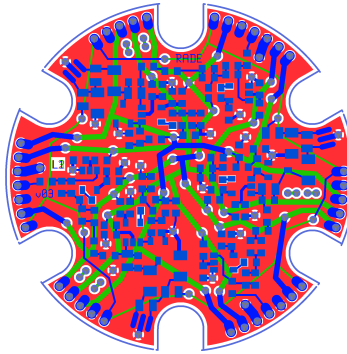


# MSL RAD Front-End-Electronics

## RADE PCB layout

Stephan I. Böttcher

v03: November 7, 2006



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 an insulating layer **TBD** 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 3 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 on the front side only, *no* silk screens.

## 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  $-70\text{ V}$  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	v03.group0.gbr
2 ground layer copper	v03.group3.gbr
3 power layer copper	v03.group4.gbr
4 back layer copper	v03.group1.gbr
board shape	v03.group2.gbr
front solder mask	v03.frontmask.gbr
drill file	v03.plated-drill.cnc
back solder mask	(v03.backmask.gbr)
front solder paste	(v03.frontpaste.gbr)
back solder paste	(v03.backpaste.gbr)
front silk screen	(v03.frontsilk.gbr)
back silk screen	(v03.backsilk.gbr)
fab, please ignore	(v03.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 silk layer (not to be printed on the board).

