

VTA Main PCB Modifications

- **Topside Cuts should be made before the board is stuffed.** See Mods 03, 06, 11, 17, 18, 24 and 25.
- **Defining direction (north, south, east, west):** When looking at the board, the PCB is oriented with the outboard TO220 devices at the top. Directions are from your perspective as you are looking at the board, whether that is the bottom or the top. So west is towards the heater regulator if you are looking at the bottom and towards the plate regulator if you are looking at the top. North is always towards the outboard TO220s.

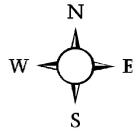
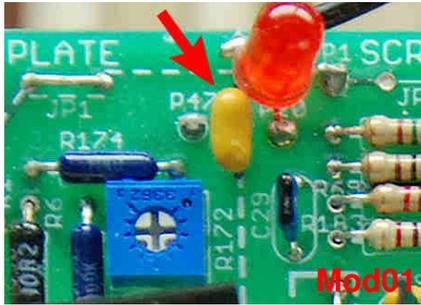


Plate Regulator



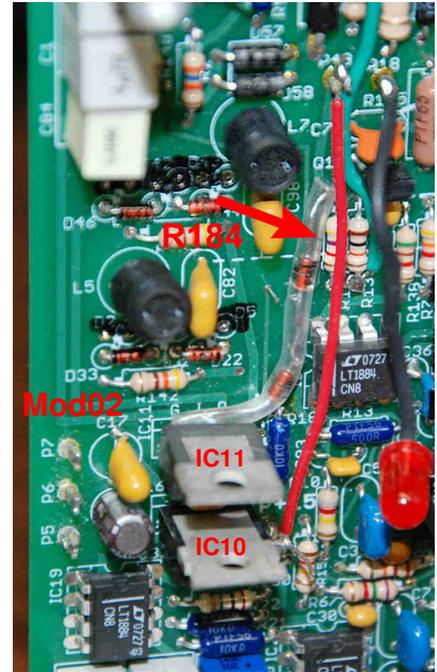
Mod01

(At left and below) [LED was for testing.] Add a 0.1uF 500V cap (CM2) from the node at terminal P1 to Gnd. Requires two holes drilled (arranged vertically) between P47 and P20. The P1 trace is nearby on the bottom. Run a jumper on the bottom from the cap to the gnd side of C74 (nearest R125).



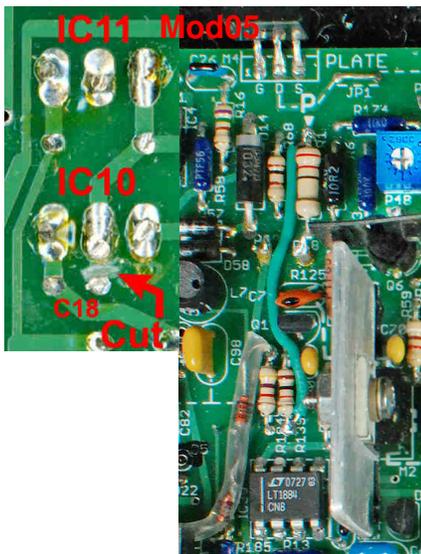
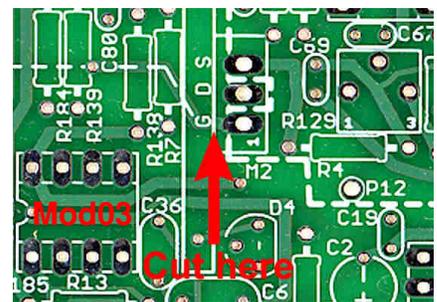
Mod02

(At right) [Red wire was for testing.] Add a series anode-to-anode pair of 6.8V 1N5235 Zeners (DM1, DM2) from the north end of R184 (Q6C) to the “G” pin of IC11 (point-P). This is done on the topside in insulating sleeving.



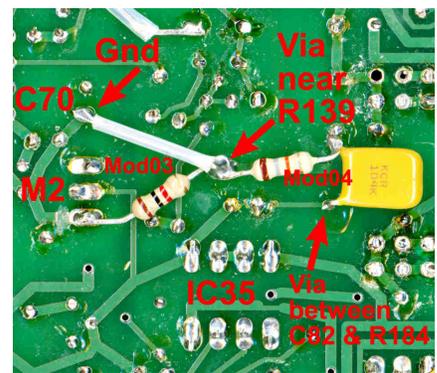
Mod03

(At right and below) Add 1K resistor RM3 between M2G and gnd. First, cut the topside trace running west from M2G. Add the 1K on the bottom between M2G and the via close to R139. Run a bottom side jumper from the via to the ground (lower) side of C70. The 18K resistor in the next mod also connects to the same via.



