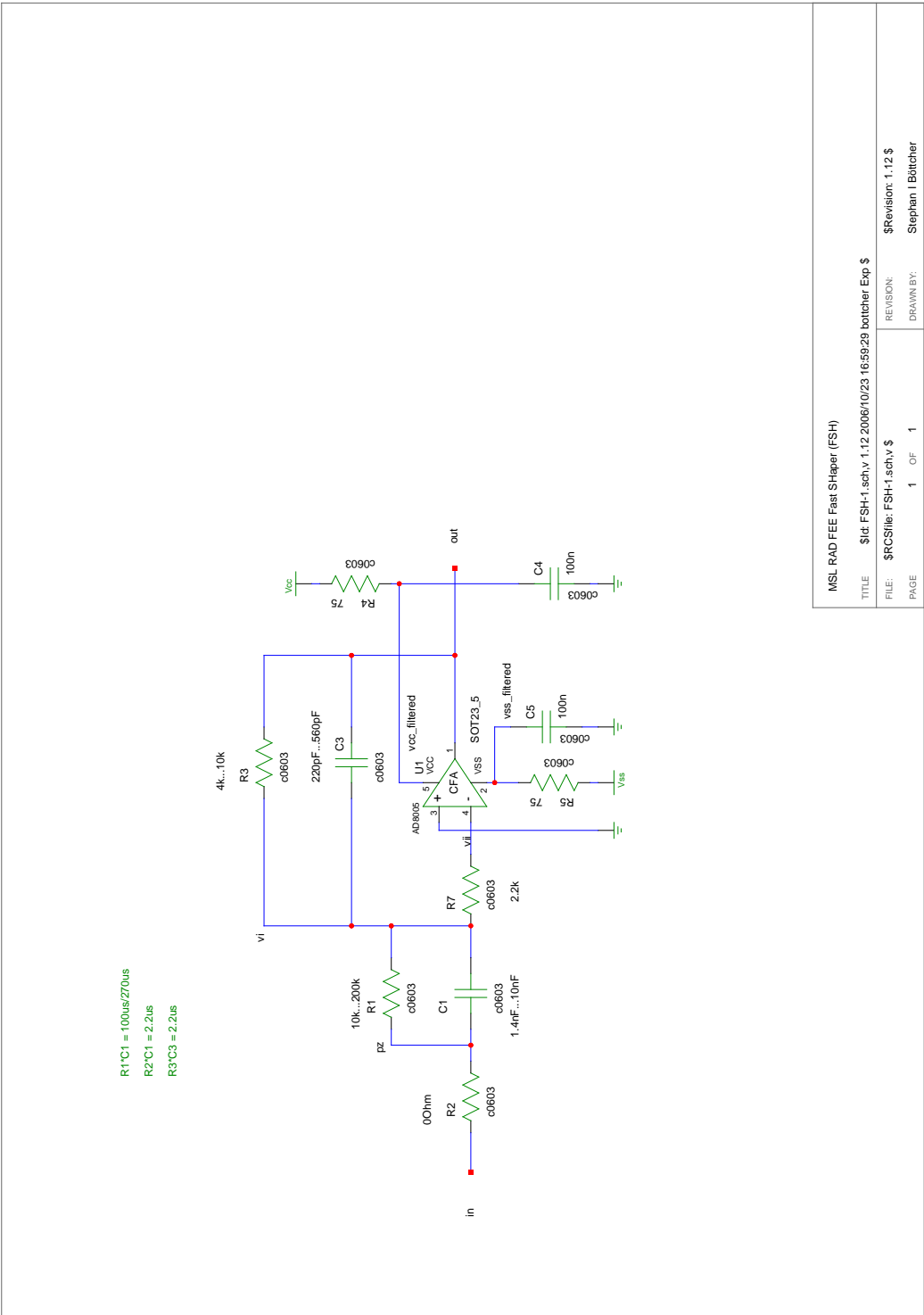


Figure 3: Fast shaper schematics: FSH-1.sch

