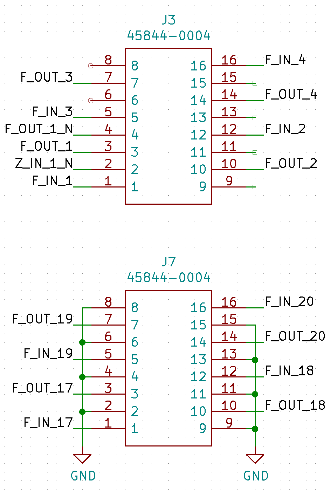
KiCAD Filter model updates March 21st, 2022

Job Summary:

I’m hoping that Kristian Jordan can make a few updates to the design of filterboards and motherboards. The main changes desired relate to keeping signals differentially paired, and not having the negative pairs grounded on the board.

Job Details:

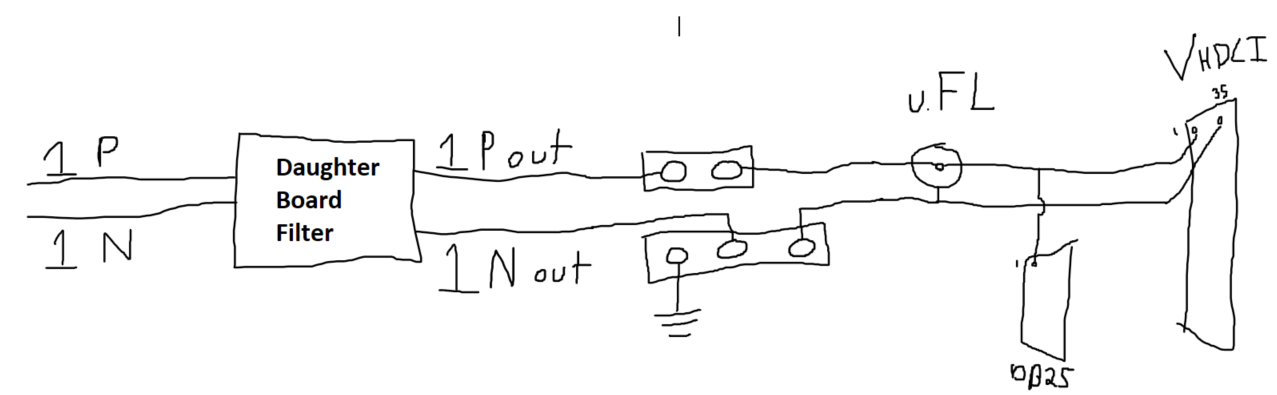
1. The output connector, which was previously changed to a 68 pin VHDCI just like the input, also needs to have negative pairs passed through as an option. The pin numbers need to match the input VHDCI, even though this will necessitate a large number of overlapping traces and vias. Previously I thought it would be acceptable to switch them, but I was incorrect.
2. **The daughter boards must be modified so that each channel’s paired negative signal is used as ground, and passed through to output. I’ve already made this change in the schematic for channel 1:**

See Z\_IN\_1\_N goes to pin 2 instead of GND ->

Not like this ->

1. The daughter boards still have two inner ground planes, these need to ground through a 10M resistor to one of the negative input signals.
2. The motherboards need a number of connectors added:
   1. Between the filter boards and the output, the positive signals should go through a jumper **(0.1” header 2x32)** so that they can be interrupted if desired. **Please arrange these all in a row so that a 2x32 header array can be used. Feel free to increase board size.**
   2. Similarly, the negative signals should go to the center of a 3 pin header, so that they can either be grounded or passed along to the output VHDCI. This means a **large 3x32 header including all 32. Please arrange all in a row, increase board size in vertical direction so 68 pin connectors are farther from each other.**
   3. After the jumpers, before the output VHDCI, the signals should go to a u.FL pickoff, positive on the center pin, negative on the outer shield. **Leave a little more space for these, the wires come out sideways, and make sure they are in order. 4x8 is okay, but must go 1,2,3,4 in one row not 2,1,4,3.**
   4. Positive **Output** Pins 1-25 **should also go to a DB25 connector**, as they used to in the older version of the motherboard. **This connector should be added after all the jumpers, close to the output VHDCI68 connector.**
3. The motherboards also need all RC filters removed.
4. All gerbers, and assy files (BOM, centroid, paste, assy drawing) need to be re-generated.