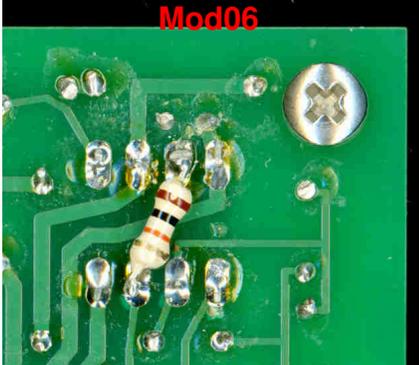
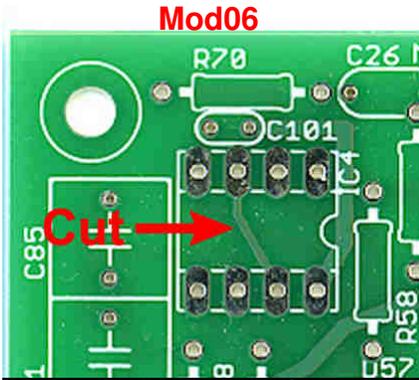
Mod04

(Lower right) On the schematic, this adds a series 18K resistor RM2 and 0.1uF 500V cap CM1 from IC5-3 to Gnd. On the bottom side, the cap lead is connected to the via between C82 and R184. RM2 goes between the other terminal of CM1 and the Gnd via used in Mod03.



Mod05

(At left) Change the routing of the “P” signal: Cut the bottom trace between IC10-2 and C18. Run a topside wire (green wire seen at left) from the north end of R1 (where the “P” label points) to the south end of R139.



Mod06

(At left and below) Insert 10K resistor RM1 between IC4-7 and IC4-3. Cut the topside trace connecting them before soldering the IC in place. Tack RM1 on the bottom as shown.

Grid Regulator

Mod07

(At right) Add 1N4148 diode DM6. Tack on bottom side of the PCB. The anode goes to the north end of R107, cathode to the north end of R8.

Mod08

(At right) Add 1K resistor RM7 between M6G and Gnd. For this, cut both bottom side traces going to M6G. Tack RM7 from M6G to R118, leaving the lead on the R118 end long. Bend that back and tack it to the right side of C83.

Mod09

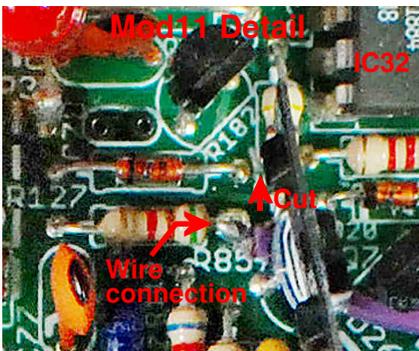
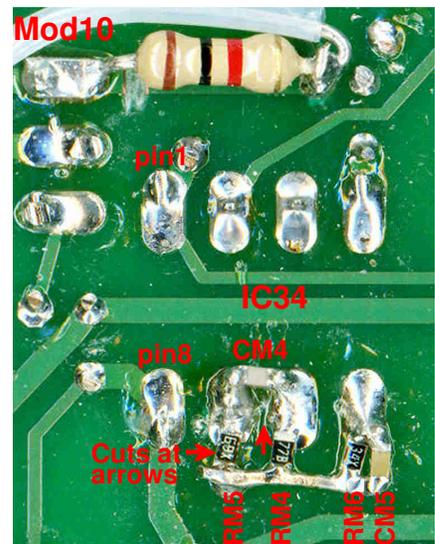
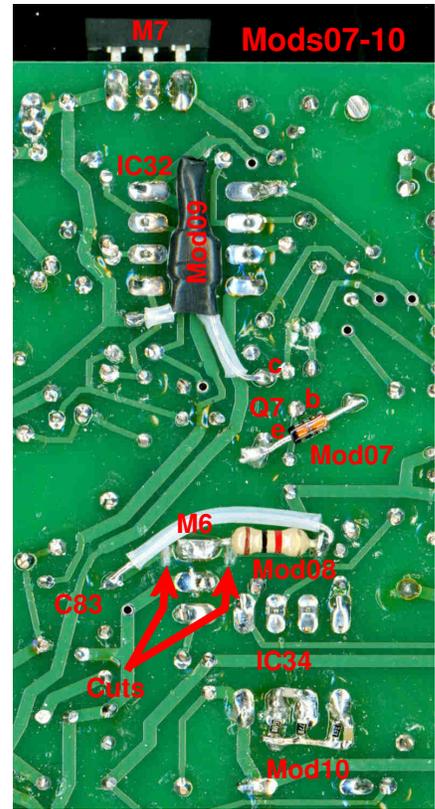
(Above right) Add a series anode-to-anode pair of 6.8V 1N5235 Zener diodes, DM3 and DM4. They can be soldered at the anodes and covered in heatshrink. Positioned under IC32, one lead goes to the west end of R135 and the other to the collector pad of Q7.

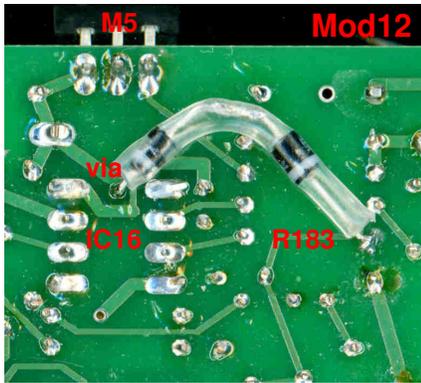
Mod10

(At right) Add the GSB Buffer Network at IC34: Use surface mount parts CM4, 180pF; CM5, 0.22uF; RM5, 51R; RM6, 22R and RM4, 6.2K. Cut bottom side traces at two places. The cuts are made to the two very short traces on the bottom going from IC34-7 to pin-6 and from IC34-7 to R132 just south of pin-7. A thin wire is soldered to that R132 pad, tying it to RM5, RM4, RM6 and CM5.

