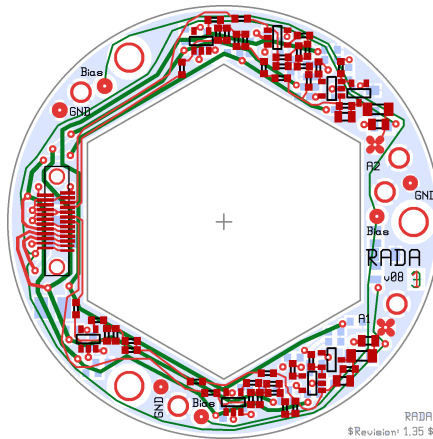


MSL RAD Front-End-Electronics

RADA PCB layout

Stephan I. Böttcher

v08s: June 27, 2007



The RADA board supports the readout of the topmost *Solid State Detector* (SSD) of the RAD instrument.

The board will be mounted in close proximity to the detector, with the backside facing the detector, via three mounting holes.

The particle telescope field of view runs through a hole in the center of the board. The board's outer shape is circular with a diameter of 57 mm, and the central hole is hexagonal with a flat to flat width of 36 mm.

The detector is connected via Teflon insulated wires (AWG 32). These wires will be routed through holes in the RADA board and soldered to pads on the front side. The RADA board connects to a flex strip via an Omnetics Dualobe connector, 25-pin, vertical surface mount receptacle.

The electronics on the RADA board include two *Charge Sensitive Amplifiers* (CSA), four *Fast Shapers* (FSH), a separate detector bias filter for the detector guard ring, a thermistor, and a FET power filter.

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.

Solder masks on both sides conforming to IPC-SM-840 Class III.

The silk screens on both sides shall be epoxy-based material.

The provided Gerber for the silkscreens must be masked by the respective soldermasks to prevent printing on the solder pads.

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 both sides. All components which carry AC signal are located on the top 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).

The provided silkscreen layers shall be masked by the soldermask layers by the board house, to prevent silk on the solder pads.

Layer order	Gerber file
1 front layer copper	v08s.group0.gbr
2 ground layer copper	v08s.group3.gbr
3 power layer copper	v08s.group4.gbr
4 back layer copper	v08s.group1.gbr
board shape	v08s.group2.gbr
1 front solder mask	v08s.frontmask.gbr
4 back solder mask	v08s.backmask.gbr
1 front silk screen	v08s.frontsilk.gbr
4 back silk screen	v08s.backsilk.gbr
drill file	v08s.plated-drill.cnc
1 front solder paste	(v08s.frontpaste.gbr)
4 back solder paste	(v08s.backpaste.gbr)
fab, please ignore	(v08s.fab.gbr)

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

1.4 Outline

The central hexagonal cutout shall be made with a corner radius not larger than 1 mm.

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_ and L2_ for the CSAs, and S1L_, S1H_, S2L_, and S2H_ for the shapers. The RADA schematics (Fig. 1) defines connections between the amplifier instances and to the connector, as well as the bias filter and the thermistor.

