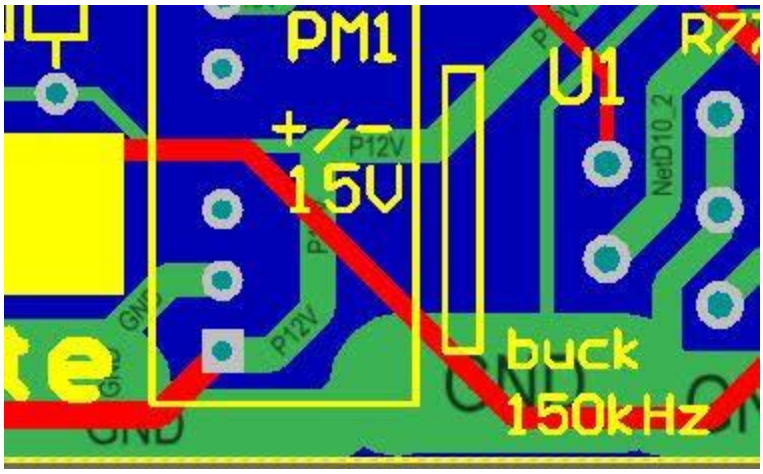
Issues described by Winfield Hill, email of 3 June 2019:

PM1 connections
With board shipments coming up, this may be a good time to remind (or mention) everyone about my pin-1 & pin-2 PCB layout error for PM1, where pin 1 was supposed to go to ground. So you need to swap these two pins. You'll want to make the painful cut-jump fix on the top and bottom, before starting to mount parts, so you can get a clear access with your X-Acto knife. Here's the faulty as-delivered PCB layout image.



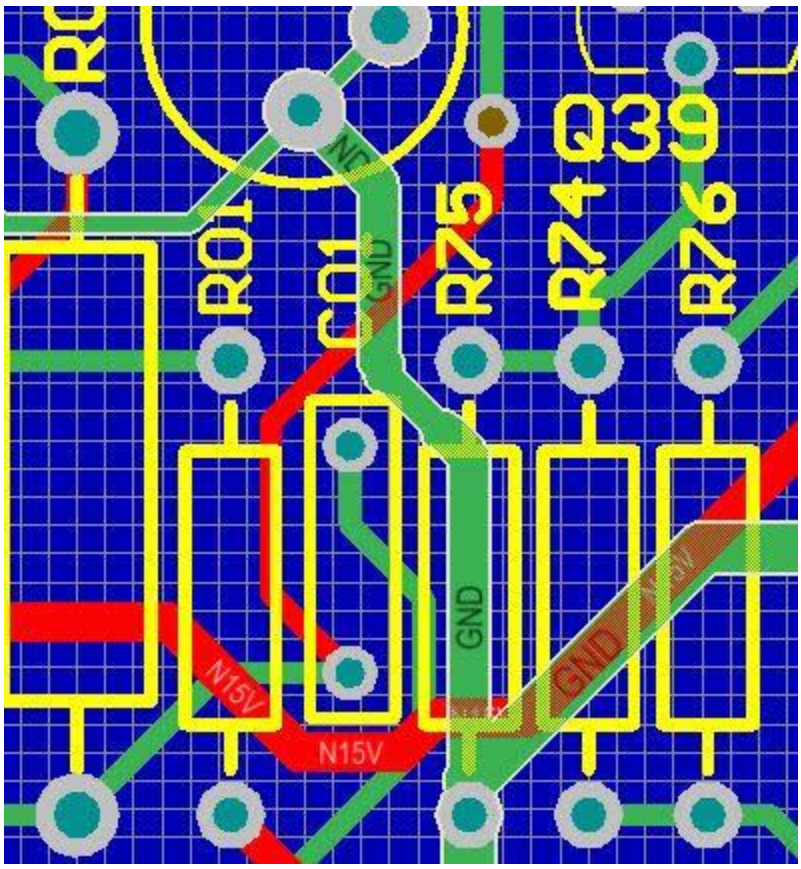
JD on 6Jun2019: I have modified the connections to the +/-15V regulator as detailed above, and can confirm it works correct. The buck converter supplies 12V and PM1 outputs +/-15V. I cut all traces to pin 1 and pin 2 of PM1, then rerouted the connections. For the top connection I scratched some mask off the track and soldered directly to it. Mod pictures below; it is not beautiful, especially close-up, but functional ;-)

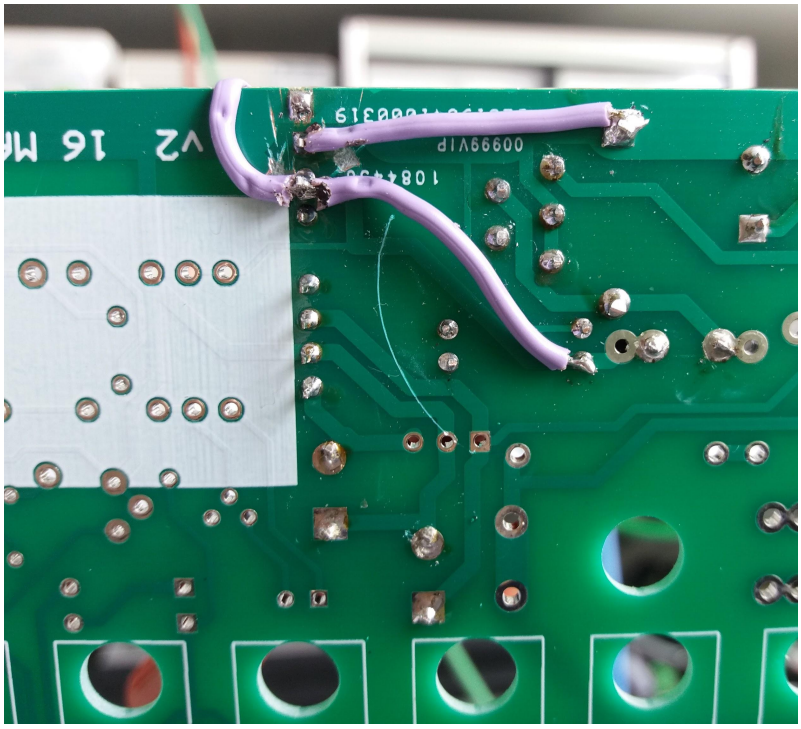
C01 connections
And this is a good time to consider making the change to C01 wiring recommended by the German experts, returning the output side to a Vas node, such as the emitter bus of Q25. This change lowers the impedance of the Q19 Q25 emitter-follower drivers to the output transistors, at high frequencies where C01 becomes effective. That's helpful, considering the declining beta of the output transistors above 1 to 2 MHz.

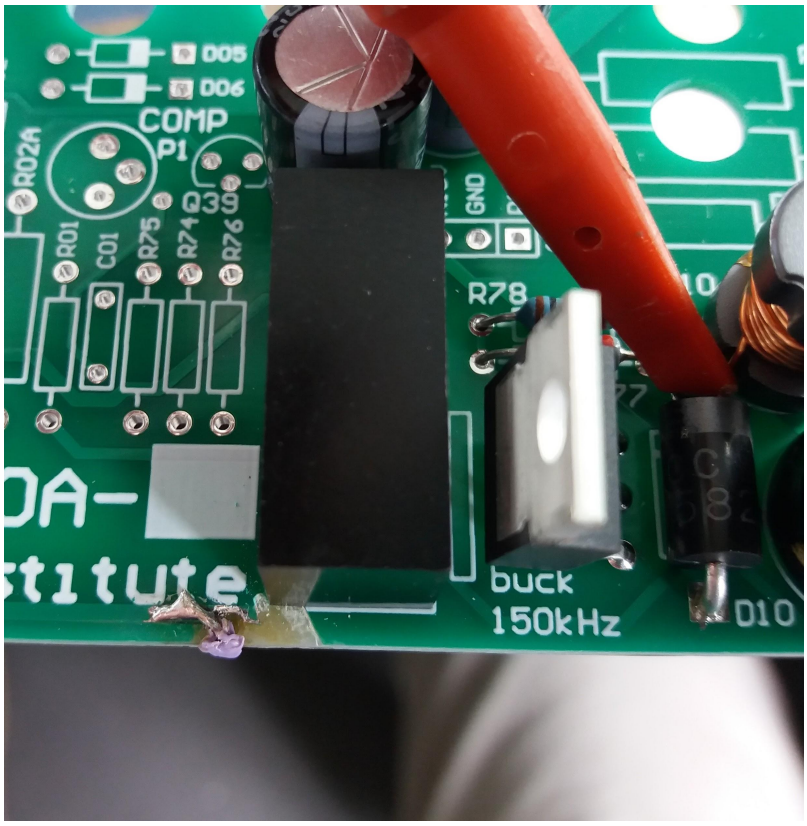
C01 is an empirically-determined part. According to my notes, I used 2pF. That value becomes effective at 15 MHz, suitable to cancel a troublesome

pole, given the 7 to 10 MHz bandwidth I was seeking. I don't recall experimenting much with that value, larger may be better. Also, if you aim for a little lower f-3dB, you'll want a larger C01 value.

Here's the layout for C01. You can see there's a via between the trimpot and Q39. You can cut the continuing trace under, and add a wire from the via, over to Q25. But, it'd be convenient to do this before mounting the surrounding parts. If you want to experiment, you can always move the far end of the wire later (that end of the cap is at low-Z, without sensitive wiring issues). And C01 is well exposed, so you can play with its value. Tacking an extra cap to the pins underneath may be a good way to experiment, or you may want to stand up the part a bit so you can snip wires and solder parts on top. Yessir, I'm sure y'all have experience doing things like that.





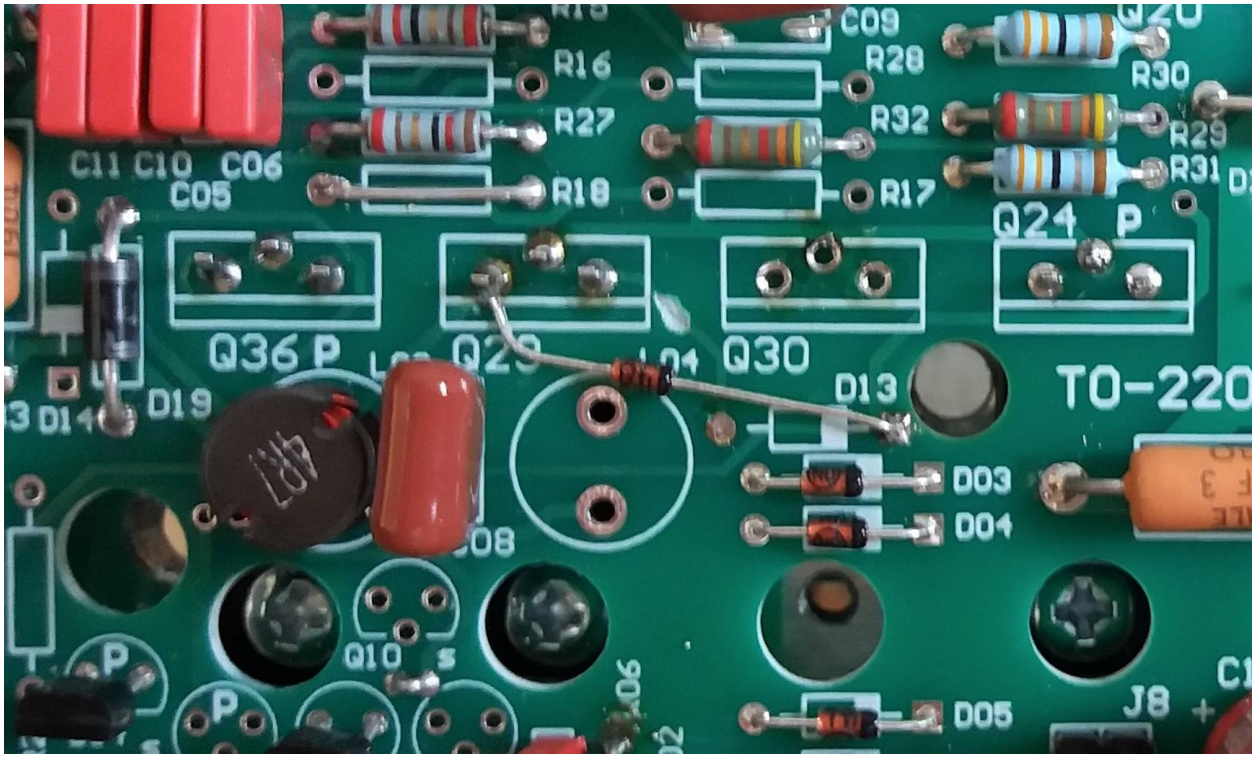

D05
D06
COMP
P1
R02A
R01
C01
R75
R71
R76
Q3
GND
R78
buck
150kHz
D10

Issues found with 1st assembly JD

1. Space between PM3, PM3 and electrolytics C26, C27 is tight. Use smallest diameter electrolytic available.

2. There is a difference in connection between clamp diodes D12 and D13. D12 runs from Vc to **E**Q15, while D13 runs from **B**Q29 to Ve. It appears that the D12 connection is correct. The D13 connection to **B**Q29 should be changed to **E**Q29.