Mod11 (Do Mods23, 24 first.)

(Above left and left) (Red LED is only for testing.) R85 and DM5 (was R127) must be connected to G-15V instead of DG-15V: Cut the top trace from DM5 (was R127) to IC32-4. Run a wire (purple wire shown) on top from the east side of R85 to the south end of R181.

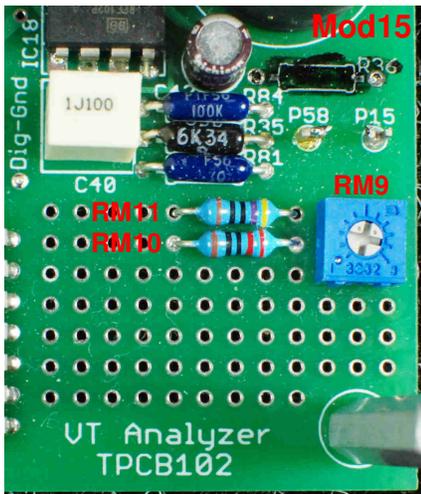




Screen Regulator

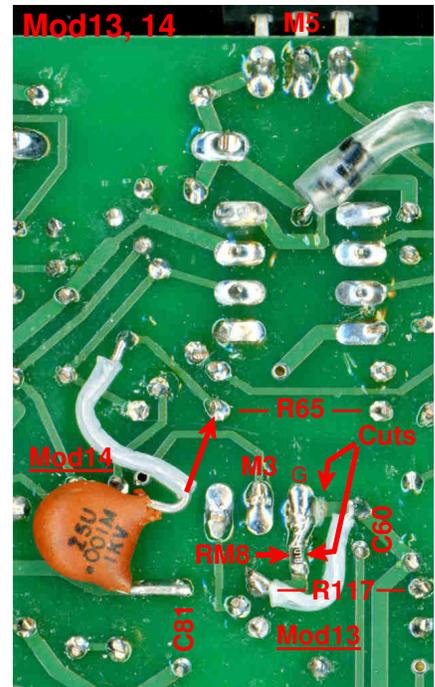
Mod12

(At left) Add on the bottom, a series anode-to-anode pair of 6.8V 1N5235 Zener diodes, DM7 and DM8. They can be soldered at the anodes and covered in heatshrink as shown. One end goes to the via under the north end of IC16. (This is signal "S".) The other end is tacked onto the east end of R183.



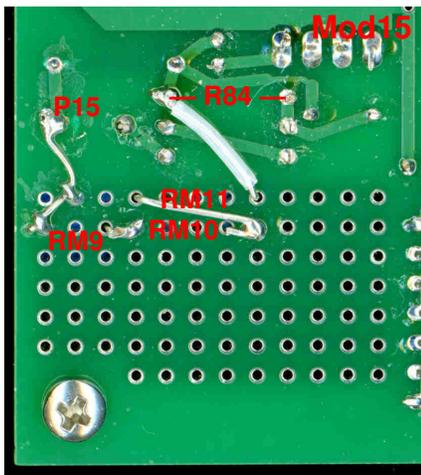
Mod13

(Above right) Add a 1K resistor RM8, between M3G and gnd. For this, cut the bottom traces going south and east from M3G and bridge the southern cut with surface mount resistor, RM8. Add a bottom jumper from the north side of C60 to west end of R117 to complete the ground path.



Mod14

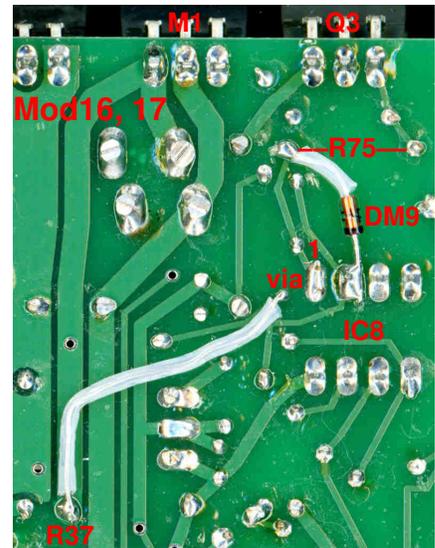
(Above right) Add 1nF 500V cap, CM3. On the bottom, tack one side of the cap to the north end of C81. Tack the other lead of the cap to the west end of R65. [The photo shows it going to a less convenient point on that trace.]



Heater Regulator

Mod15

(Left and above left) Don't stuff R36. Replace it with a series string in the breadboard area: RM9, 500R pot as rheostat; RM10, 220R 1% and RM11, 470R 1%, as shown. The wiper and CW end of the pot can go to the pad for terminal P15. The east end of RM11 is wired to the west end of R84.

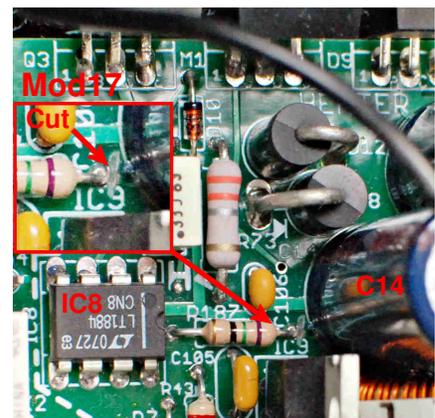


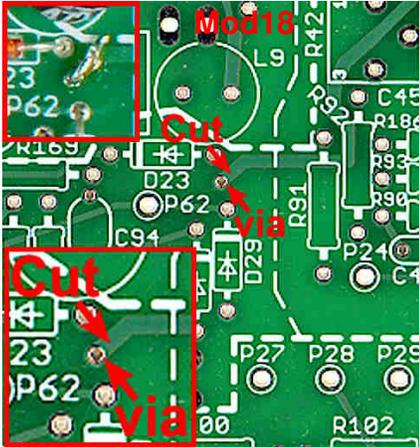
Mod16

(Above right) Add diode DM9. The cathode is tacked onto the IC8-2 pad. The anode goes to the west end of R75.

Mod17

(Right and above right) We need to reroute the trace from IC8-3 directly to R37. Cut the topside trace running east from the east end of R187. (It's just above the "9" in the label, "IC9".) On the bottom, solder an insulated wire from the via west of IC8-1 to the north end of R37.





Mod18 (optional)

(At left) Shown with enlargement in lower corner and finished jumper in upper corner. Add a jumper, so the +30V to the heater regulator can be disconnected from the rest of the board for separate testing: Cut the top trace going to the via near D29-C. Scrape off just enough solder-mask from the trace for soldering a wire. Solder a jumper from the via to the trace.

Calibration Circuit

Mod19

(At right) Add 8.2V Zener, DM10 to IC31. On the bottom, tack the cathode to IC31-1 and the anode to the east end of R159.

(Mod 20 eliminated.)

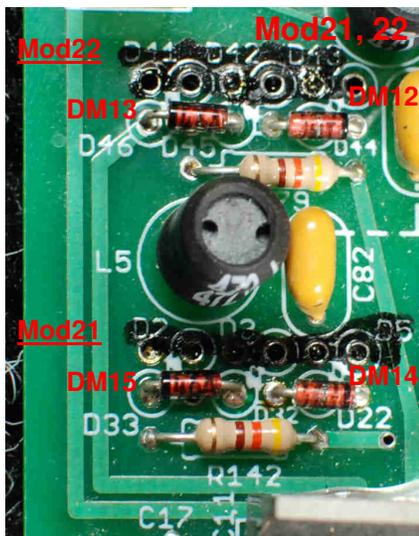
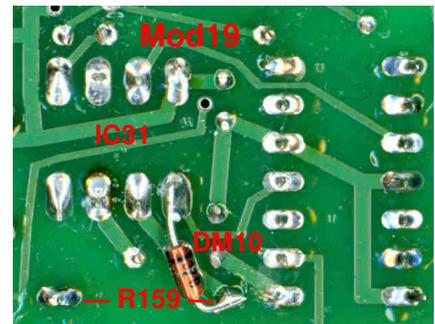


Plate Aux Power Supply

Mod21

(Above) Diodes D2,3,5,22,32,33 [the ones located between L5 and R142] are replaced by Zener pair, DM14 and DM15. Those are tied in series, anode-to-anode. They are soldered in the southern row of diode holes, with their anode leads connected under the board in the middle. The cathodes are in the holes marked D33 and D22.

Mod22

(Above) Diodes D41, 42, 43, 44, 45, 46 [the ones located between L5 and C84] are replaced by Zener pair, DM12 and DM13. Those are tied in series, anode-to-anode. They are soldered in the southern row of diode holes, with their anode leads connected under the board in the middle. The cathodes are in the holes marked D46 and D44.

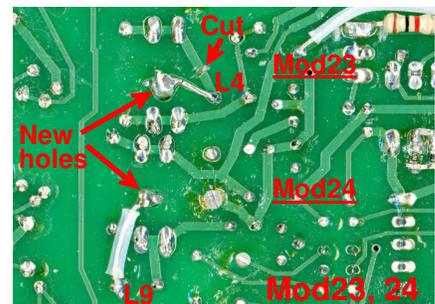
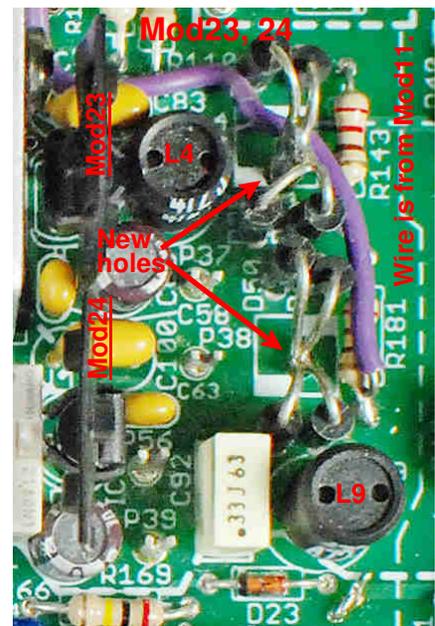
Grid Aux Power Supply