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



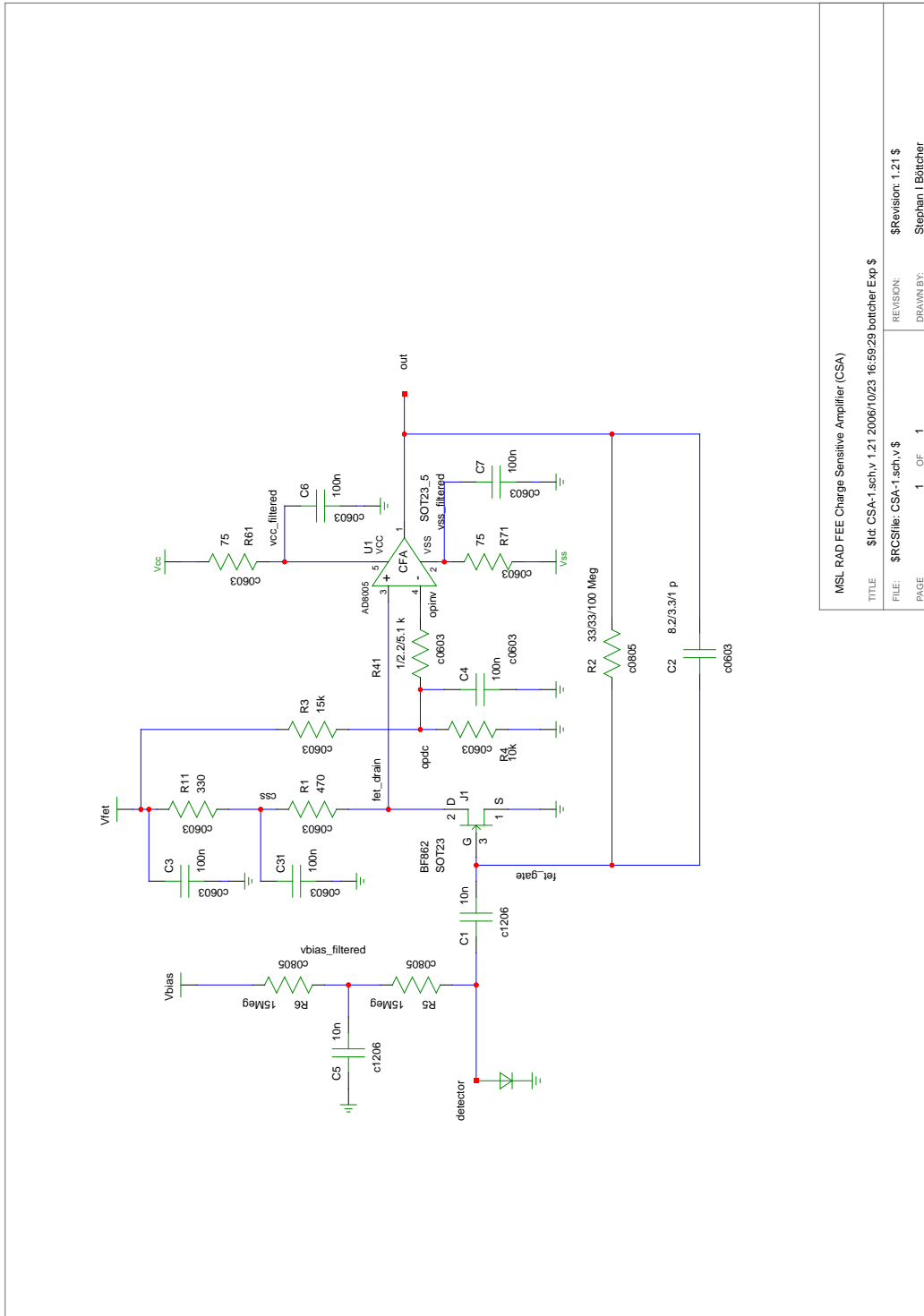


