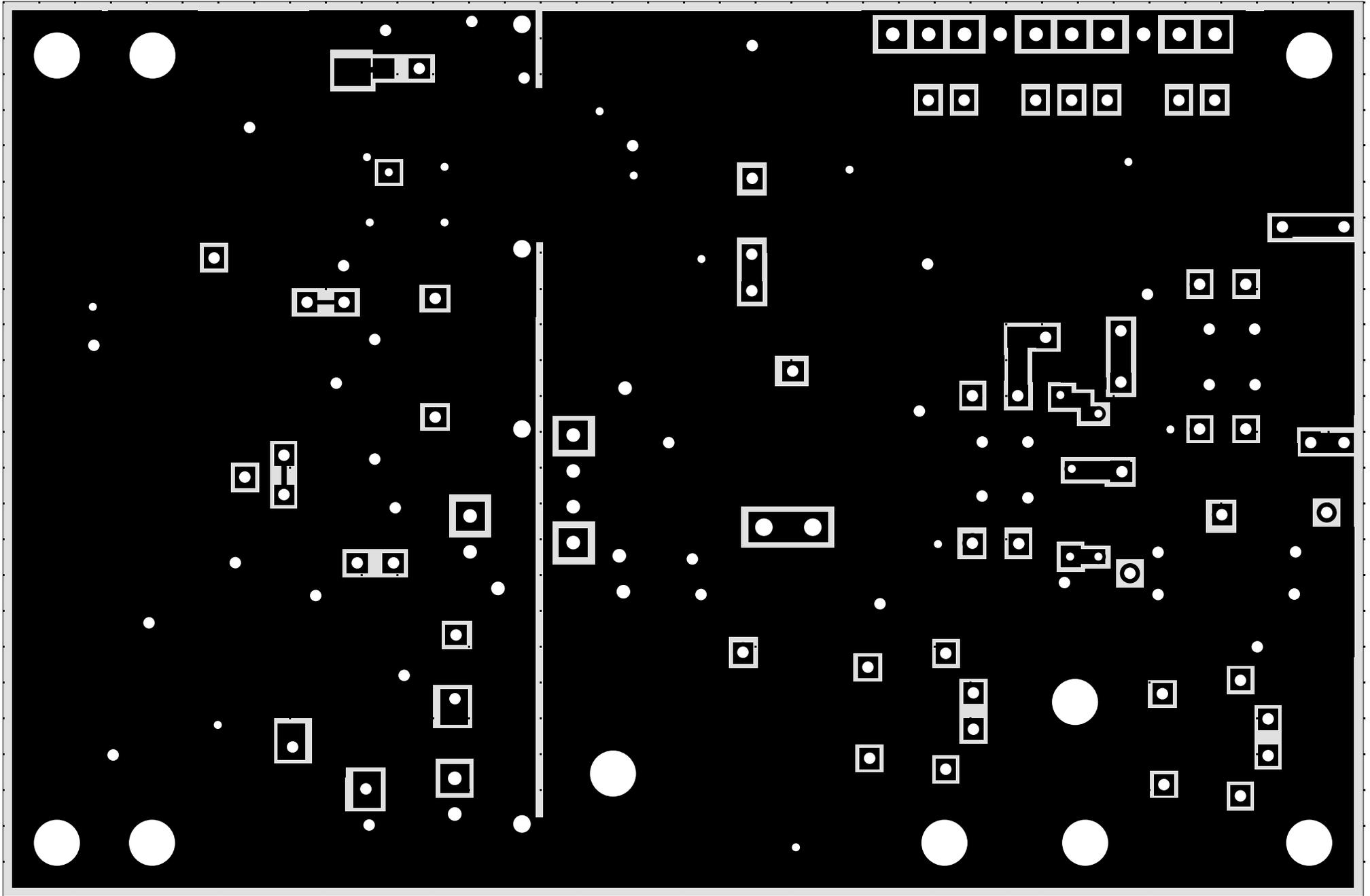