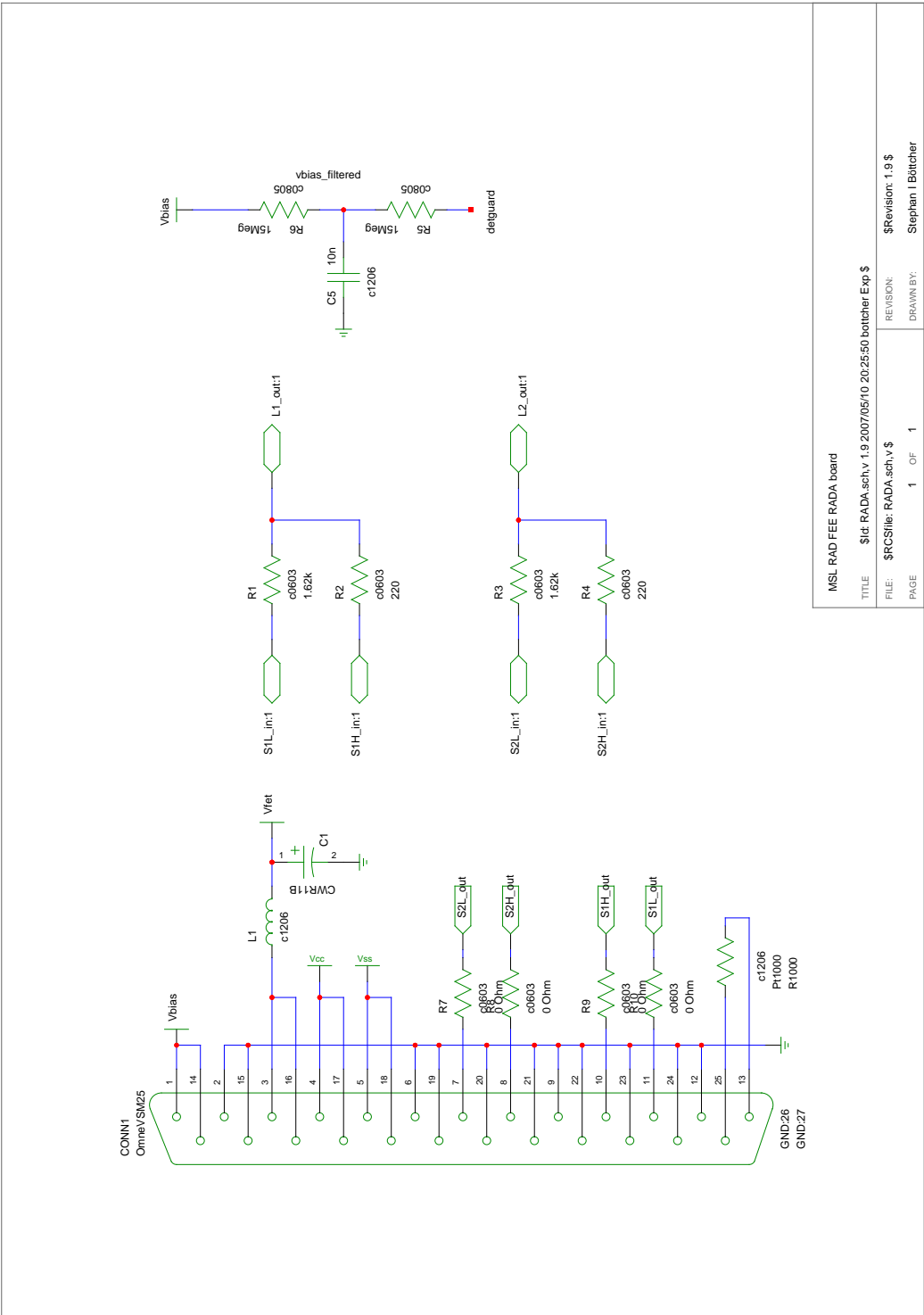
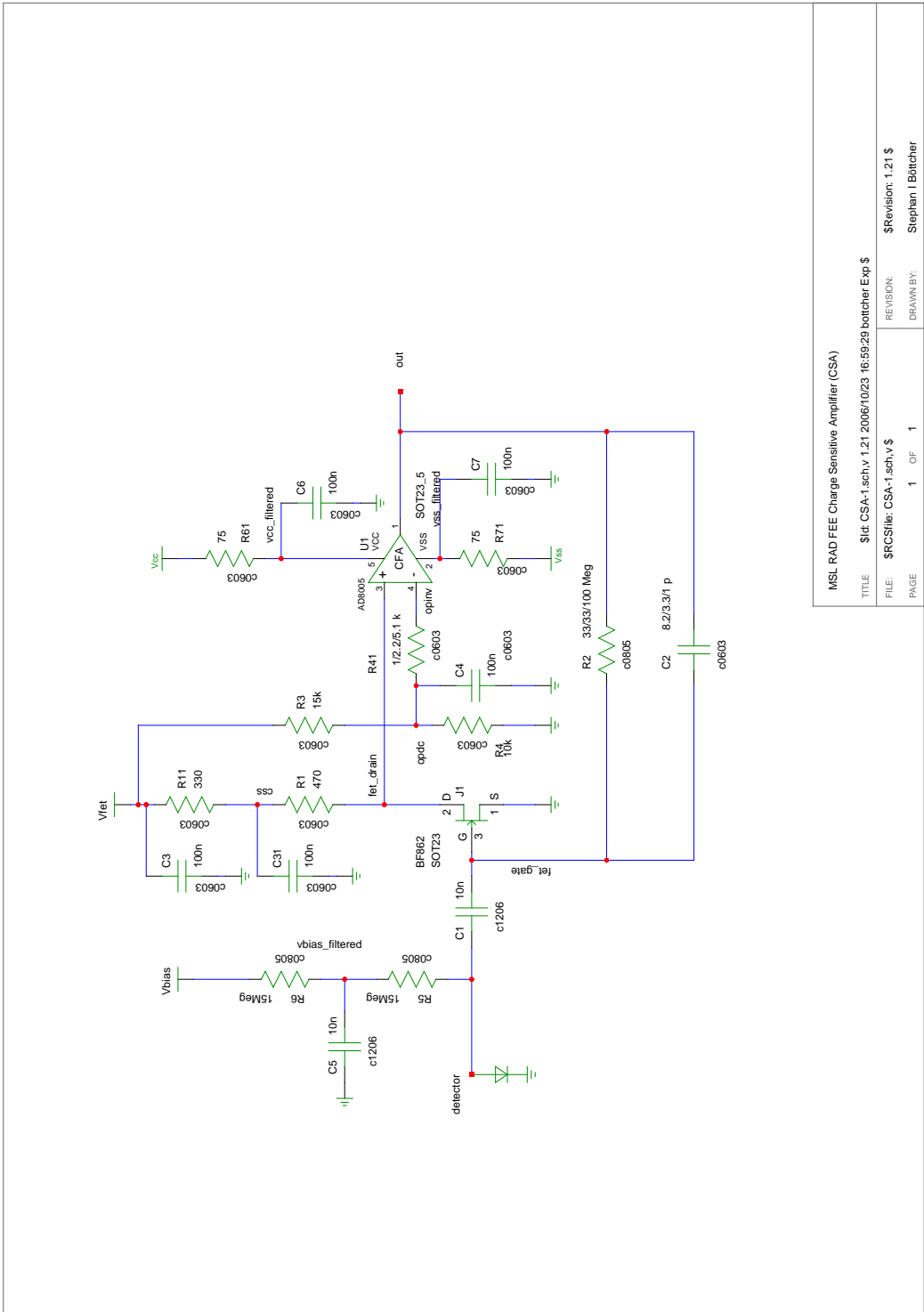