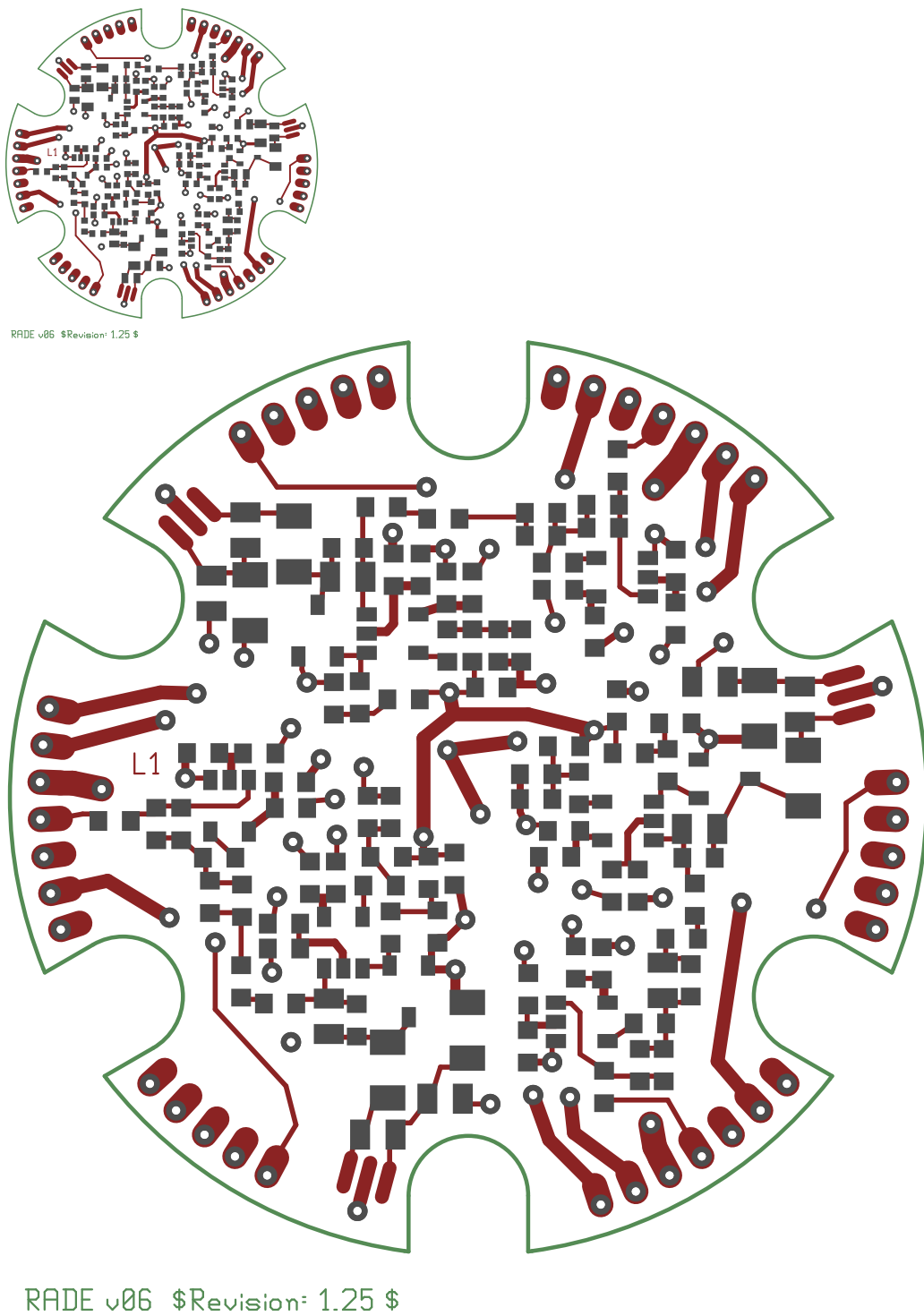
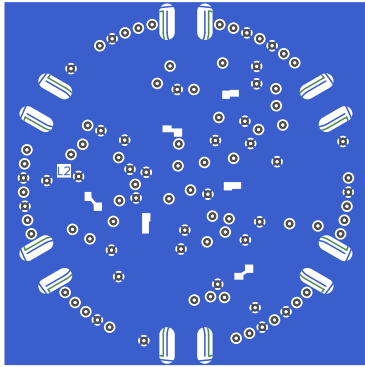
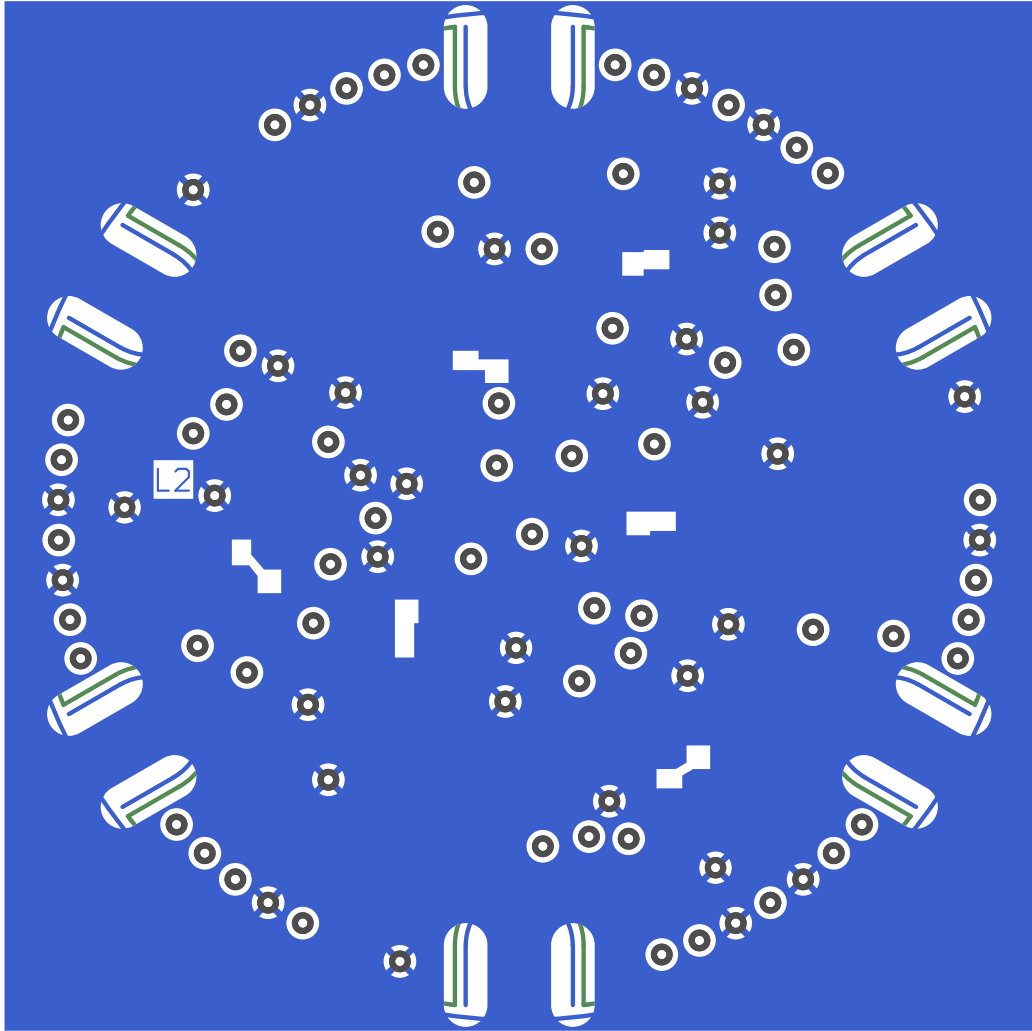


Figure 4: Front layer.



RADE v06 \$Revision: 1.25 \$



RADE v06 \$Revision: 1.25 \$

Figure 5: Ground layer.

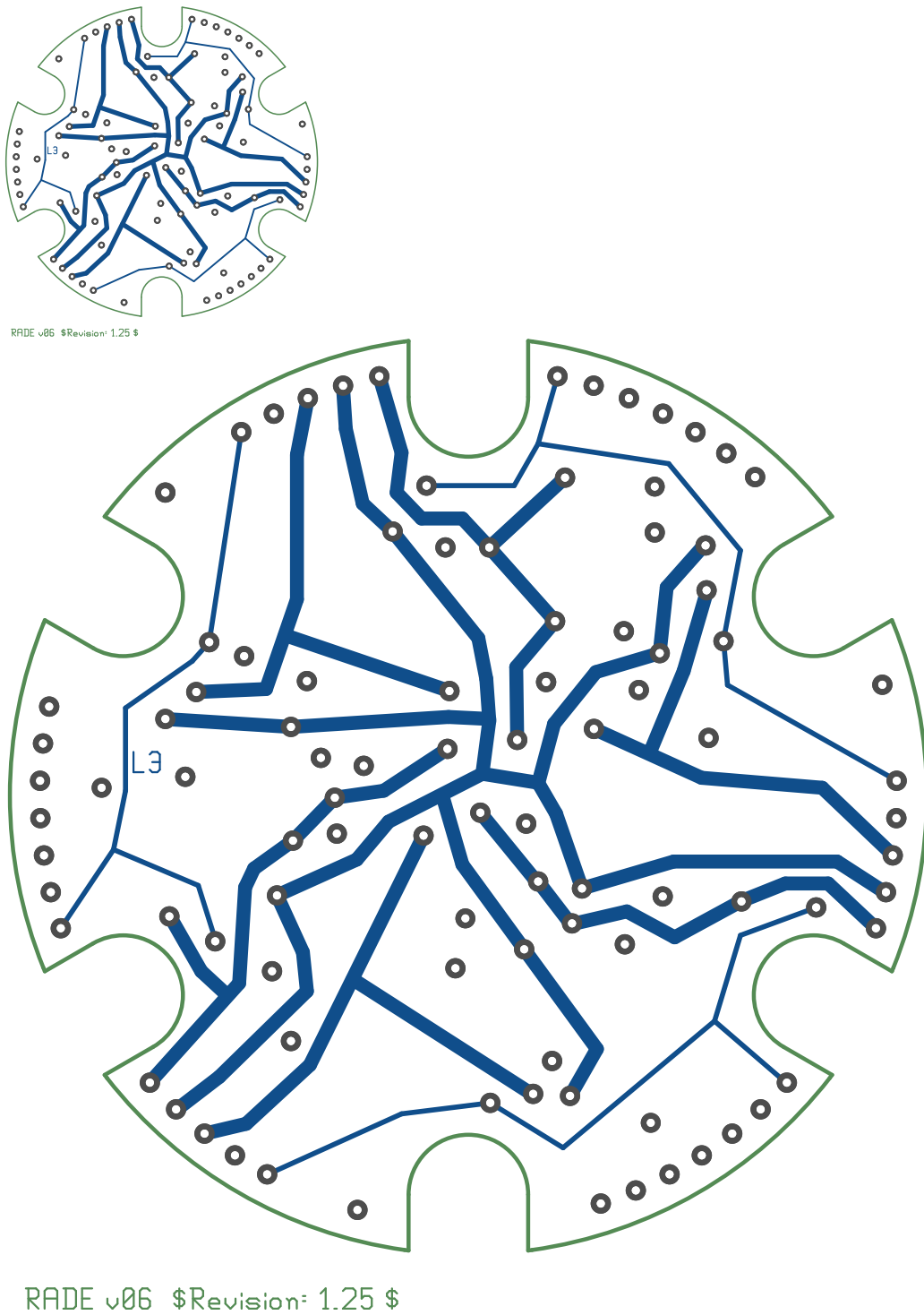


Figure 6: Power layer.

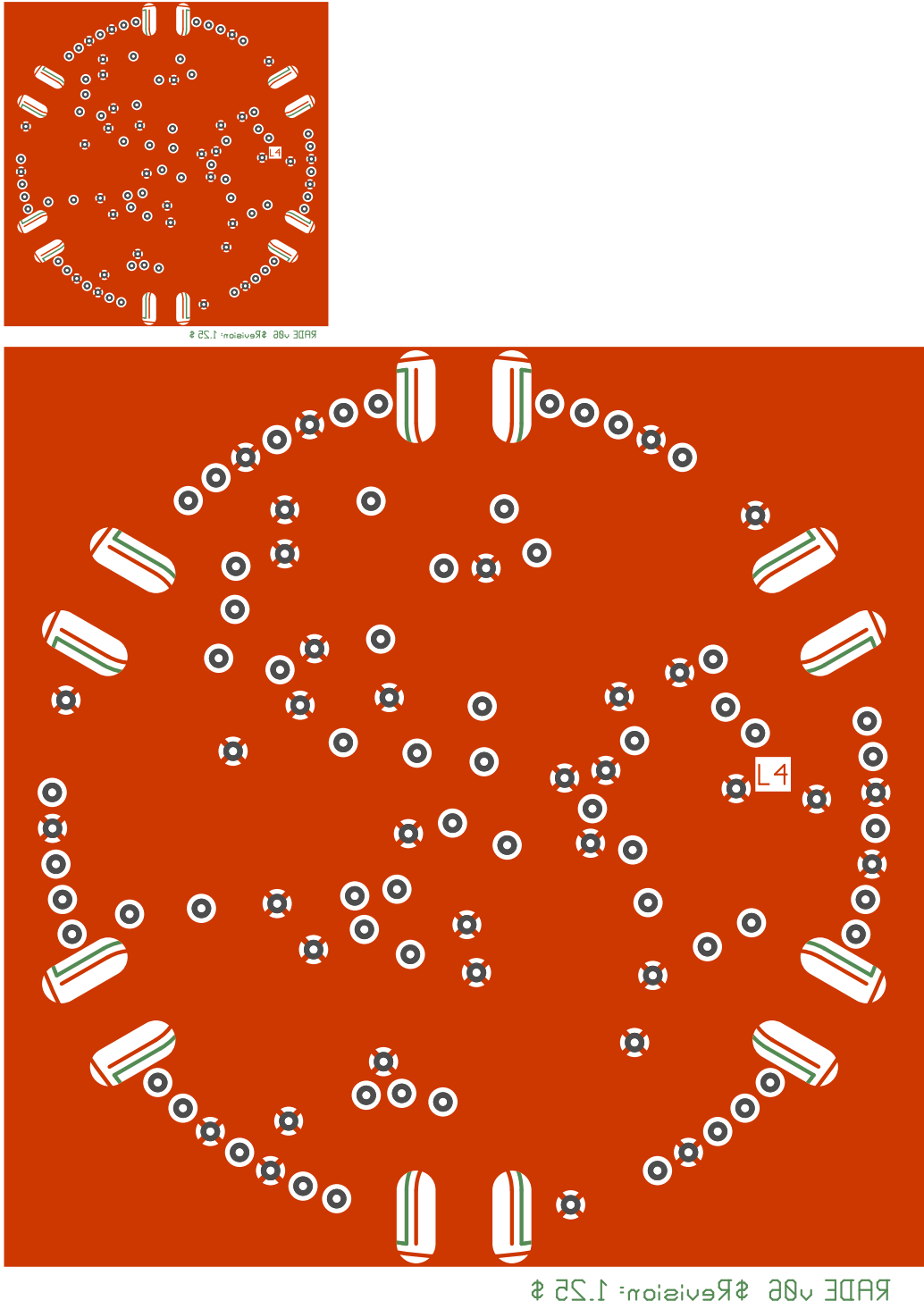


Figure 7: Back layer.

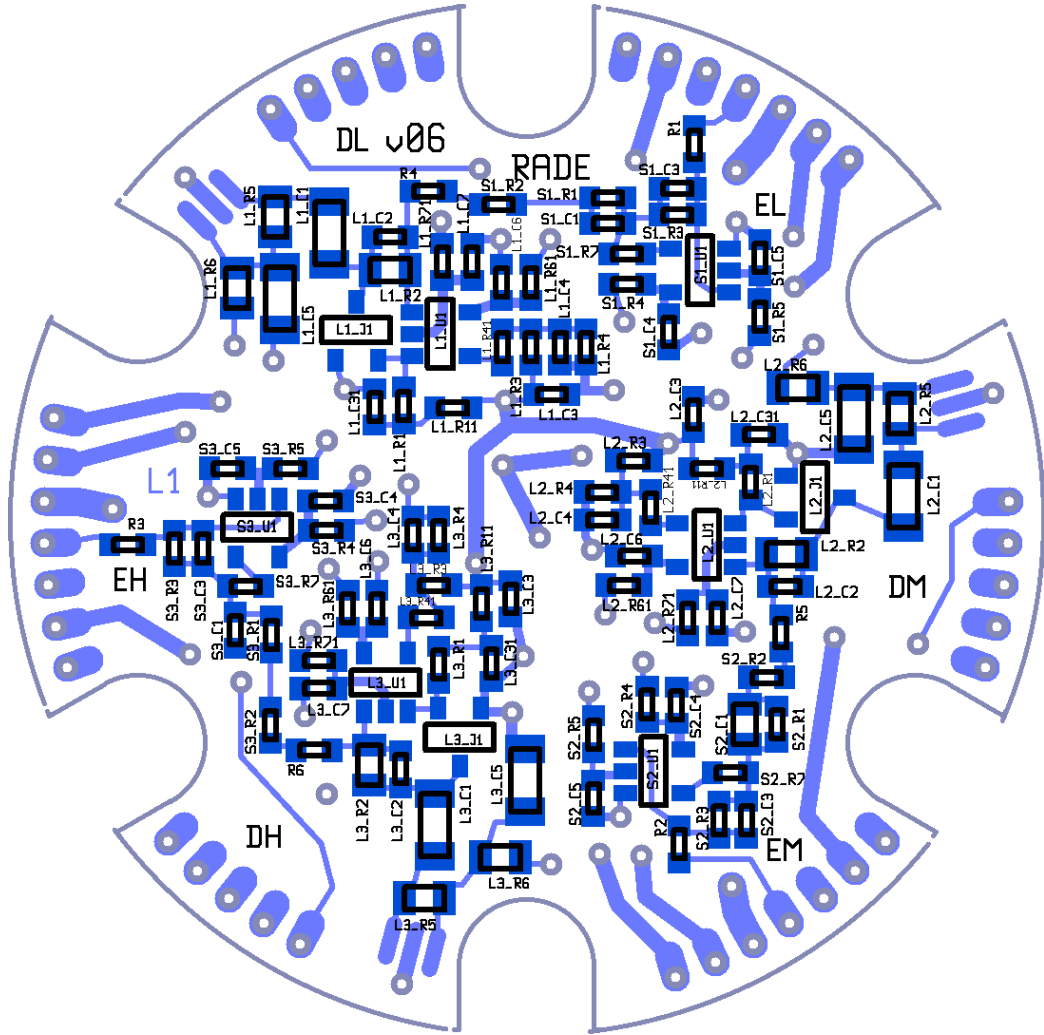


Figure 8: Assembly.