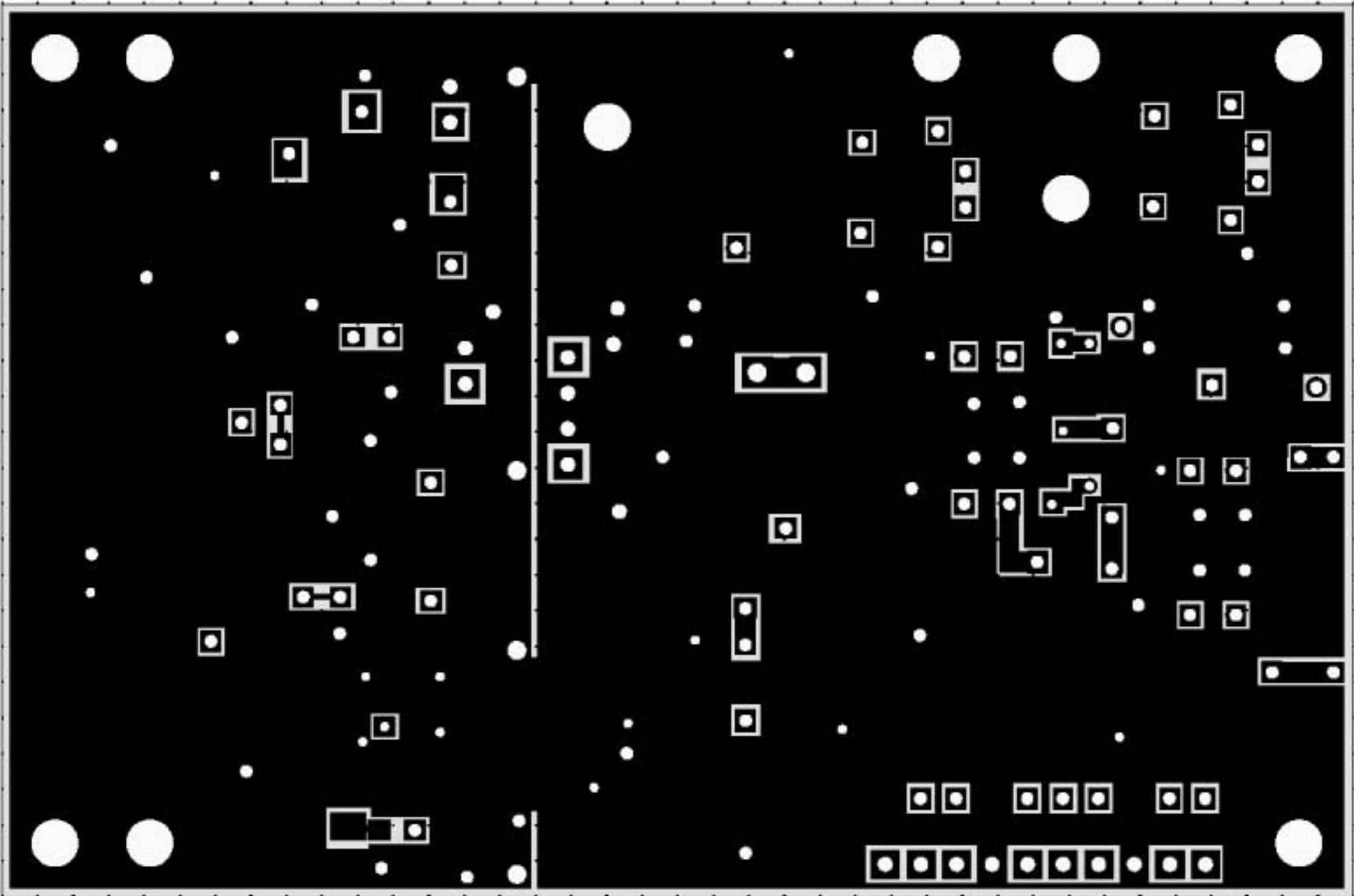