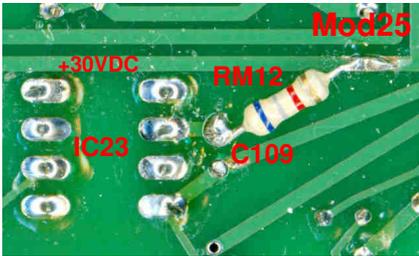
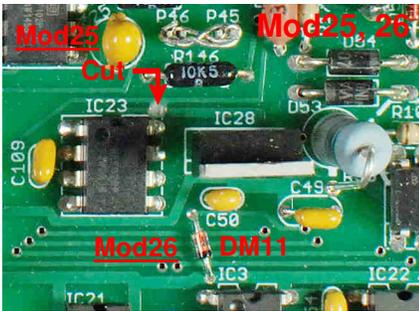
Mod23

(At right and below) On the bottom, cut the trace from D35 anode to L4. Instead of installing D34, 35, drill a 3/32" hole in the middle of their area. Solder four 1N4007 diodes vertically in the four diode pads. Make cathodes and anodes match the existing silkscreen. The four other leads bend and go through the hole. On the bottom, solder the four leads and run one to L4.

Mod24

(At right and below) On the top, cut the trace from D49 cathode to L9. Instead of installing D49, 50, drill a 3/32" hole in the middle of their area. Solder four 1N4007 diodes vertically in the four diode pads. Make cathodes and anodes match the existing silkscreen. The four other leads bend and go through the hole. On the bottom, solder the four leads and run one to L9.





Generator

Mod25

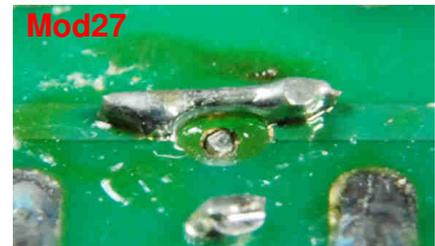
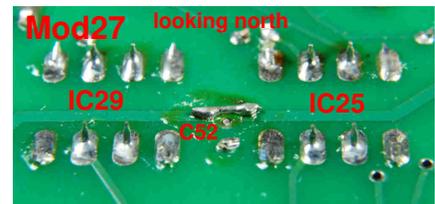
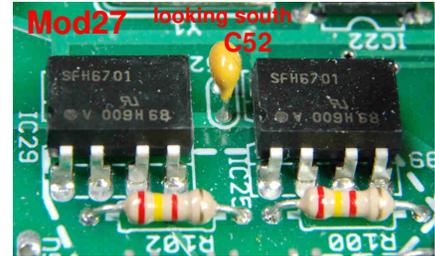
(At left and below) Powering the reference separately: On the topside, cut the trace from IC28-1 to IC23-2. On the bottom scrape some solder mask from the +30VDC trace running just north of IC23. Tack 6.8K resistor RM12 from the north end of C109 to the +30VDC trace.

Mod26

(Above left) Add Schottky diode DM11 topside with the cathode in the via just north of IC3-1 and the anode in the via just west of C50.