Motherboard Schematic for the 1st Channel:

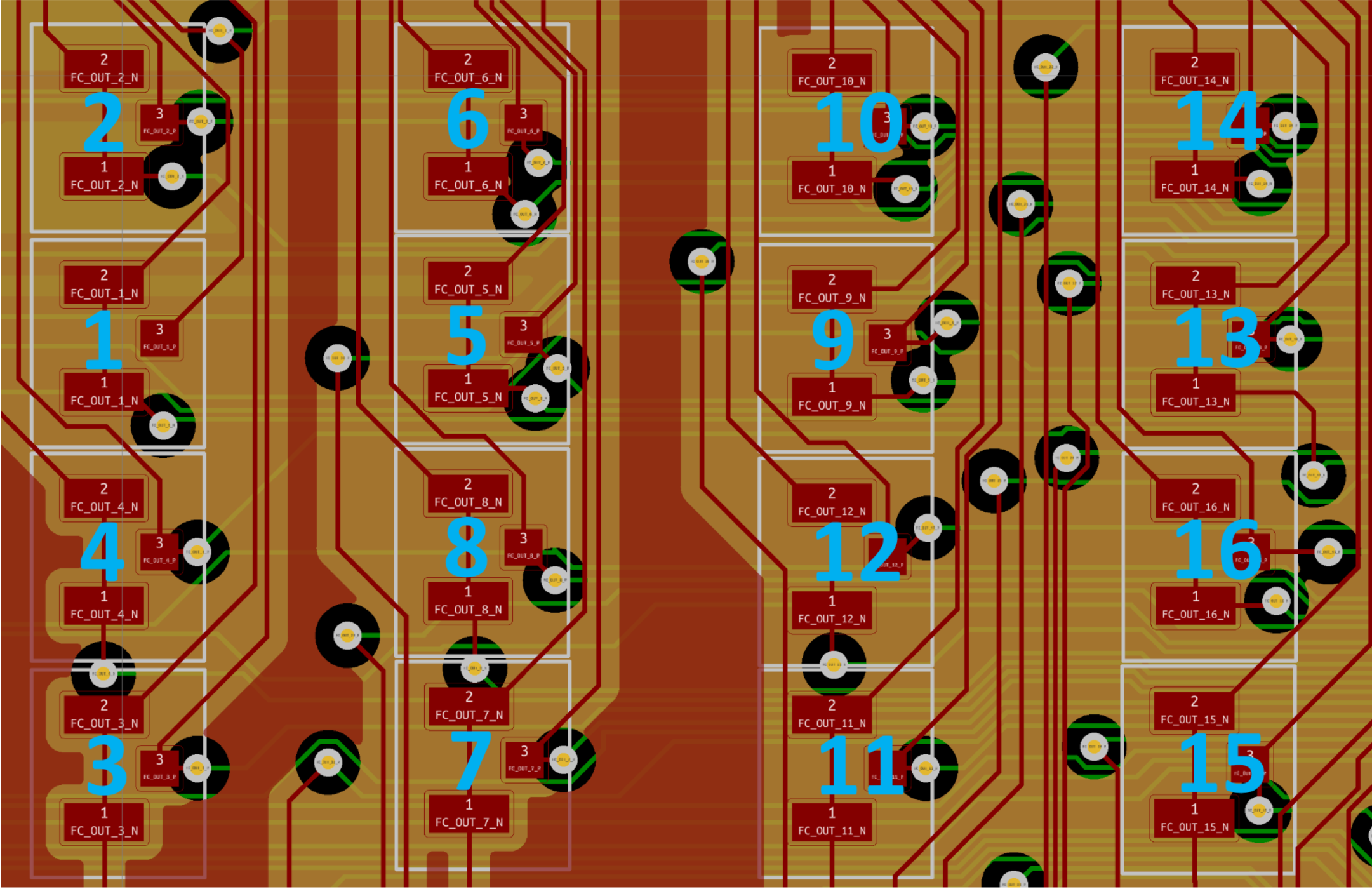


Part List: (still needs some updating!)

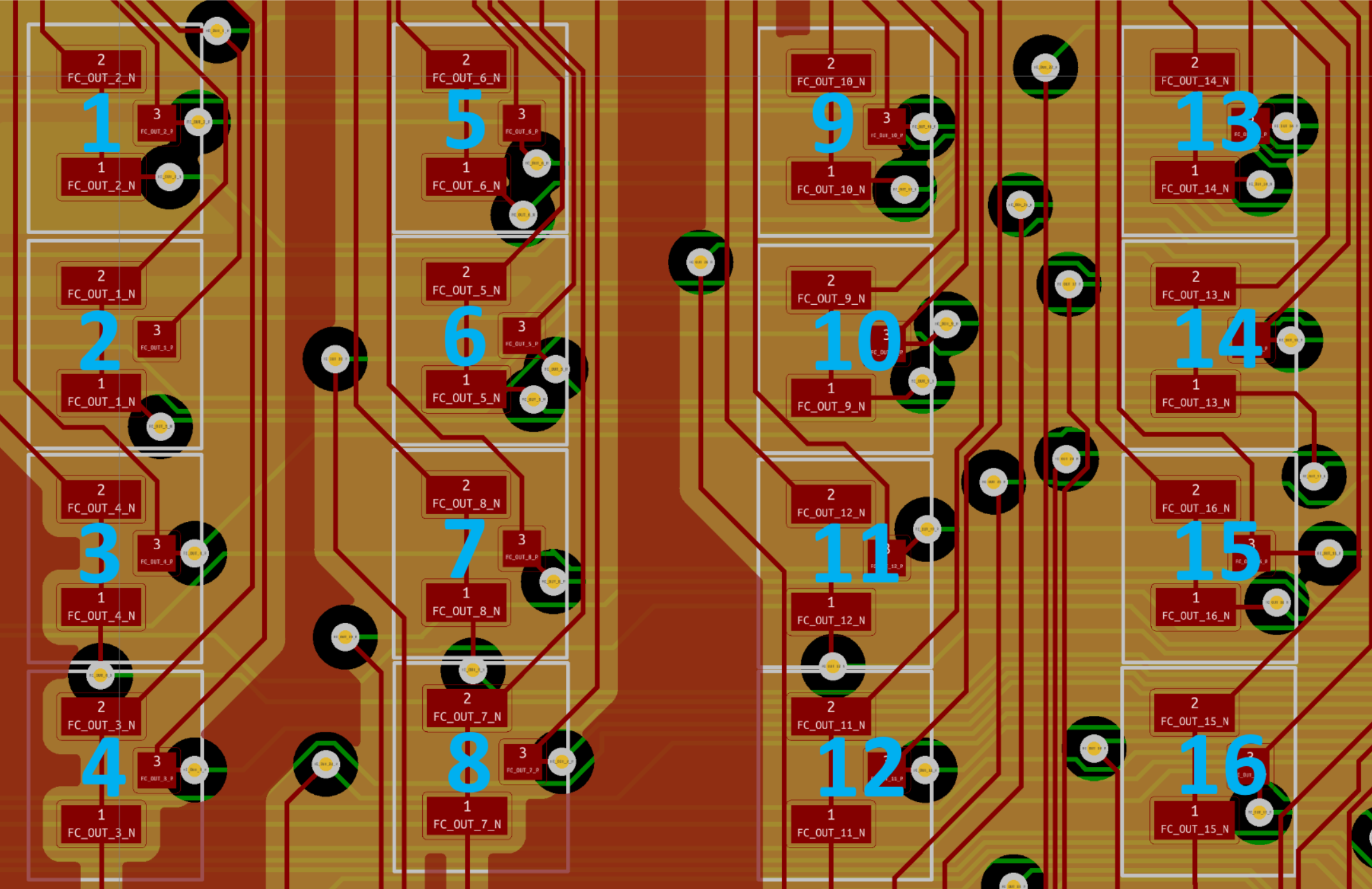
1. Motherboard:
   1. A31814-ND, HD68 connector, J1, J2
   2. WM7543-ND, board edge connector, J3-10
   3. RMCF0603ZT0R00CT-ND , R1-6, 13-18, 25-30, 39-44, 51-56, 63-66, 71,72: 0 ohm
   4. All other R’s, do not place.
   5. All C’s: do not place.
2. Daughterboard OLD:
   1. H11891TR-ND, u.FL connector
   2. 587-2511-1-ND, PI filter inductor, L1
   3. 311-69.8HRCT-ND, R2, 70 ohm
   4. 399-C0603C222J2GAC7867CT-ND, C2-3, 22n 100V NP0 0603
   5. RMCF0603ZT0R00CT-ND, R3, L2, 0 ohm
   6. C1, 587-6323-2-ND
   7. R1, do not place
   8. Same for other four filters.
3. Daughterboard NEW:
   1. RMCF0603JJ1K00TR-ND, R1,2,3,4: 1k
   2. 587-6323-2-ND, C1,2,3,4: 0.1u