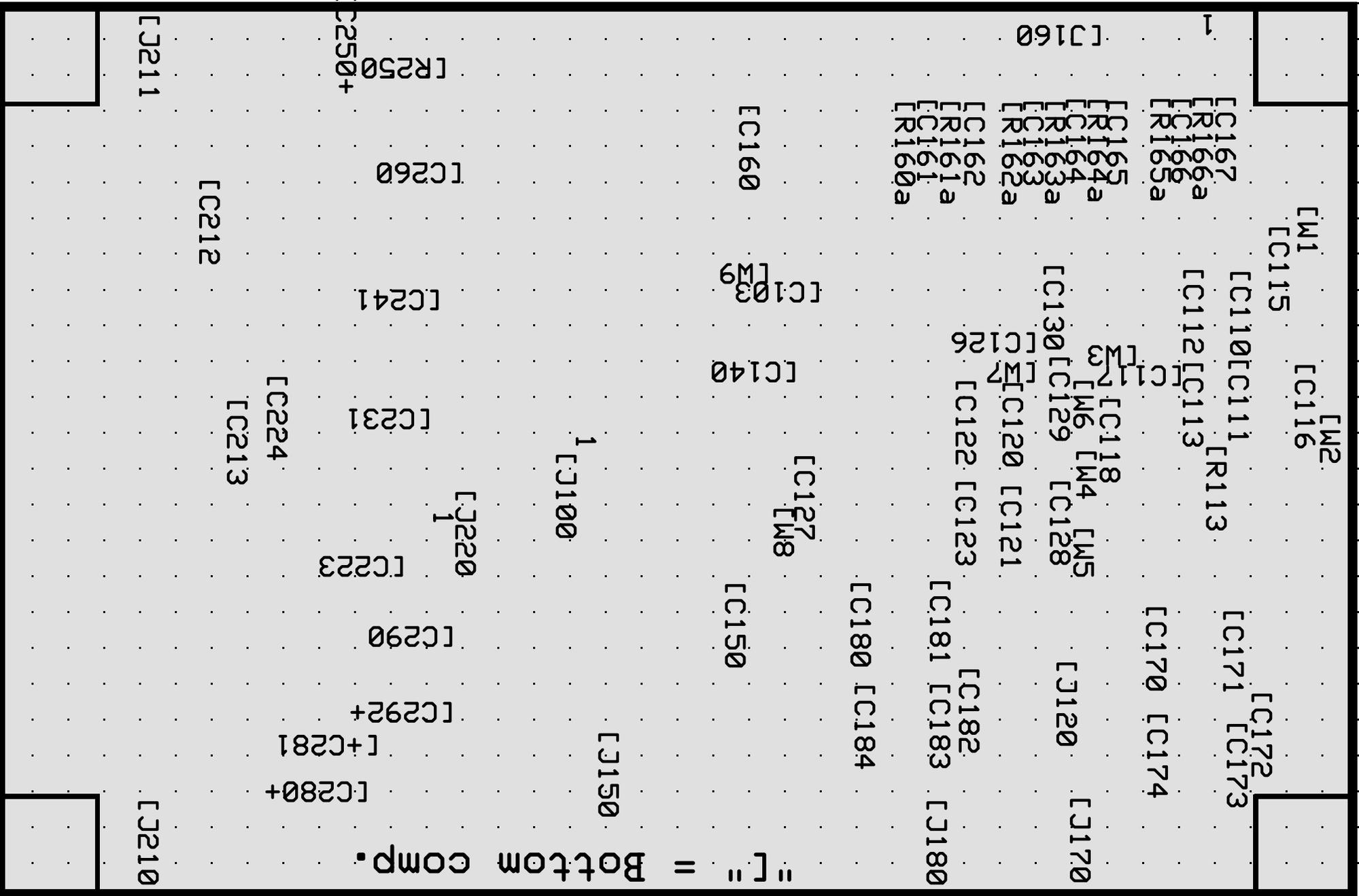


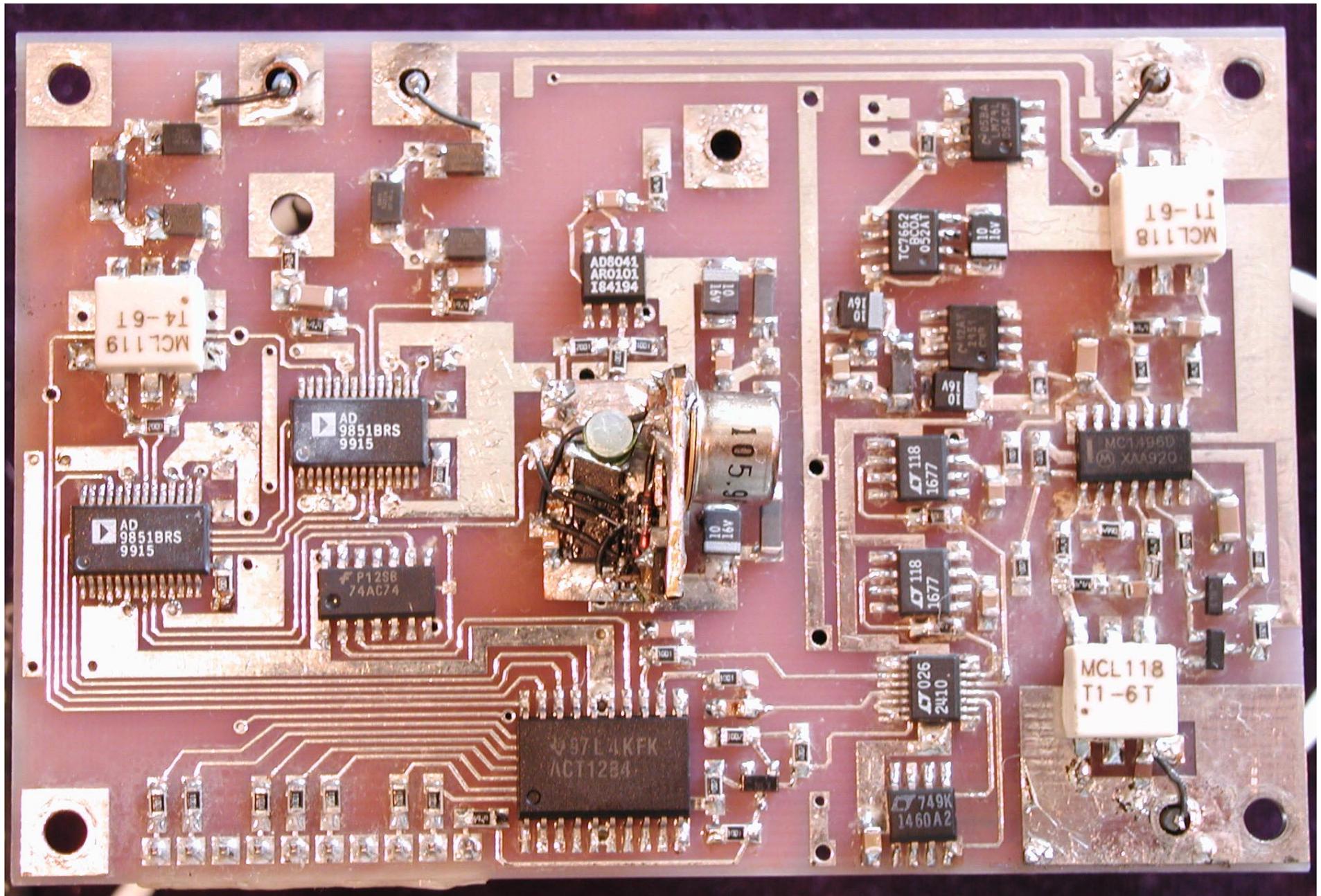
VNA PCB - Top Layer





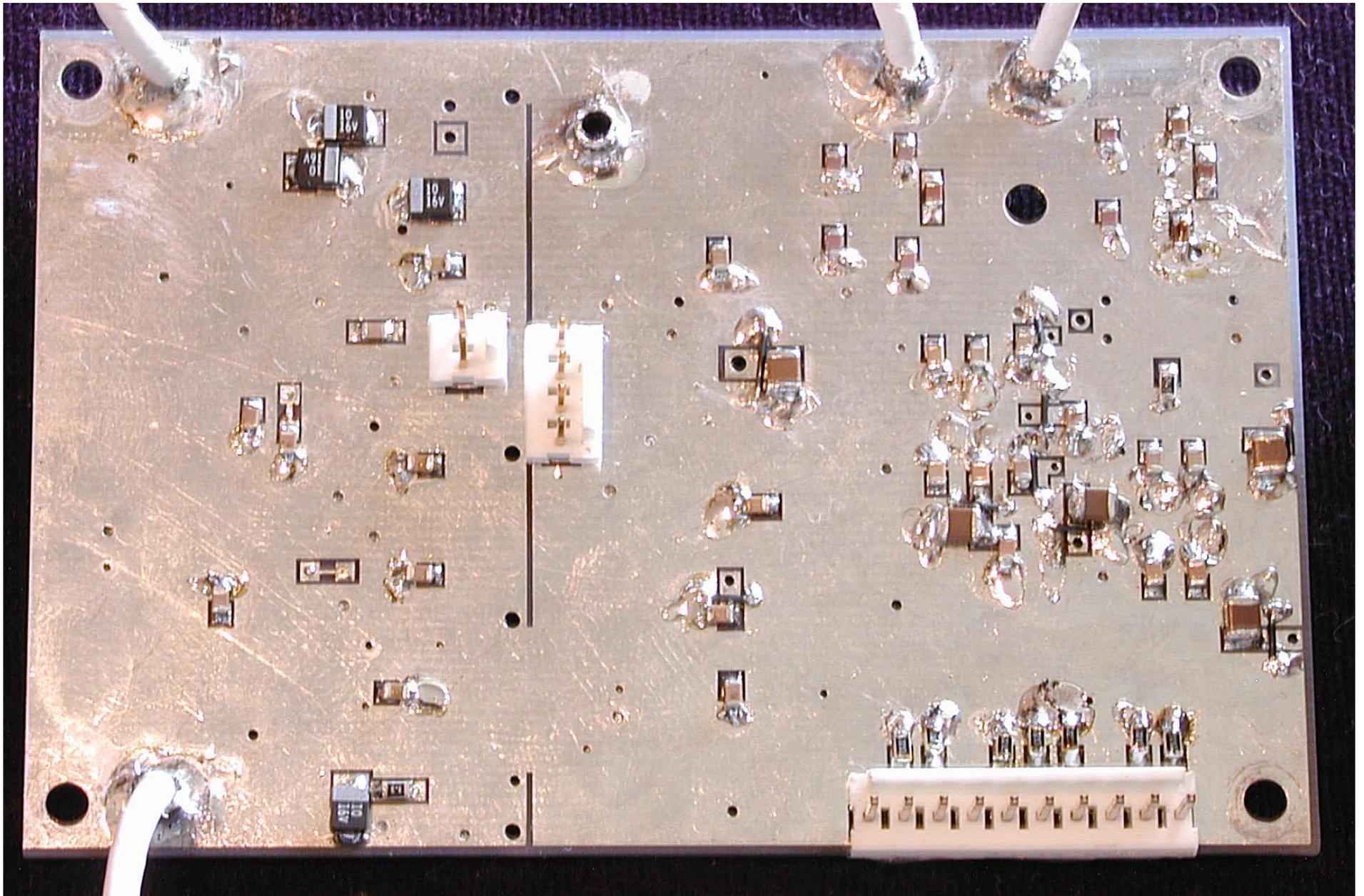


VNA PCB - Virtual Silkscreen for Bottom Components (as viewed from top)  
 Staple this page to the previous page with the printed sides facing one another  
 and board perimeters aligned to aid in populating the bottom components.



Populated VNA PCB - Top

Note that the photos are from an early build. So there are some differences, particularly on the homebrew master oscillator. Resolve differences using schematics & drawings. Also see G3SEK's annotated photos at the website.



Populated VNA PCB - Bottom

## G3SEK VNA Build Notes

-----  
(minor edits by N2PK)

Print each annotated image in color at full-page size, and cross off each component as they go. The annotated images are available from the N2PK VNA website.

The build sequence that worked for me was as follows.

1. Install all rear-side components (not in schematic order, just working systematically across the board). Note that R161 is stacked on C161, and the same for R162/C162...R167/C167.
2. Install 9 rear-side wire links.
3. Very thoroughly wash the rear of the board with flux removing solvent, scrubbing with an old toothbrush. Rinse with solvent until the board is completely clean and not sticky. Then use a magnifier or microscope to check for missed connections and solder bridges.
4. Install all top-side passive components (again, working systematically across the board). Do not install transformers at this time. R213 is stacked on C217. Install wire links to replace C210 and C211, and connect L182 through to T210 via two links. (N2PK - This last step only applies if the internal LO path is used.)
5. Repeat the cleanup and inspection procedure (step 3) for the top side.
6. Take a break, and build the +5V regulated and +12V semi-regulated DC power supplies.
7. Make power connections to the board via J100 and J220. Check that +5V and +12V appear on the board at appropriate places.
8. Install U290. Check for -12V DC at R280.
9. Install U280. Check for -5V where U210 pin 14 will be (do not install U210 yet).
10. Install U220. Check for +5V where U210 pin 6 will be (do not install U210 yet).
11. Install U250. Check for +2.5V where U260 pin 3 will be (do not install U260 yet).
12. Install U230, U240, D270, Q270, T210 and T211.
13. Install U210. Check for +3.06V at R232 and also at R242 (these voltages should be very similar, but will change later when the DDSs are operational).
14. Install U160, U130, U150, and Q190.
15. Repeat the top-side cleanup and inspection (step 5).
16. Install U260 - your first practice at SSOP soldering. Repeat the cleanup procedure around U260 - don't let dirty solvent run back into clean areas of the board. Now inspect *very* closely! Working under the magnifier, run a very small quantity of solder on to each pin wherever necessary. When you are happy with the technique...
17. Install U110 and U120.
18. Once again, repeat the cleanup around U110 and U120. Again, don't let dirty solvent run back into clean areas of the board. And again, inspect *very* closely indeed. Working under the magnifier, re-solder pins wherever necessary.

19. Install T110 and J160.

20. Install the MO (see separate build info). The MO should already have been checked out on the master oscillator test board.

21. Power-up. You should see 148MHz at J150, and also routed to pin 9 of each DDS. There should no output from either DDS (they don't have any instructions yet) and the chips should be only very slightly warm. (N2PK - Probing the master oscillator (U140) output is best done at or near its output pin. A 10X 'high impedance' probe, if used, should have a maximum of 3 pF capacitive loading. A resistive divider probe, 2 Kohm or greater, to a scope 50 ohm vertical channel is recommended in place of a conventional 'high impedance' probe for best displayed waveform fidelity. In all cases, an extremely short probe ground lead is required - 1 inch or less.)

22. The board is now complete, and ready for checkout with the PC.