Figure 2: Charge sensitive amplifier schematics: CSA-1.sch

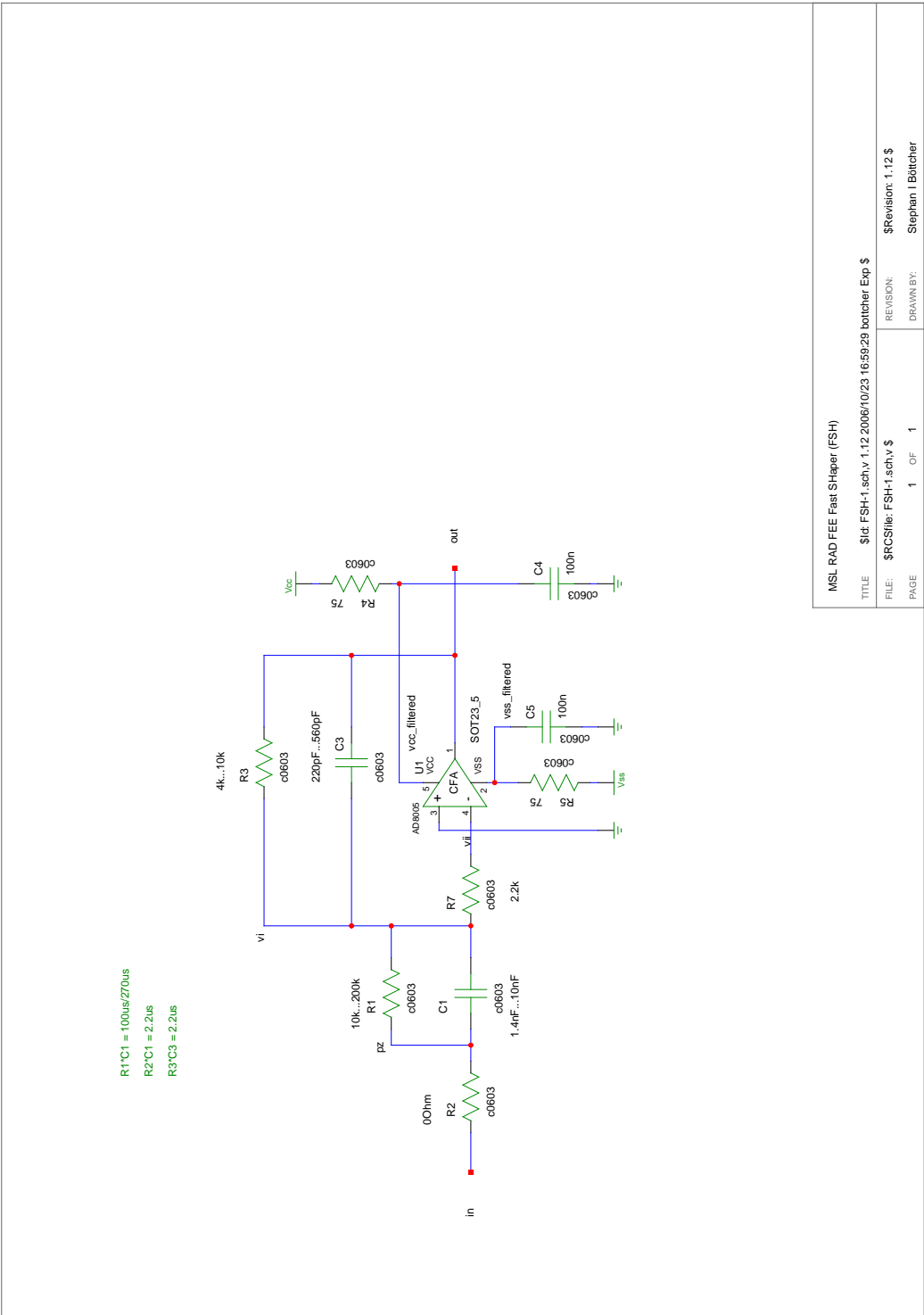


Figure 3: Fast shaper schematics: FSH-1.sch

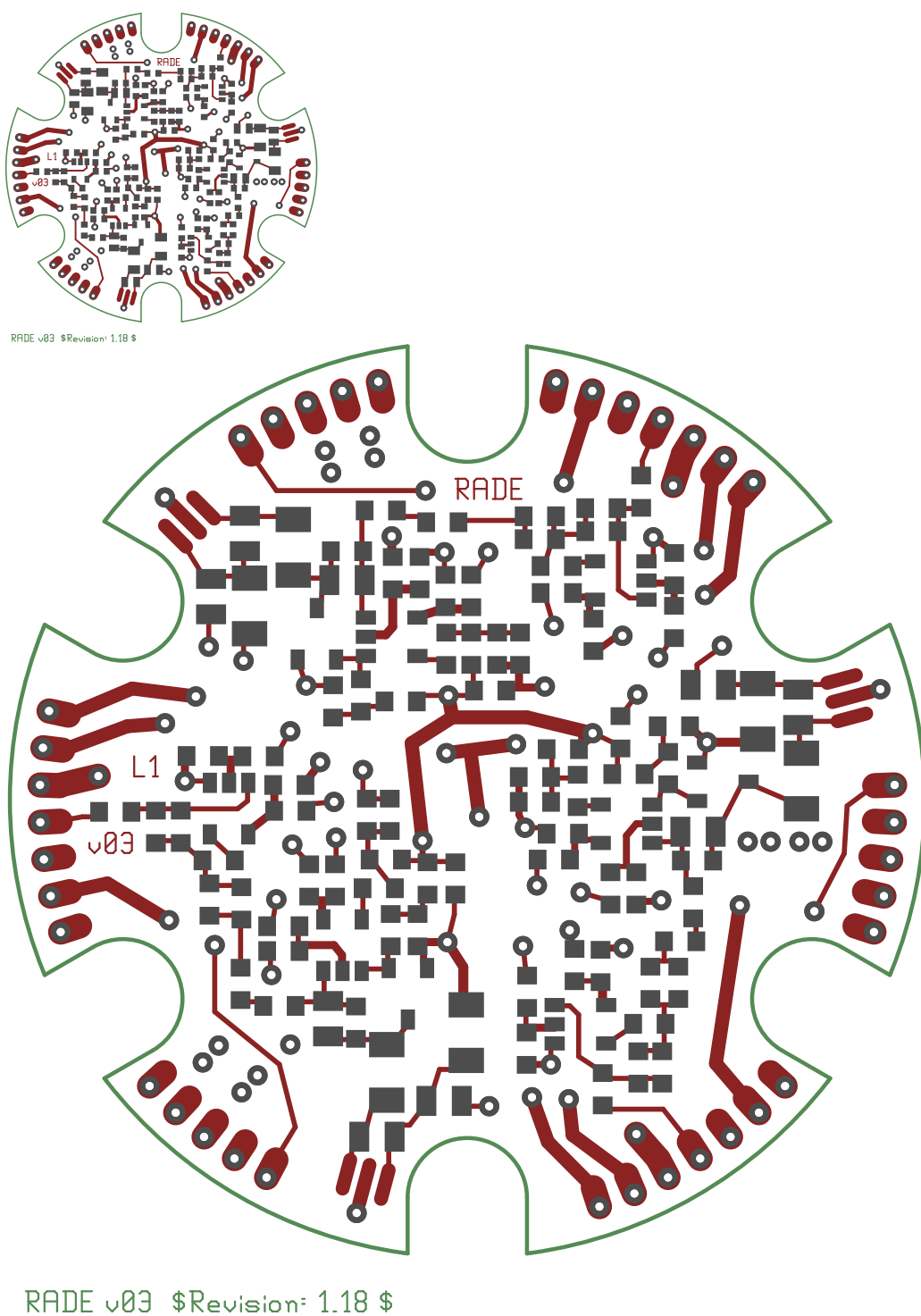