3. Did the modification: cut the trace on the top between BQ29 and Anode-D13. Then mount new D13 between original pad for Cathode-D13 and EQ29. See picture below. Functional test OK.

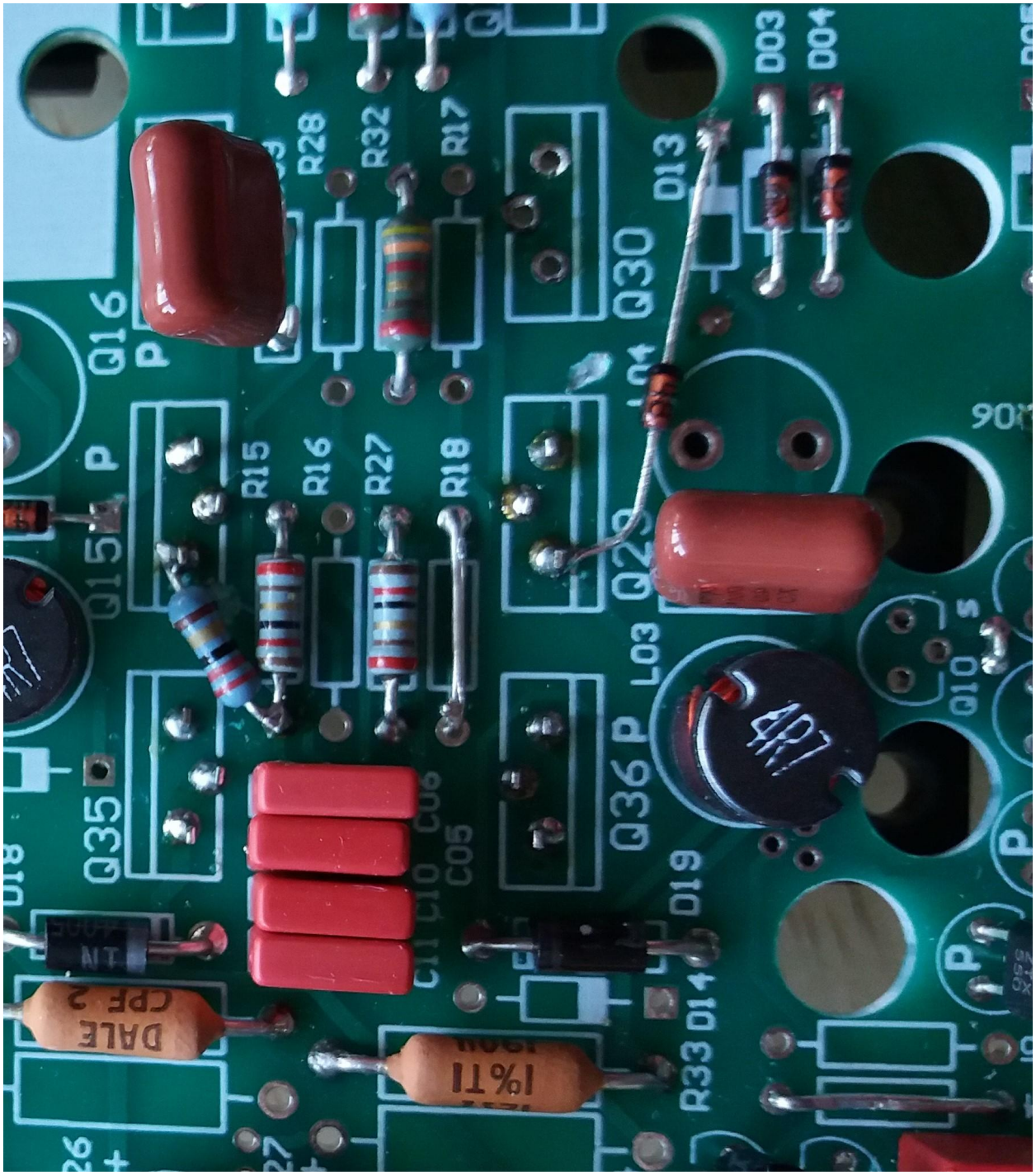


Issues found with functional testing

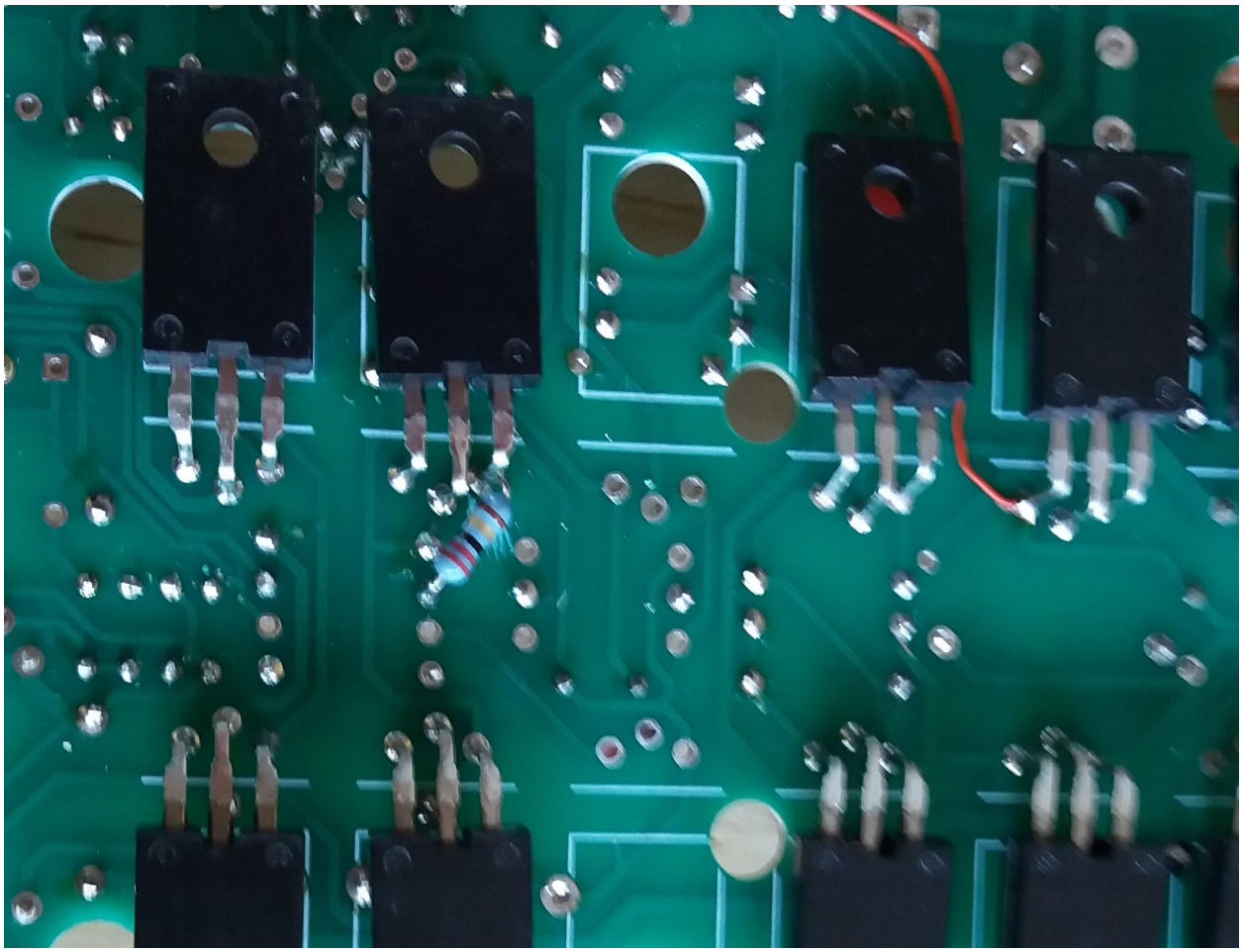
4. With the functional test, an oscillation was present, which consisted of two different signals. A high frequency oscillation could be stopped with adjusting the compensation trimmer P1. C01 was not necessary.

5. The 2nd oscillation which looked more like ‘squegging’ was not influenced by P1. Reviewing the construction and test report from the German Tigris group revealed that they had similar issues. This was solved by inserting a 22Ω base stopper resistor directly at the base of Q15 and Q29. This completely cured all oscillations. See pictures below. The reasoning was that this stopped HF feedback through the supply lines through C05 and C10.

6. Another issue was the relatively high broadband output noise (almost 1mV), which was identified by the Tigris group as emanating from the two floating references U7 and U8 (TL431). The noise was halved by placing an additional RC filter between U7, U8 and Q15, Q29 of 100Ω and 120uF. I met them halfway by replacing R15 and R27 with 330Ω resistors. Due to limited board space, C05 and C10 were left in place as-is. At this time, noise measurements have not yet been done.



Q15 base stopper



Q29 base stopper