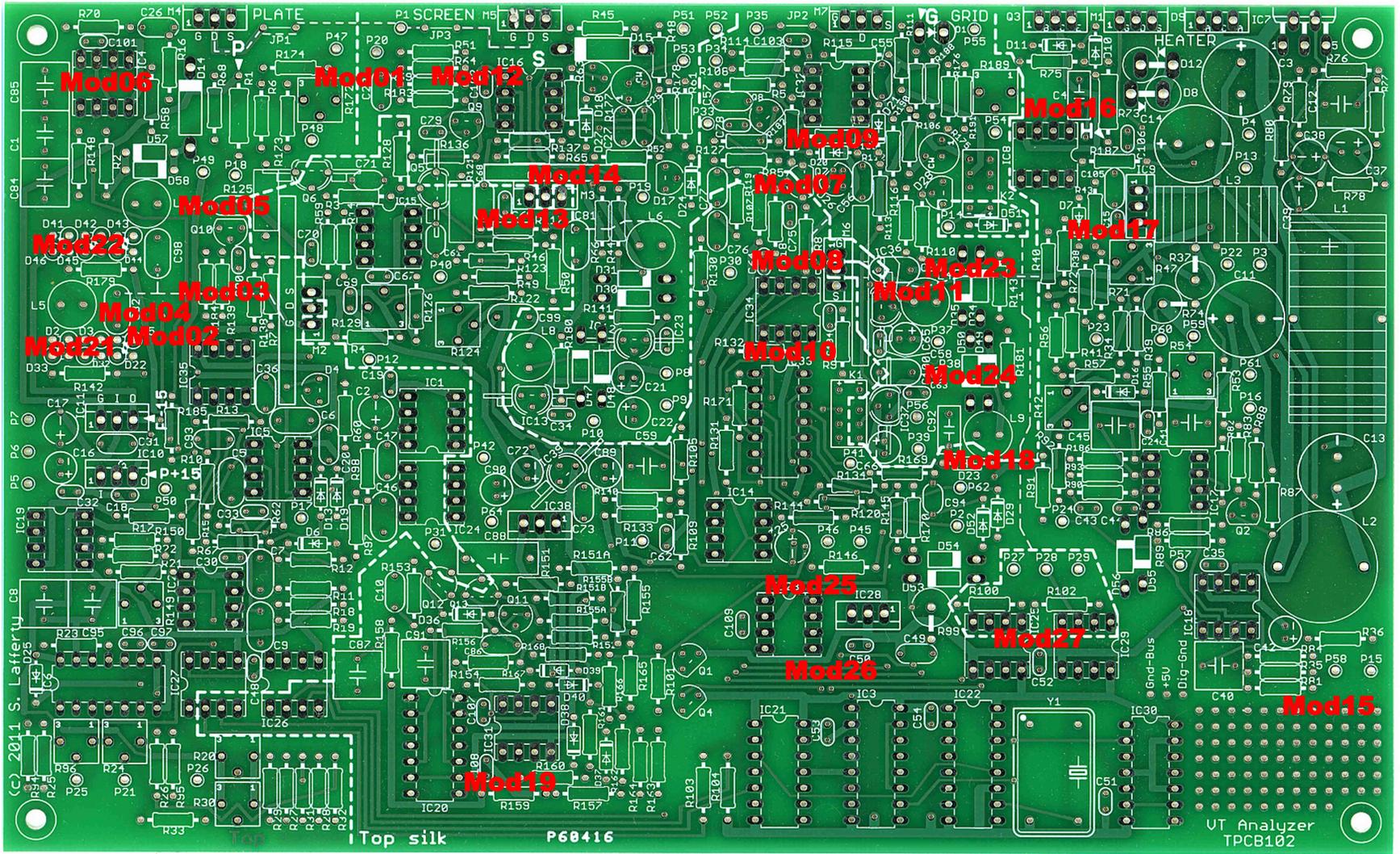
Mod27

(At right) The ground terminal of C52 erroneously ties the Dig Gnd running on top to the Gnd Bus running on the bottom. To separate these, the gnd pad of C52 must be carefully cut away without disturbing the topside pad. The gnd lead of C52 is cut short so that it solders topside but does not extend to the bottom on the PCB. Then a wire bridge is soldered to repair the interrupted Gnd Bus trace on the bottom. (Dig Gnd and Gnd Bus come together through the heater regulator ground plane.)



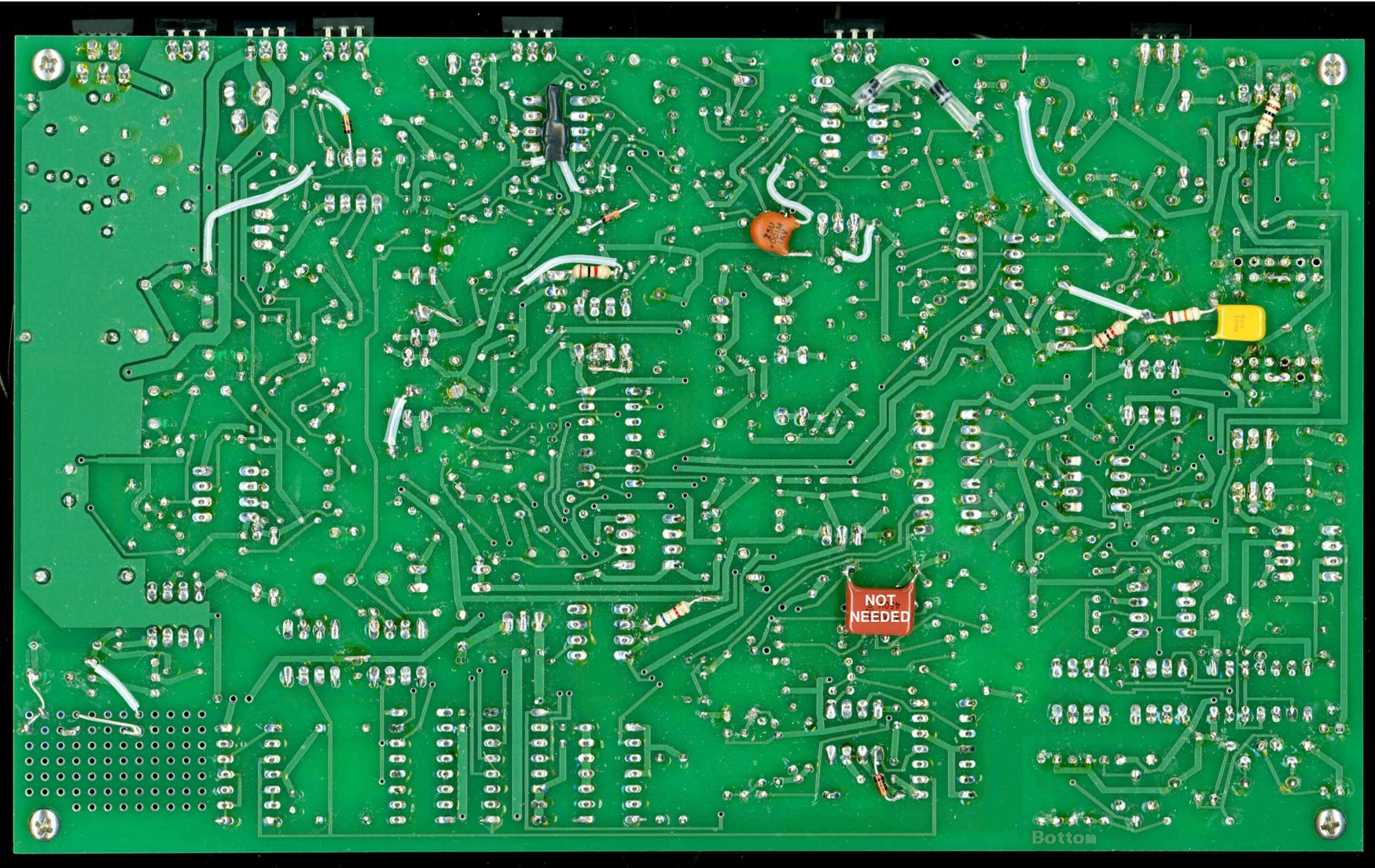
Reference Documents Follow:

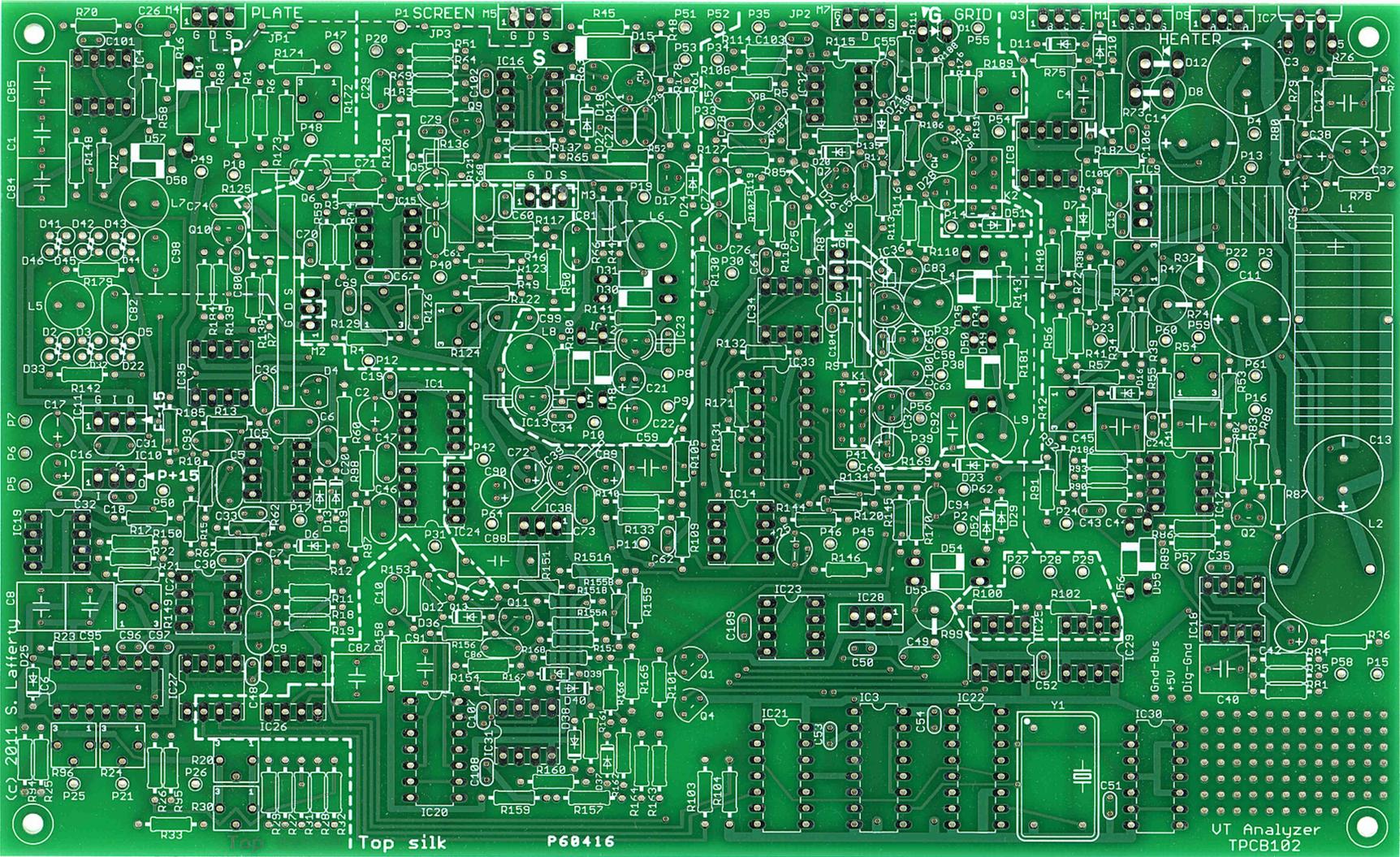
- Locations of Mods
- Bottom of Finished Main PCB
- Top of Finished Main PCB
- Top of Bare Main PCB
- Main PCB Schematic Page-1
- Main PCB Schematic Page-2

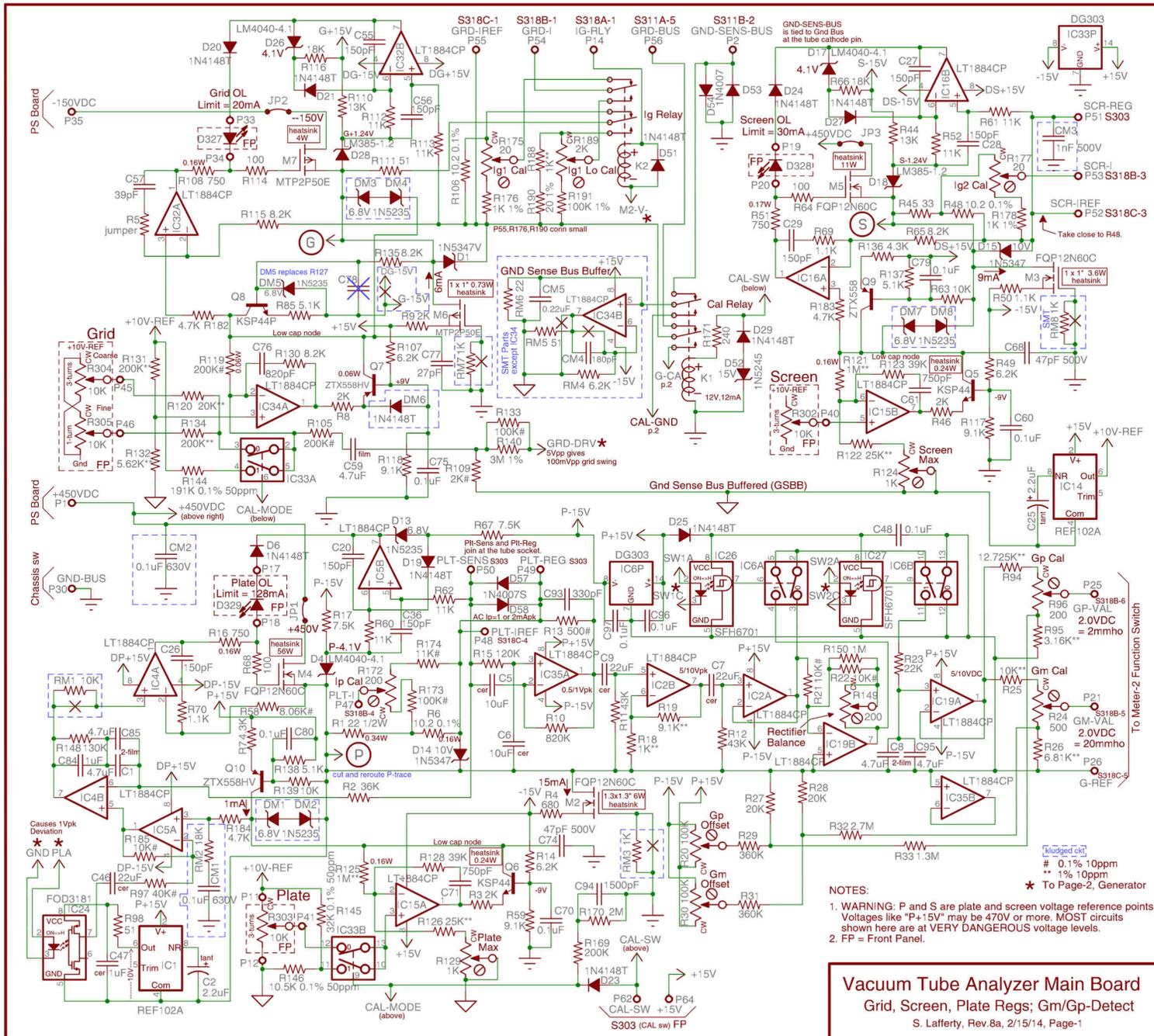


Locations of Mods

Bottom of Finished Main PCB







NOTES:
 1. WARNING: P and S are plate and screen voltage reference points. Voltages like "P+15V" may be 470V or more. MOST circuits shown here are at VERY DANGEROUS voltage levels.
 2. FP = Front Panel.

Vacuum Tube Analyzer Main Board
 Grid, Screen, Plate Regs; Gm/Gp-Detect
 S. Lafferty, Rev.8a, 2/15/14, Page-1