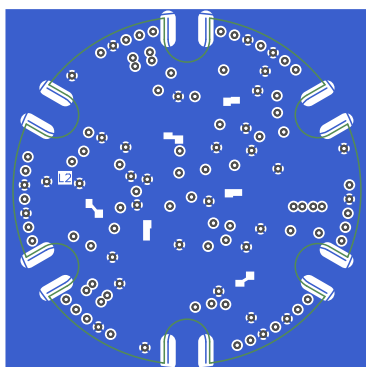
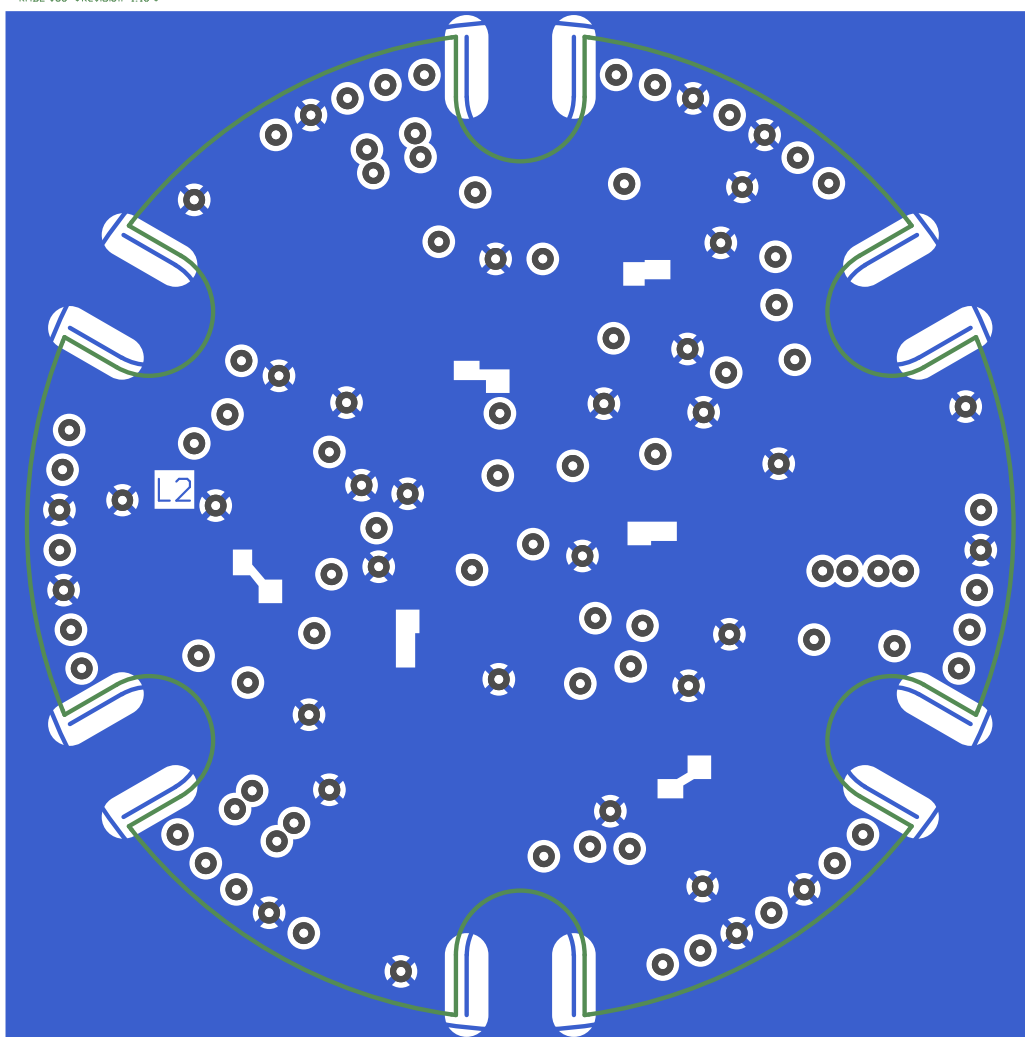


Figure 4: Front layer copper



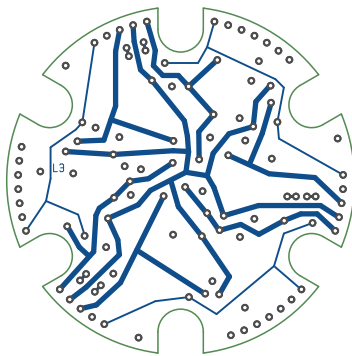
RADE v03 \$Revision: 1.18 \$



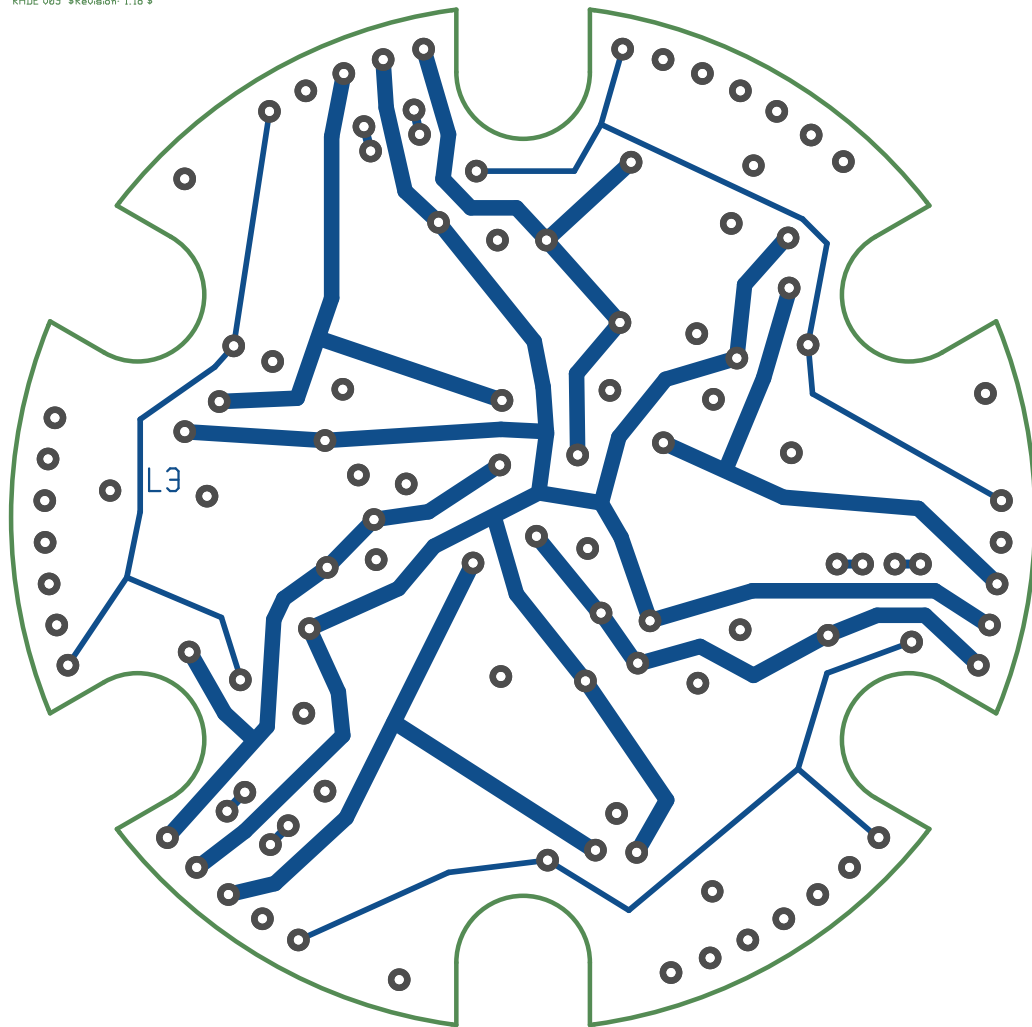
RADE v03 \$Revision: 1.18 \$

Figure 5: Ground layer copper





RADE v03 \$Revision: 1.18 \$



RADE v03 \$Revision: 1.18 \$

Figure 6: Power layer copper

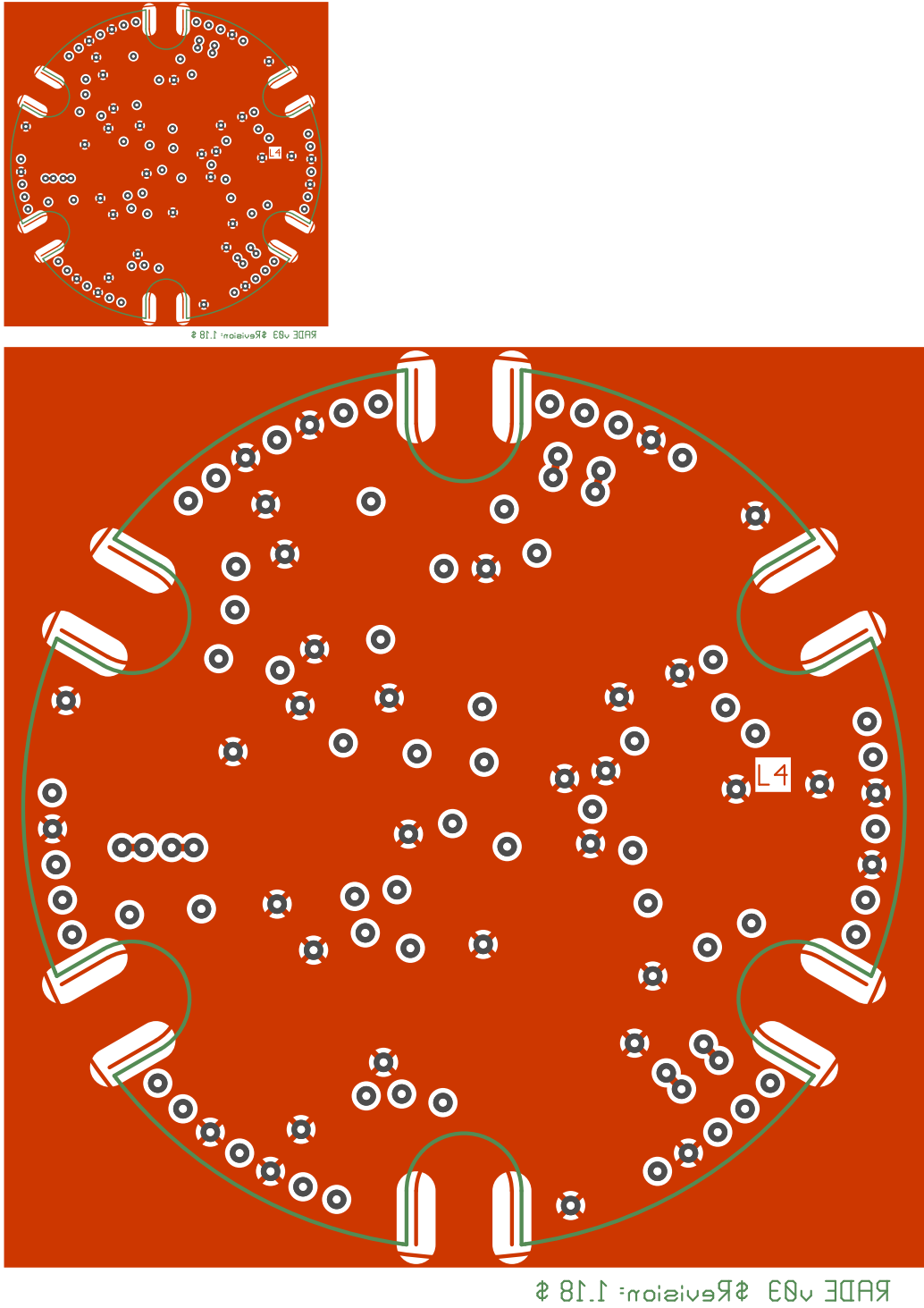


Figure 7: Back layer copper

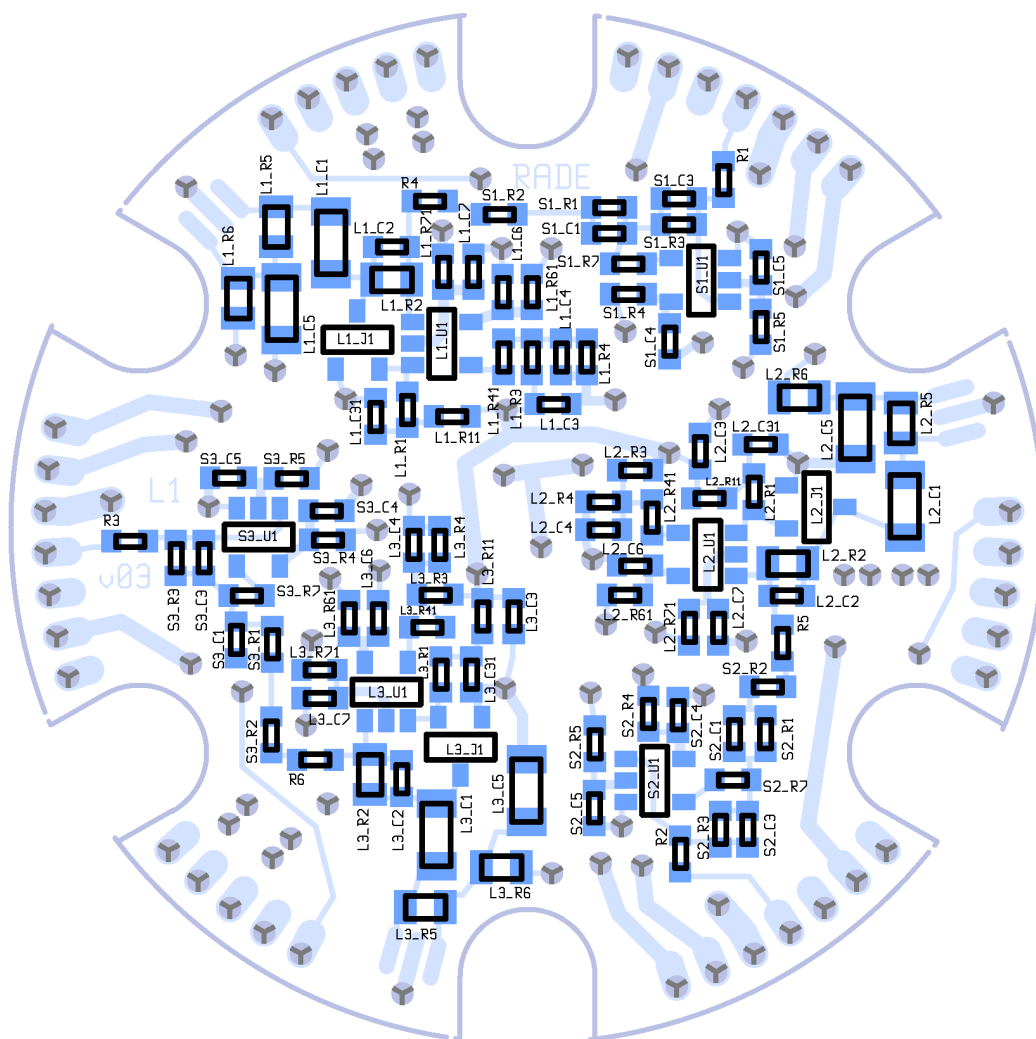


Figure 8: Assembly