Figure 1: RADA master schematics: RADA.sch



MSL RAD FEE Charge Sensitive Amplifier (CSA)

TITLE: \$Id: CSA-1.sch,v 1.21 2006/10/23 16:59:29 bortcher Exp \$

FILE: \$RCSfile: CSA-1.sch,v \$

REVISION:

\$Revision: 1.21 \$

PAGE 1 OF 1

DRAWN BY: Stephan I. Bortcher

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

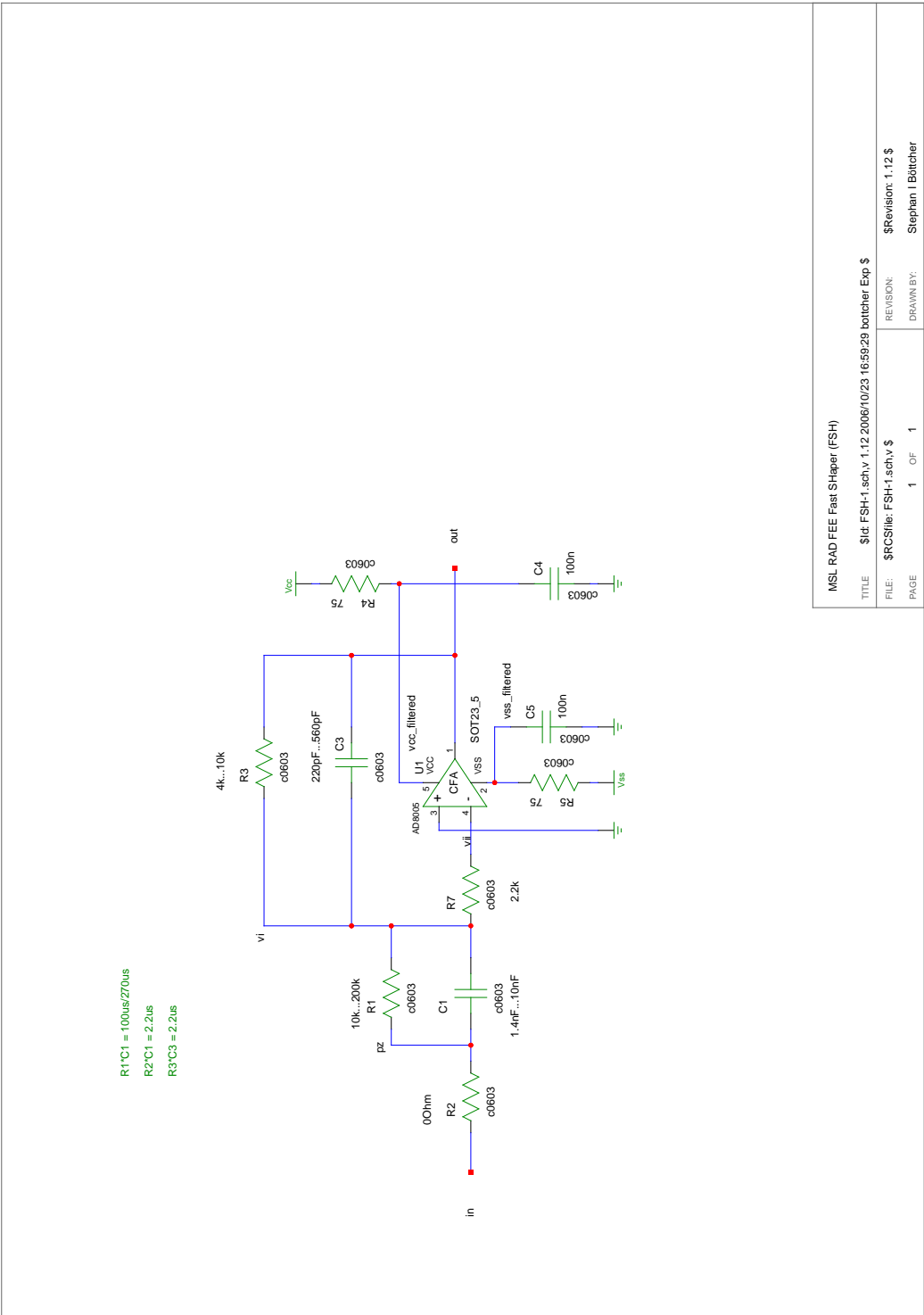


Figure 3: Fast shaper schematics: FSH-1.sch

2.3 Layout

Fig. 4 to 7 show the copper layer layouts, Fig. 8 and Fig., 9 show front and back pads with silk screen layers.

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

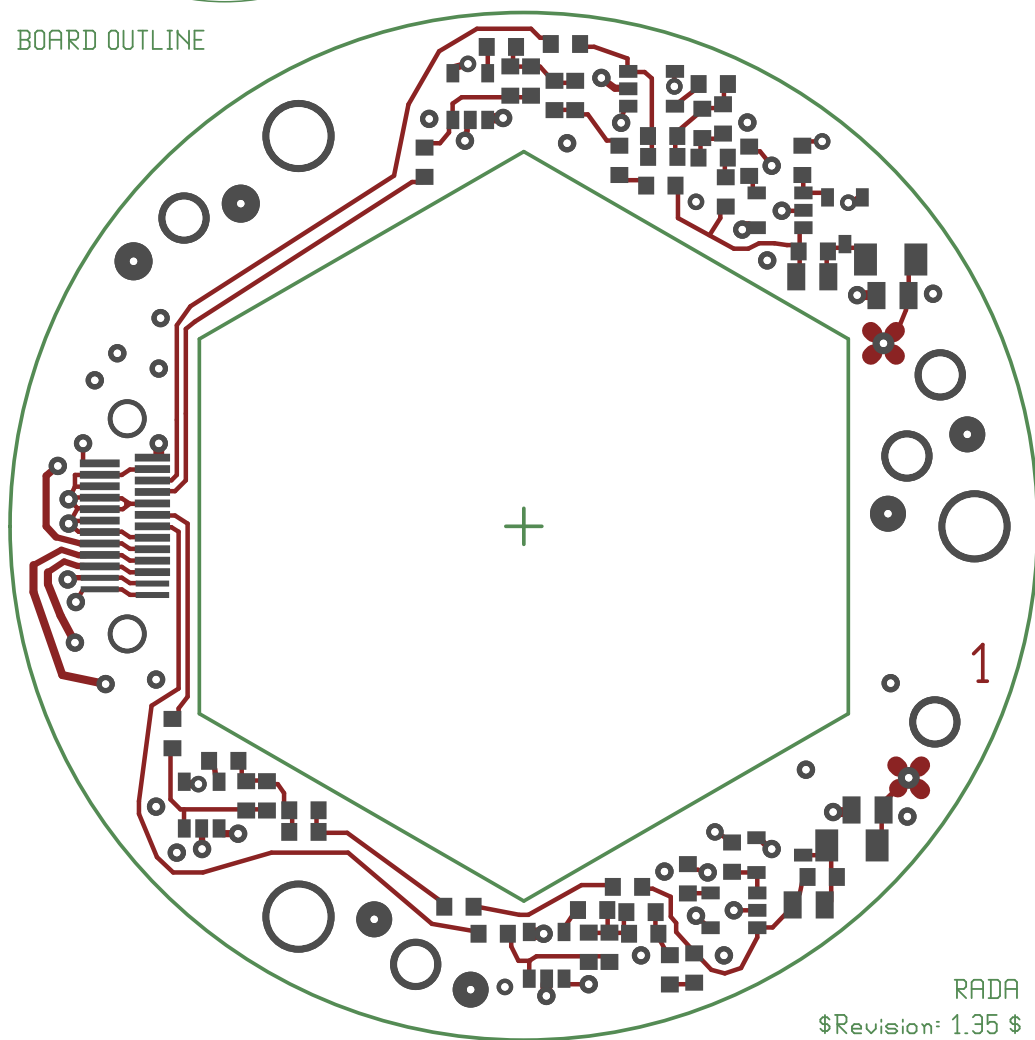
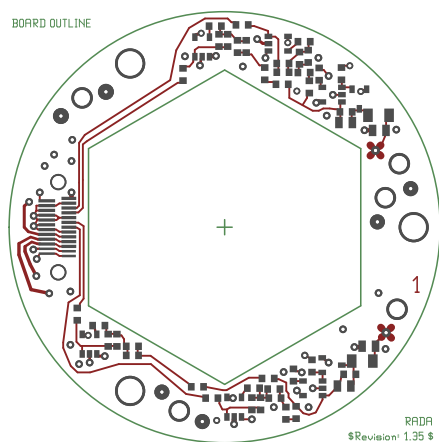


Figure 4: Front layer.

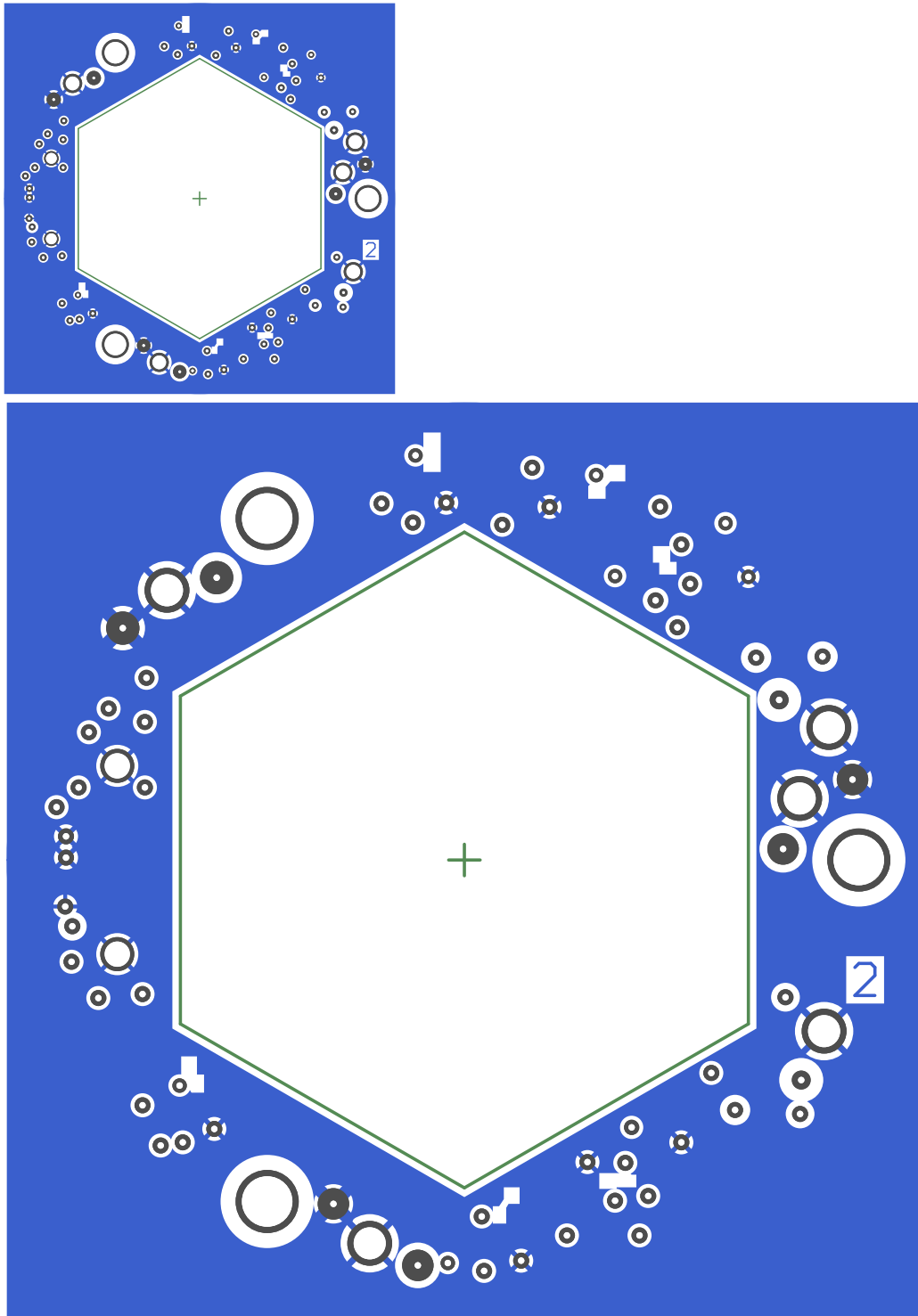


Figure 5: Ground layer. (The postscript exporter did not do the clearance on the Arcs.)

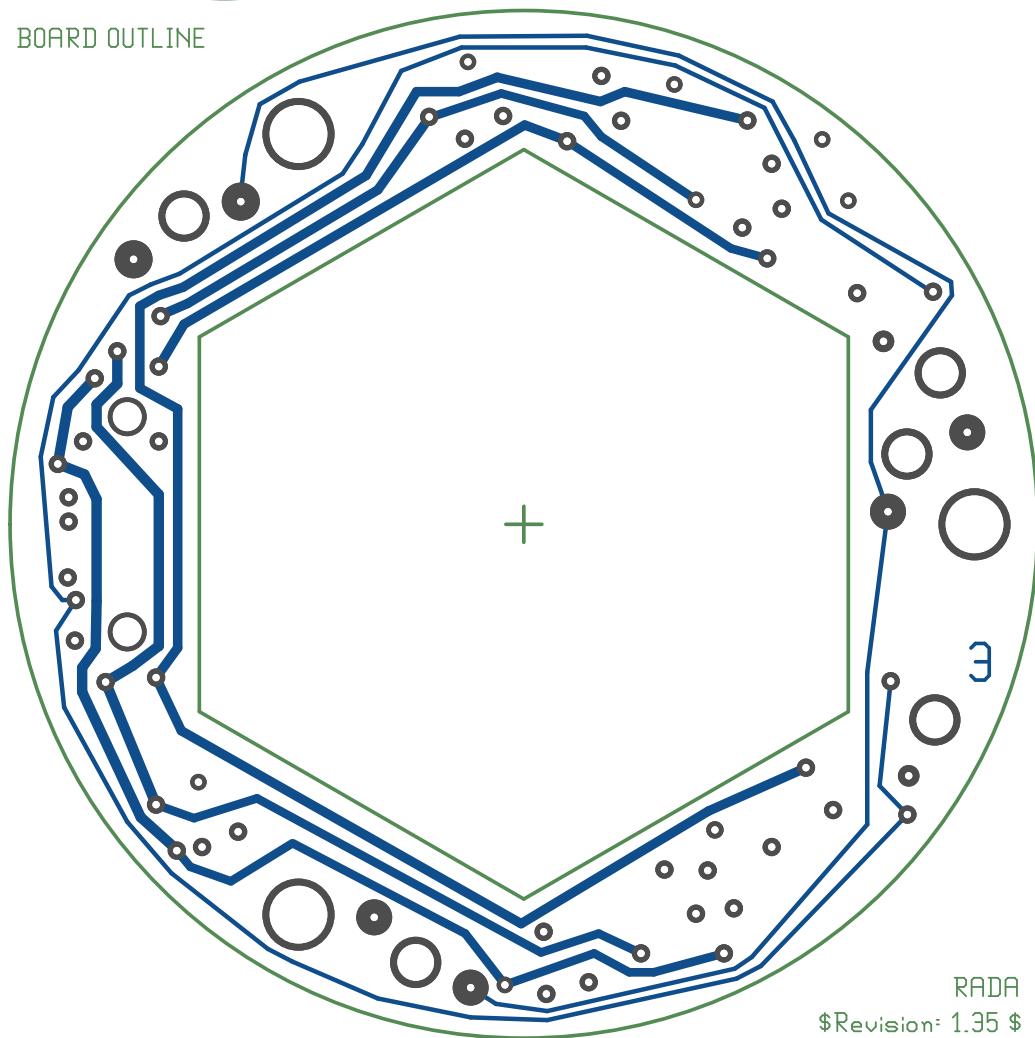
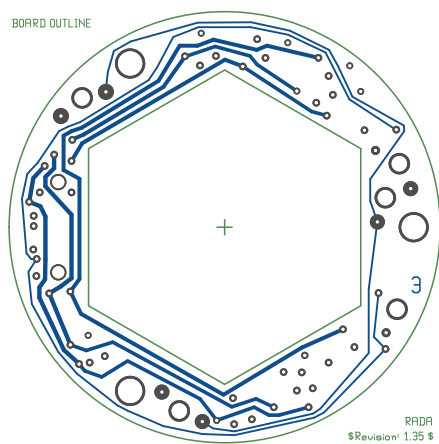


Figure 6: Power layer.

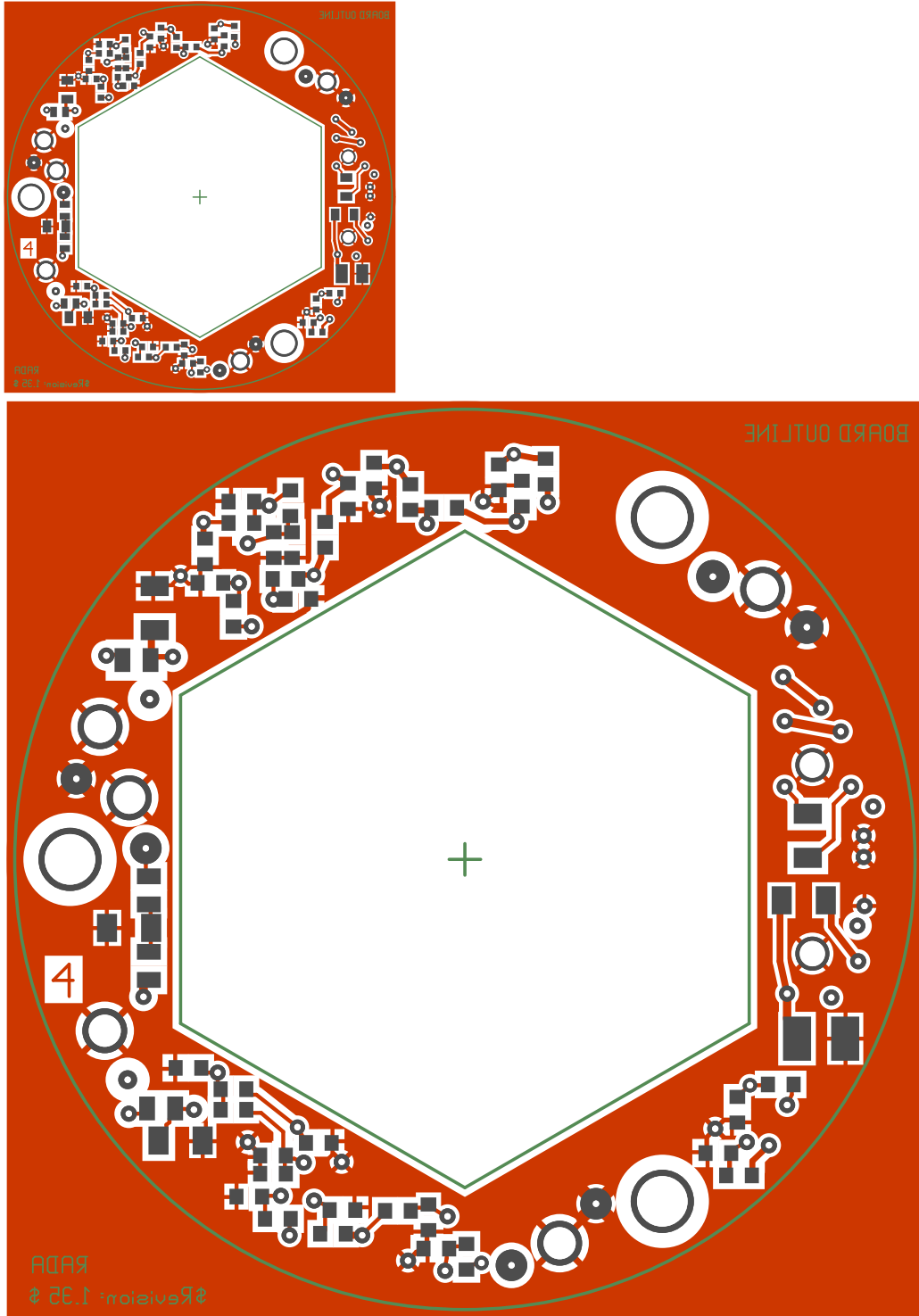


Figure 7: Back layer. (The postscript exporter did not do the clearance on the Arcs.)

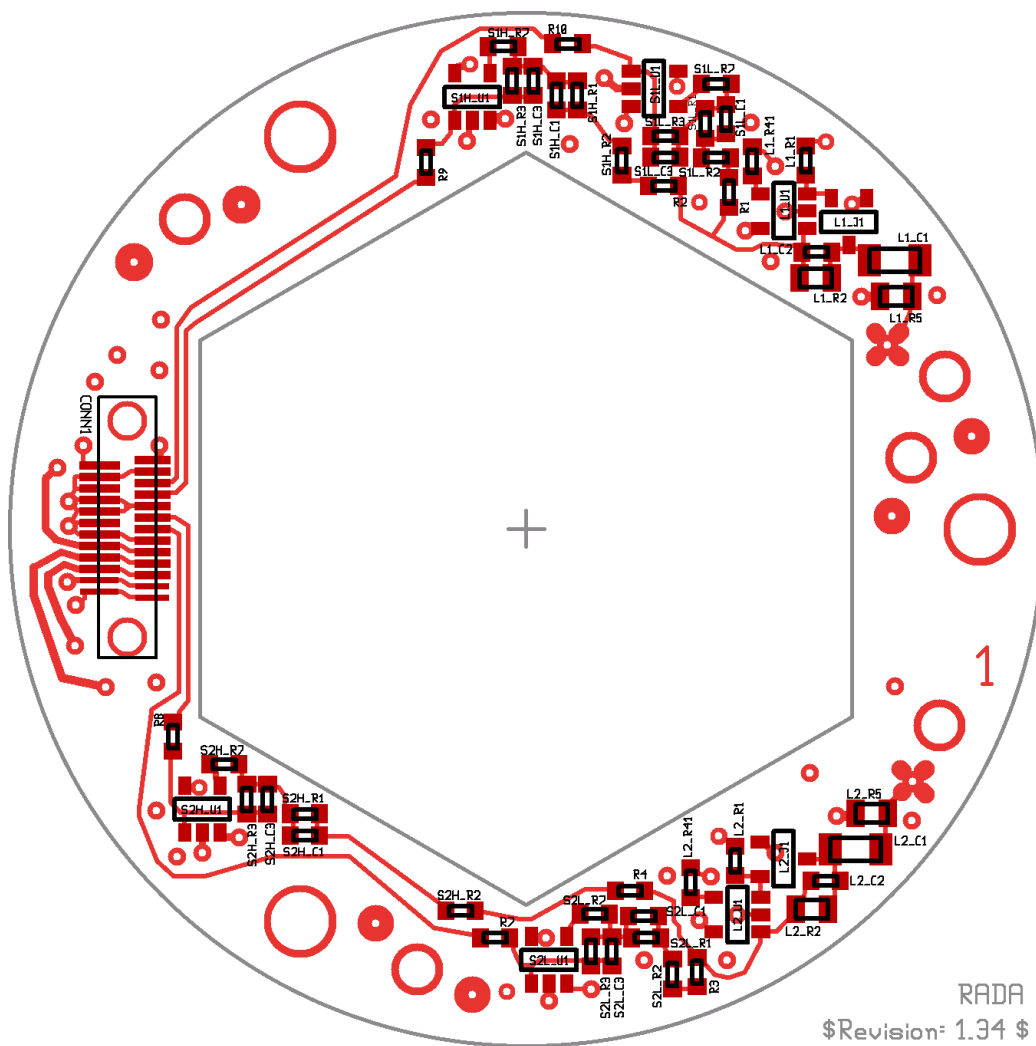


Figure 8: Front layer assembly.

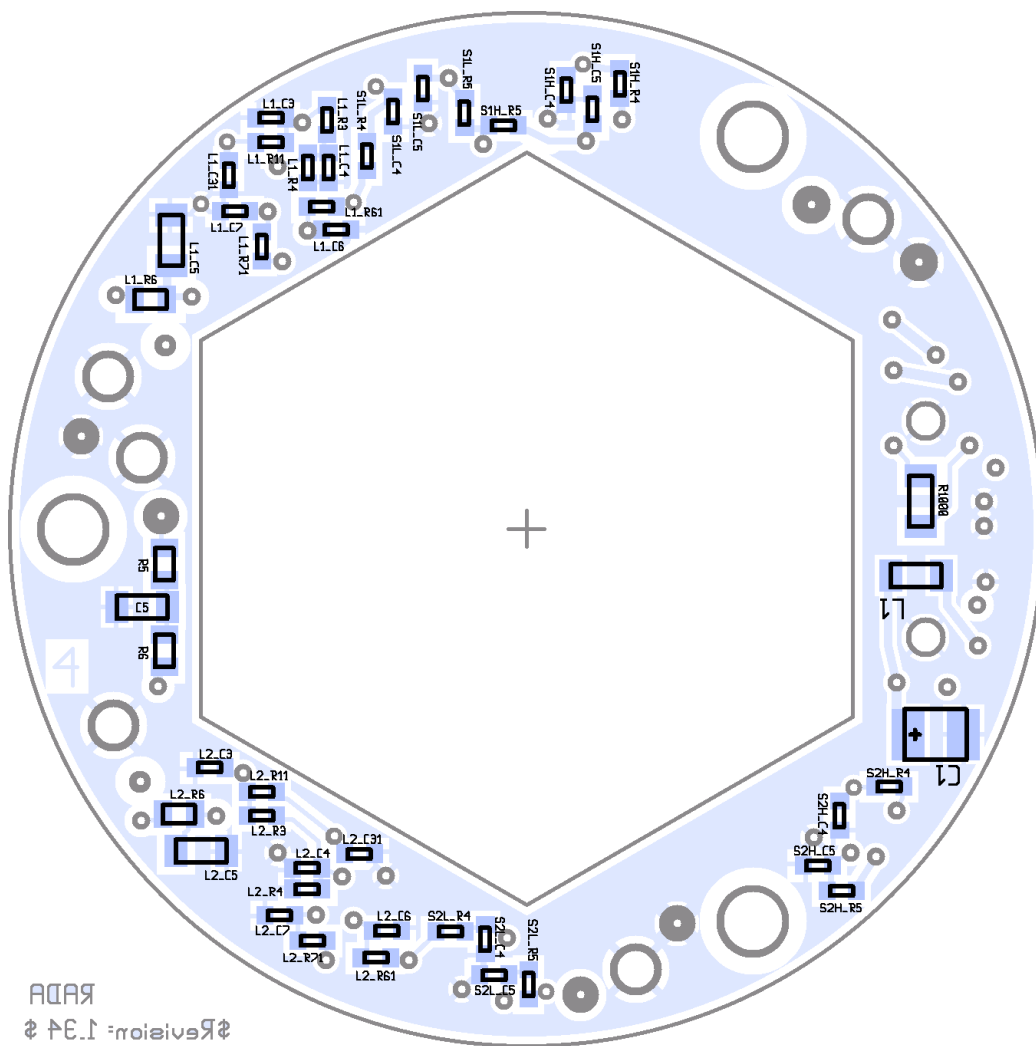


Figure 9: Back layer assembly.