Updates 4/22/22

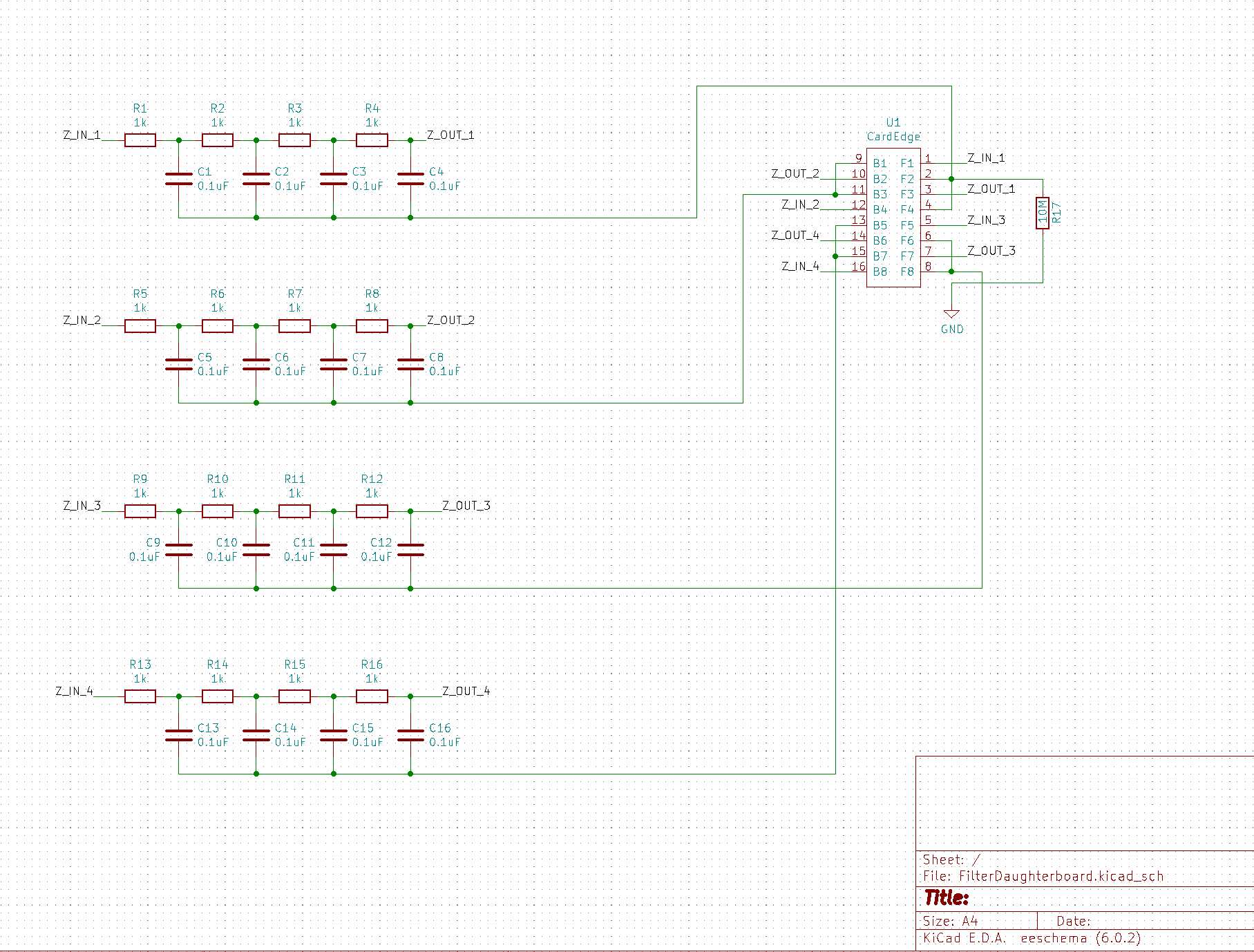
1. I have already explained these issues and written about them in BOLD in the modified document I attached ten days back, on Tuesday April 12th, see above. I will try and explain more clearly here.
2. I need the jumpers to be arranged in a grid, with 0.1” pitch both horizontal and vertical.
   1. This way I can really use a single 3x32 header (SAM12369-ND) and 2x32 header (ED1309-64-ND) for all of them.
   2. If I want to change the jumpers I can change them all at once by moving an upside down 2x32 socket (2553-2044-2X32G00RA-ND) with its leads pairwise soldered, from one part of the 3x32 to the other.
   3. I also need these to be arranged in order, 1-32.
3. I need the u.FL connectors to be rearranged so that they are in order.
   1. Right now they are this way:

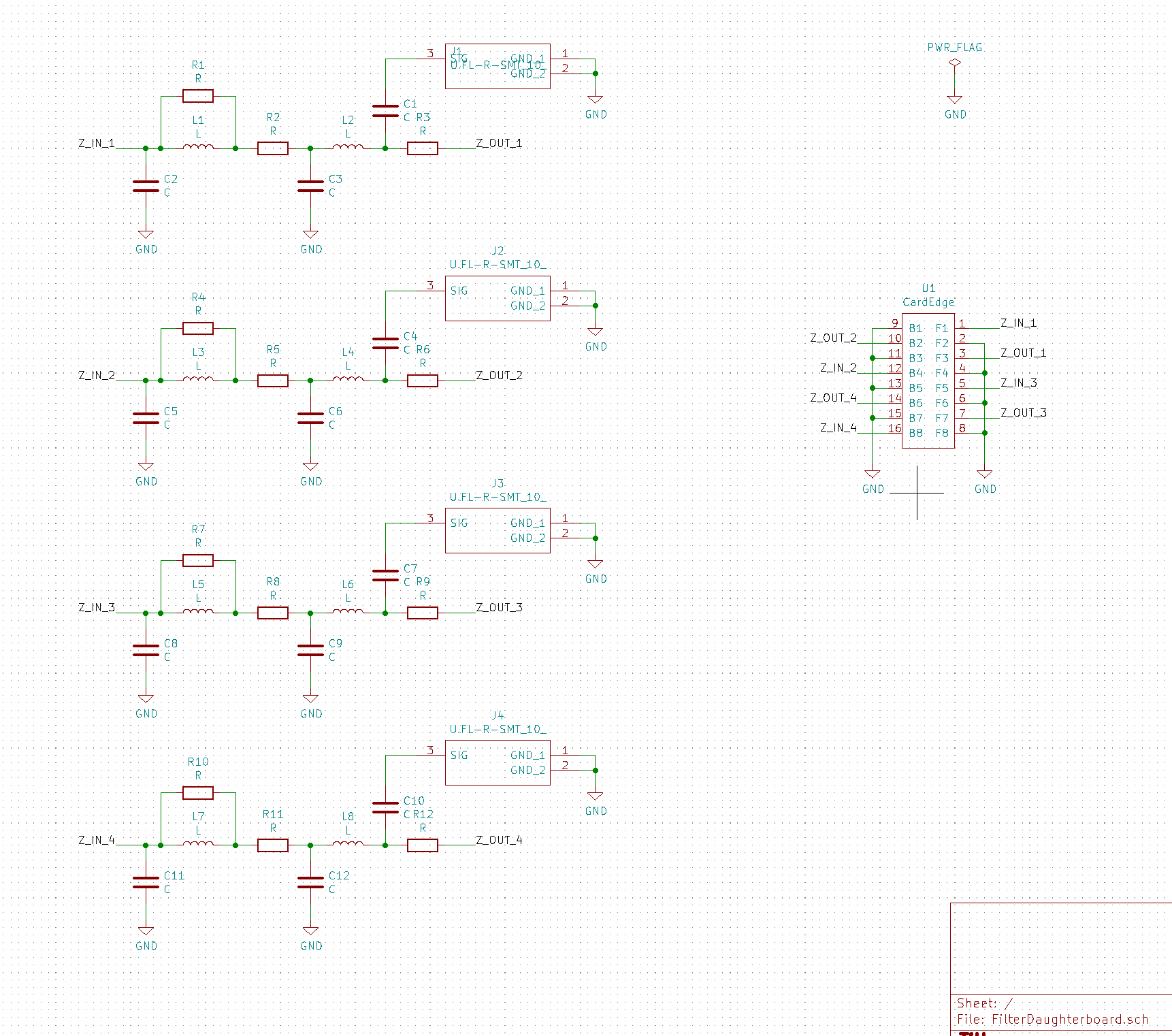


* 1. I want them to look like this:

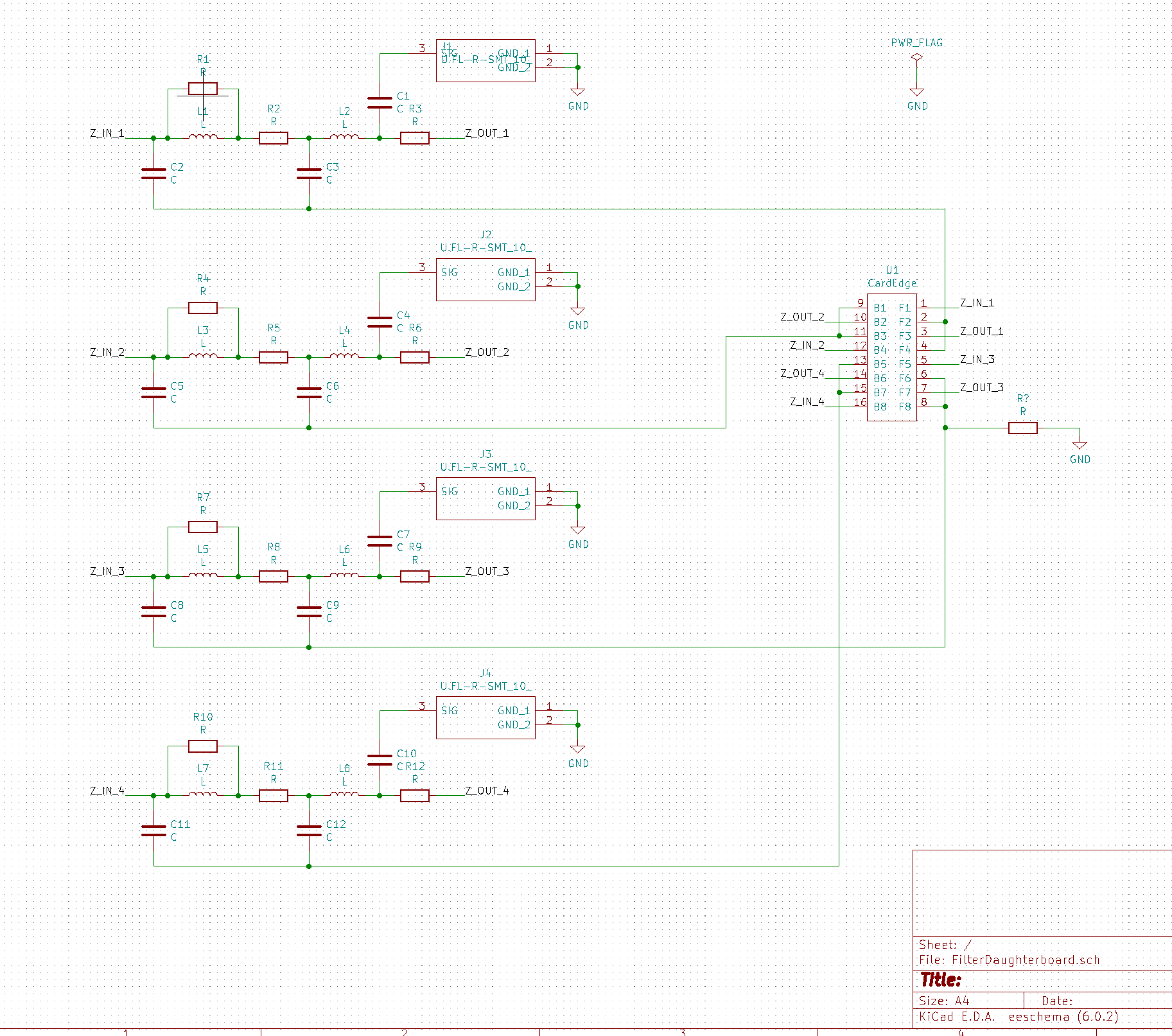


1. I need the silkscreen to include more labeling.
   1. These u.FL pickoffs need to be labeled 1-32.
   2. The 3x32 header needs #’s indicating which side is 1 and which side is 32, which row is GND and which row is N.
   3. The 2x32 header needs #’s indicating which side is 1 and which side is 32.
   4. Somewhere it should say “FilterMotherBoard V4.0, JS, DR, KJ, 04/22/22”
2. Daughterboards.
   1. The grounding still does not maintain differential pairing. I need the schematic to look like this, and keep the differential pairs separate and not shorted together. Otherwise what is the purpose of keeping them separate on the motherboard?? Notice how there



* 1. I need the same thing for the old daughterboard. Right now looks like this: 

I need it to look like this:



1. Please let me know if you would like to adjust the price on this work.
2. Thank you!

Updates 5/31/22

I want the ordering on the final DB25 updated. I’m also going to give Kristian better “private feedback” – something I wasn’t aware of previously but is important for freelancer’s ratings (paradoxically).

I posted a job with this description:

“Seeking a PCB designer who can make the final tweaks needed on a filter board that distributes 32 differentially paired signals to and from card-insert sub-PCB assemblies of different types.

The key tweak required is that the routing to one connector in particular needs to be adjusted so that the traces are connected in standard order for DB connectors. In particular, the numbering of connections in the schematic should match the numbering of the connector, see attached image.”

And this image:

