

NOTES:

Analog dies at about 2.3v

Digital also dies at about 2.4v, but the display becomes very dark at 2.7~

Looks like life is pretty good at 2.9v.

+3V3 is the main analog rail (As it's currently designed, these run around 2.9v

+3V3_CPU is the main digital rail

+5V is the power in through the USB connector

+5V_BOOST is the output of the 5V boost converter

PARTS TO FIND and include:

TODO:

LAYOUT TODO:

DONE:

Consider AD8343 to replace mixers. Looks interesting. Why didn't I find this earlier? maybe because it draws 50mA?

How about because it's lower cutoff is at 50MHz.... oh well

figure out reasonable bias resistors for AMP

remove GPS matching cap from BOM, I think

Looks like the charge voltage divider is still not going to work. Doesn't seem to charge until 5.5v

clean up silkscreens, move things around a little to avoid test points

U287 pin 1 mark!! AGAIN!!

c251 is a little close

Narrow paddle slots just a little

remove extra hole from encoder board connector.

Better copper pour for RF amp and other high power parts.

try neal's flip flop circuit

Change C93 and 94 to 0603

The earphone connector doesn't line up exactly! :0 May need to do by hand with a jig for now.

Change to F7!

Change to Diode ring mixer and new transformers

Add some filtering to USB power line

More output capacitance on 5V Reg

Add more capacitance around charge controller

upgrade to better GPS module!!

revert mic footprint back to recommended?

Now I have a free pin again. What ever shall I do with it? -Stick an LED!

Keep away from LCD screws

Attempt to better length match the I/Q lines - That was a lot of work! I wonder if it was actually helpful

Add pin one mark to U1 (it really doesn't have one? Oh it's clipped by C1) Also soldermask dam on this part is no good

Soldermask oversized? Did I do that or the board house?

reposition PSDR logos

Add test points to relevant locations

All analog pins, I2C, SPI, Clock lines, anything else important? PSU outputs. Anything else?

GPS Doesn't run down to 2.8 like some of the other stuff.

so we probably need to kick the buck reg up a little. It was at about 2.9, now its just over 3. Should work.

Change BOM to use SN74CB3Q3253DBQR or similar

Change bom to reflect new transformers and mixers

Looks like I need a new speaker. (Make it something that can be soldered!)

Special Thanks!

Gerald Youngblood, AC5OG / K5SDR

For the SDR for the Masses articles that inspired this radio

David Brainerd, WB6DHW from whose UHFSDR the current PSDR takes its mixer design.

The DSP Guide guy!

THE BACKERS!

The brave few who actually build their own examples of the PSDR

Karalee

Ryan, Joe

Later?:

TVS?

ooh, TVSSs?

Might want to double check the performance of the GPS antenna....

Use only one voltage divider for LPF circuit?

(I just don't want to have to do the reference designators again!! :(

Add UHFSDR style matching pad to LO input?

Even more test points?

More complete GPS antenna matching circuit

antenna tuner?

multitap inductor?

Add light sensor for backlight, you know?

Clean up schematic (to resemble block diagram)

So I think the interted pin isn't working as expected. Need to fix that.... don't we?

Actually, the speaker does kinda work, if the earphone interted line is pulled high by the MCU (it leaks noise into the system though.)

Add temperature sensor near freq reference

Rename pins and stuff

Add ability to turn off subsystems (like RF)

If charging with a v6 solar panel, both the boost and buck regulators will be subjected to voltages over spec!!

5v Regulator? Maybe a 5v linear reg too

Look into magnetic encoder

detect charger type (usb type) and adjust charge current appropriately?

low pass filter on the audio out?

do I like using half-vcc, or just putting voltage dividers all over the place?

Attenuate TX driver outputs to maximize resolution going into the filter, or I could just turn off the filter gain...

Change over to new LCD

Power cutoff circuit

Maybe I should use one crystal for the MCU and have it output to the Si5338 so they are locked together.

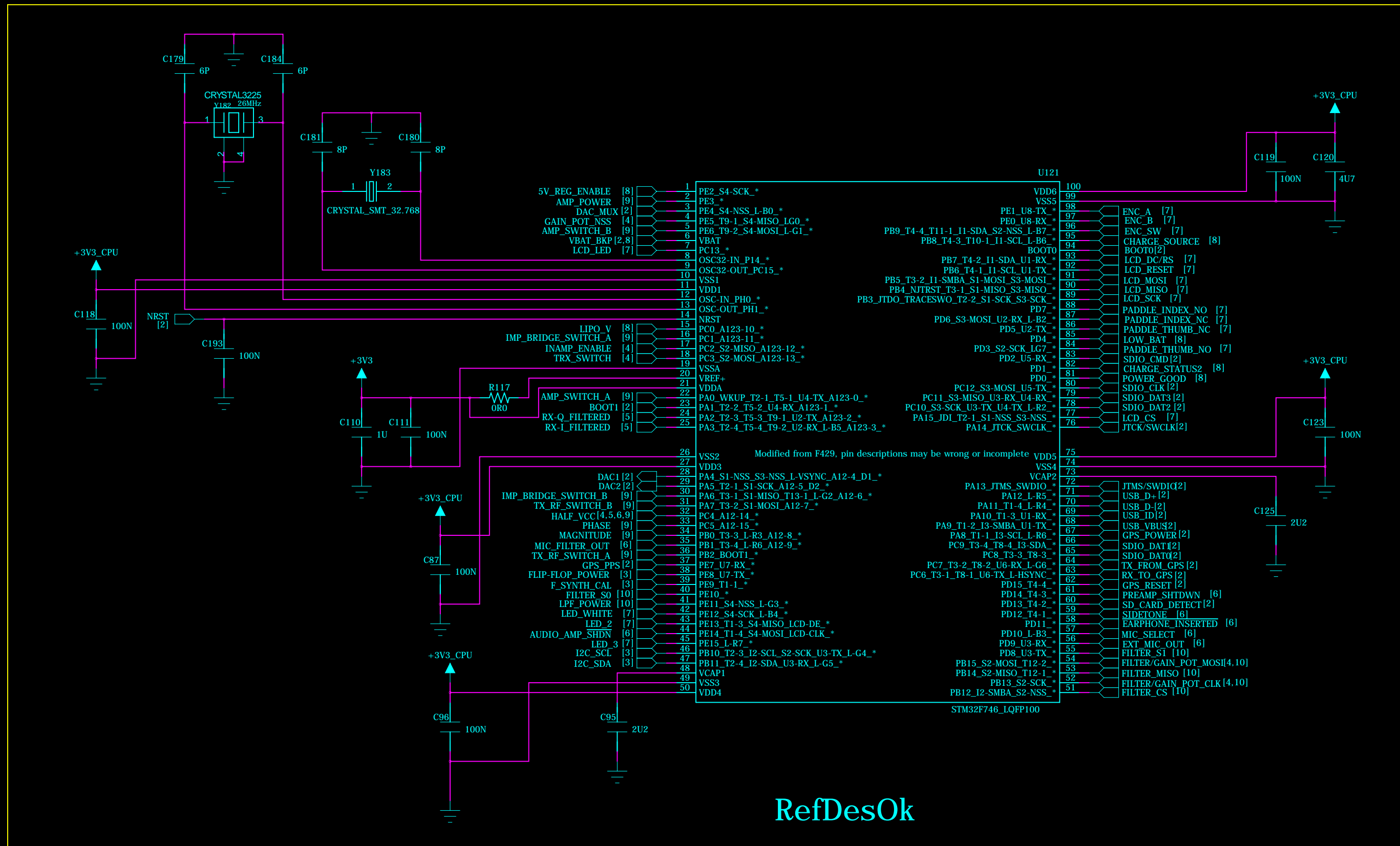
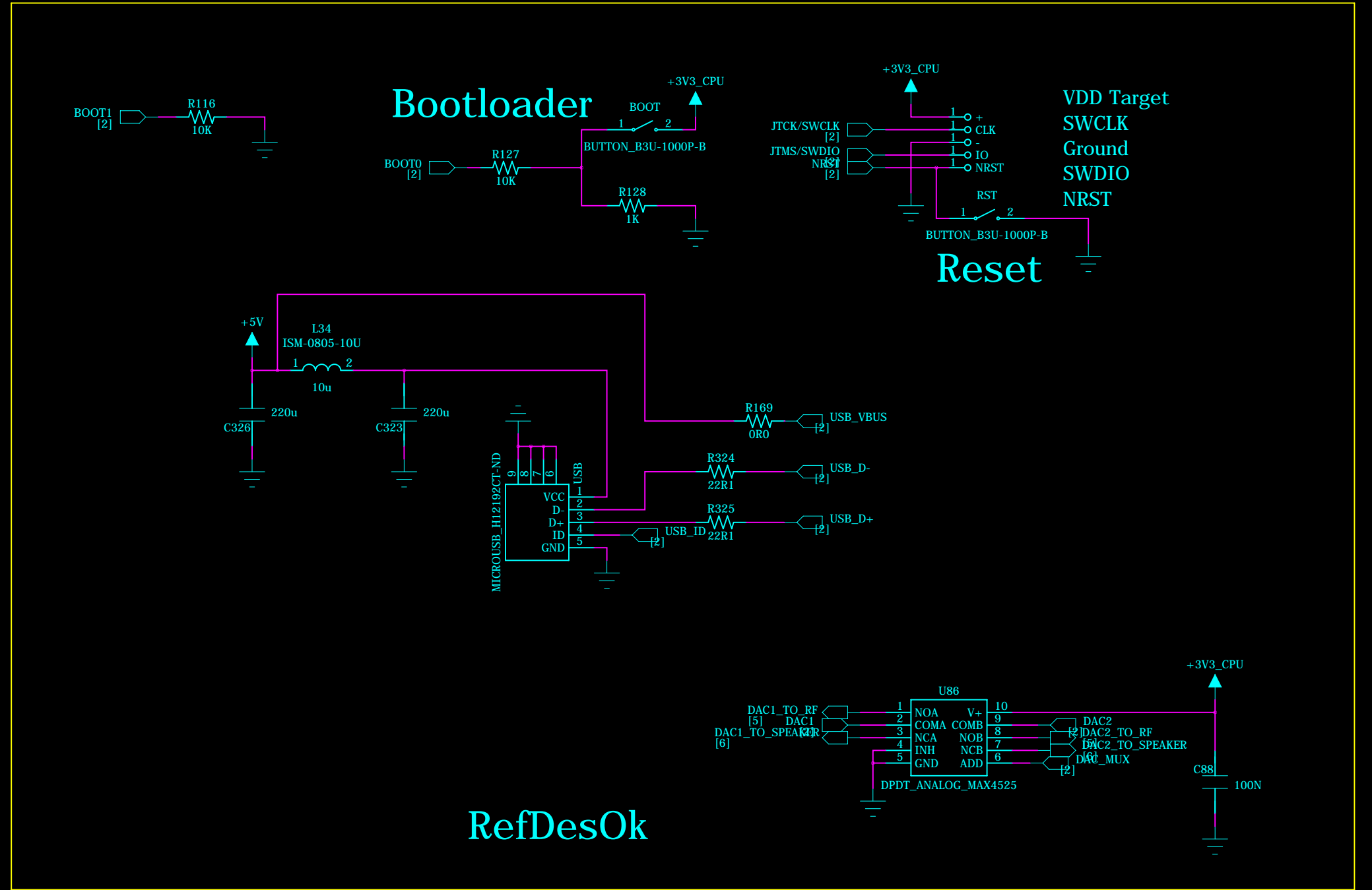
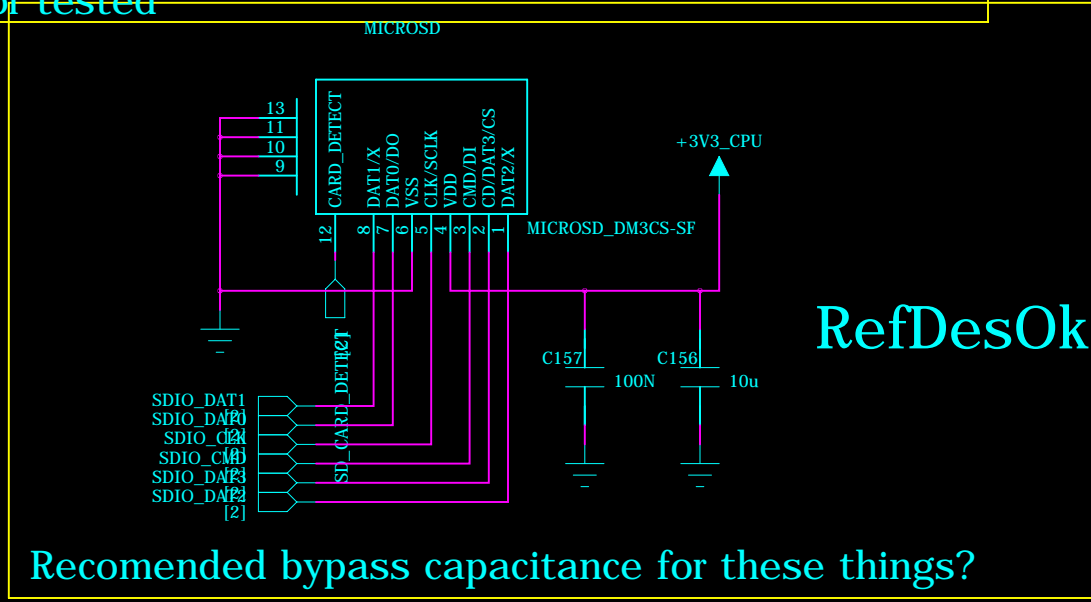
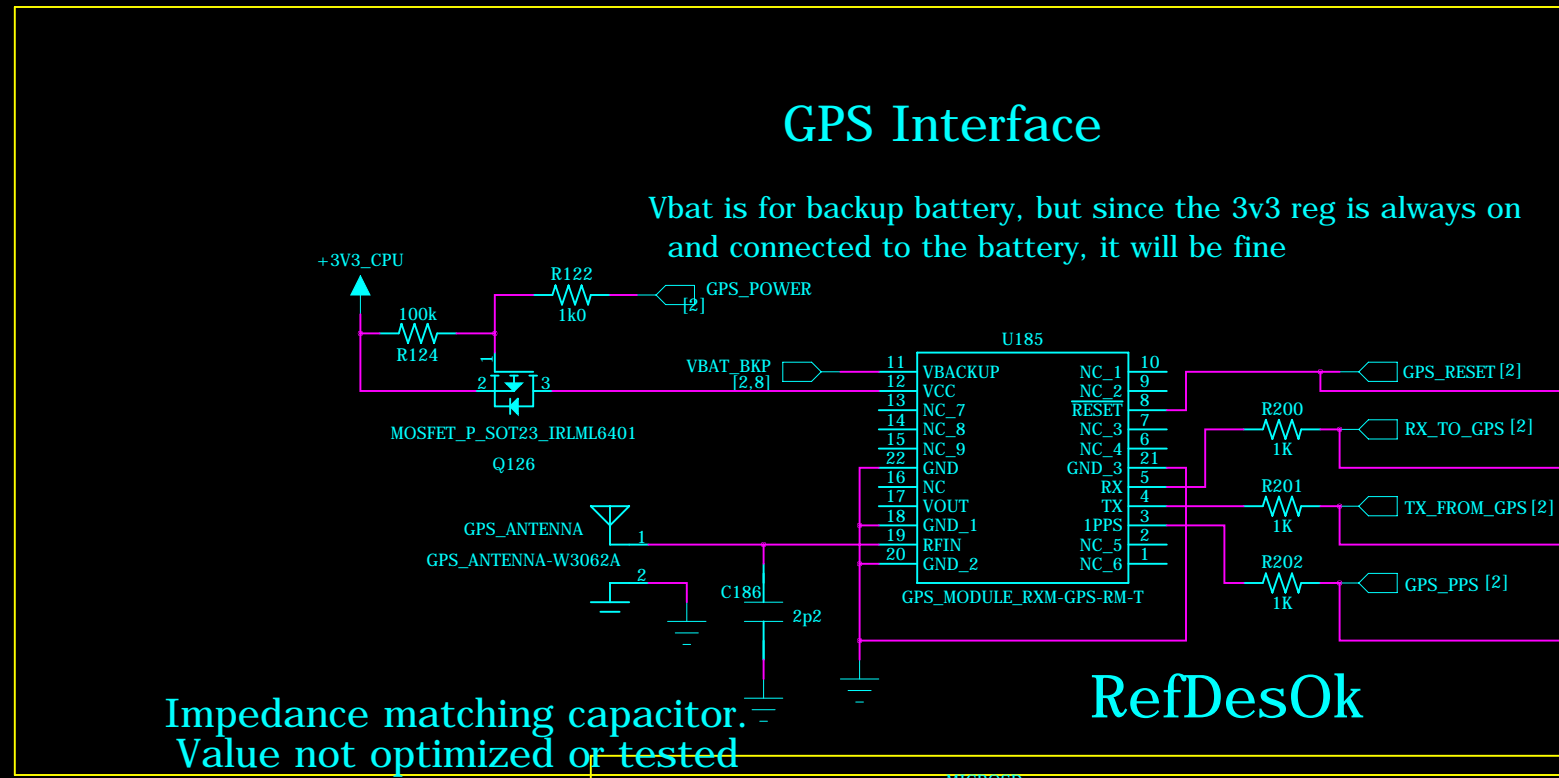
Lots of parts don't show up as the same type when they are, not sure why.

Silkscreen between pads of 0603s didn't turn out. Options?

copper keepouts around some of the ICs

Pots for bias adjustment?

Hold BOOTLOADER button while clicking reset to enter bootloader (in theory)



Trying to reconcile values with UHF SDR2 LVDS

But it is designed for 5v.....
in the UHF SDR, the caps are on the other side of R268

NOT 32MHz, SI App says 8-30MHz
Now I'm using 26MHz

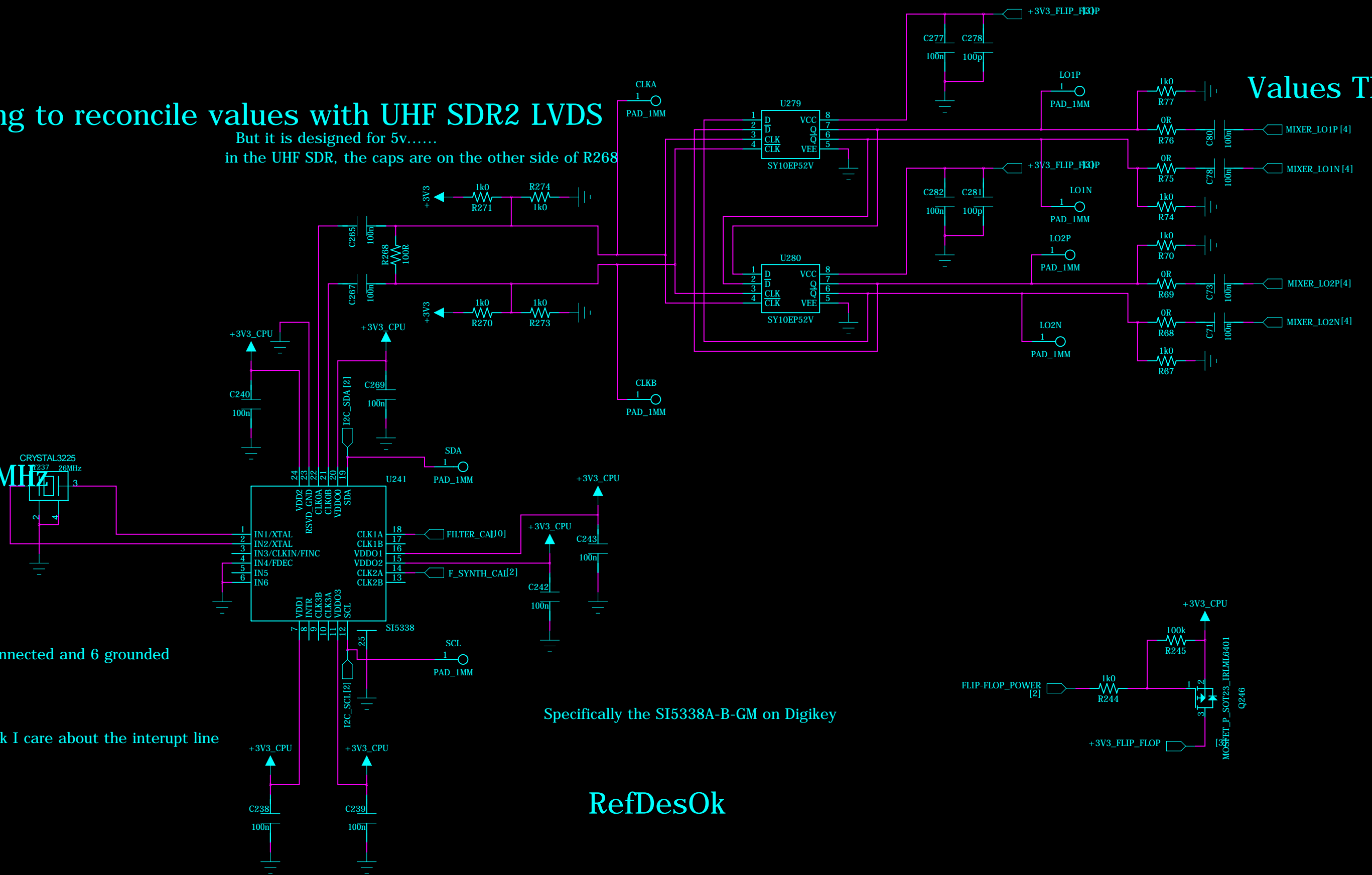
IN3: Clock in
IN4: I2C LSB
IN5/6: Feedback. Leave 5 unconnected and 6 grounded

I don't think I care about the interrupt line

Specifically the SI5338A-B-GM on Digikey

RefDesOk

Values TBD

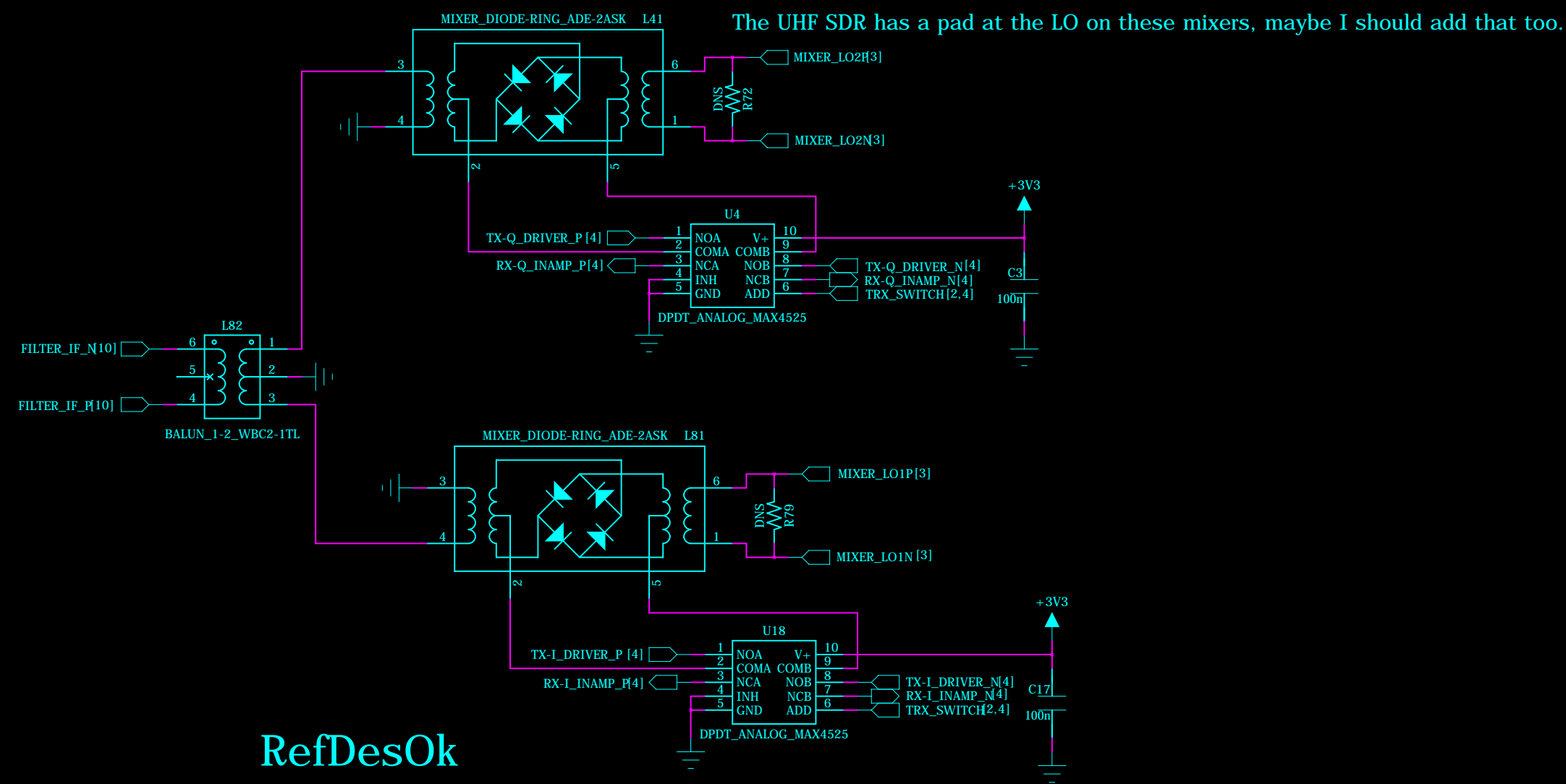


These switches might not be necessary, it might be okay to put the stuff after in parallel

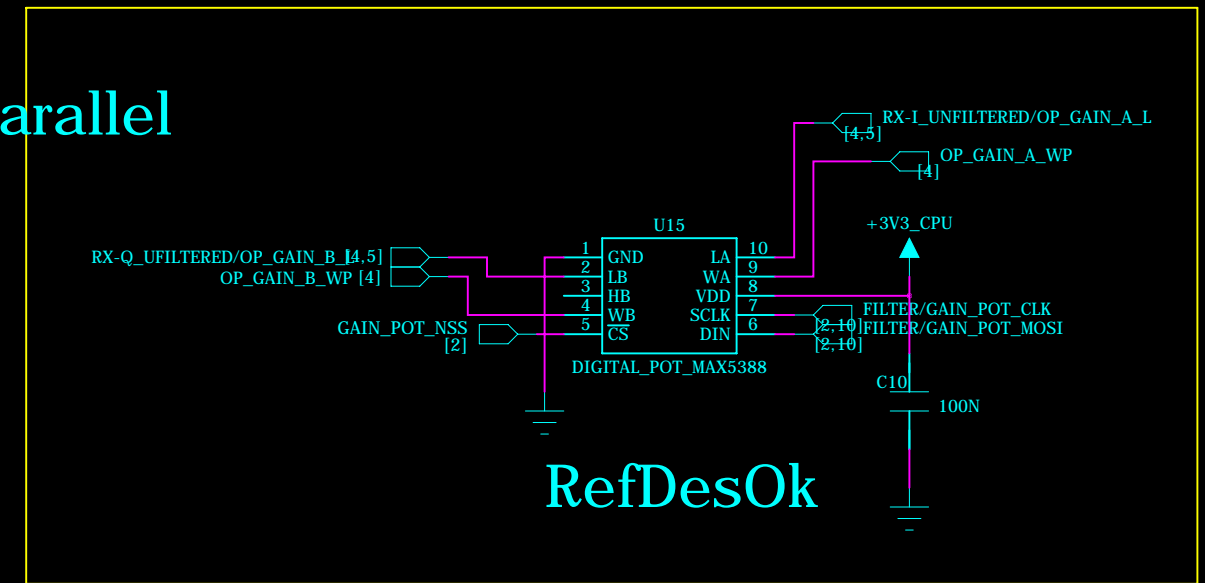
I think I could flip the IF/RF and reverse the LO on one mixer for tidier routing....

Do I want to bias these baluns? Is this design overkill?

I still feel like there is a way to make this work without the baluns

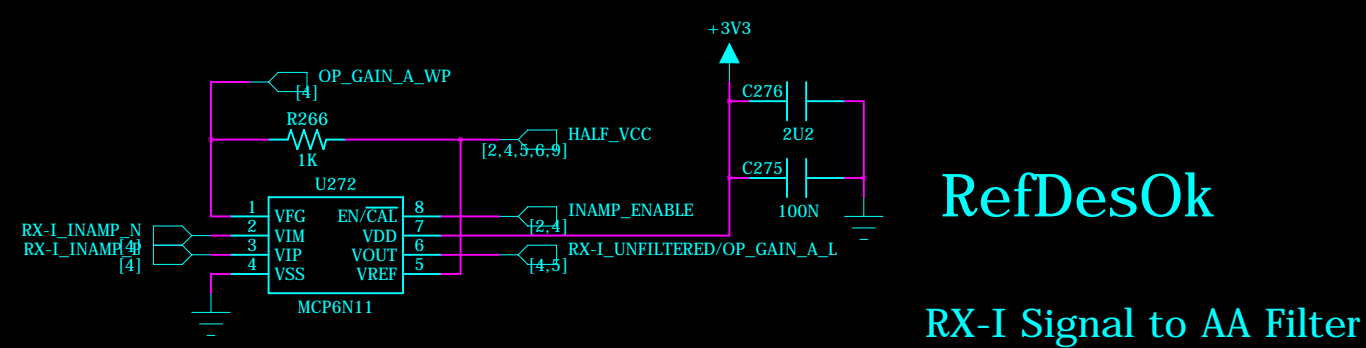


RefDesOk



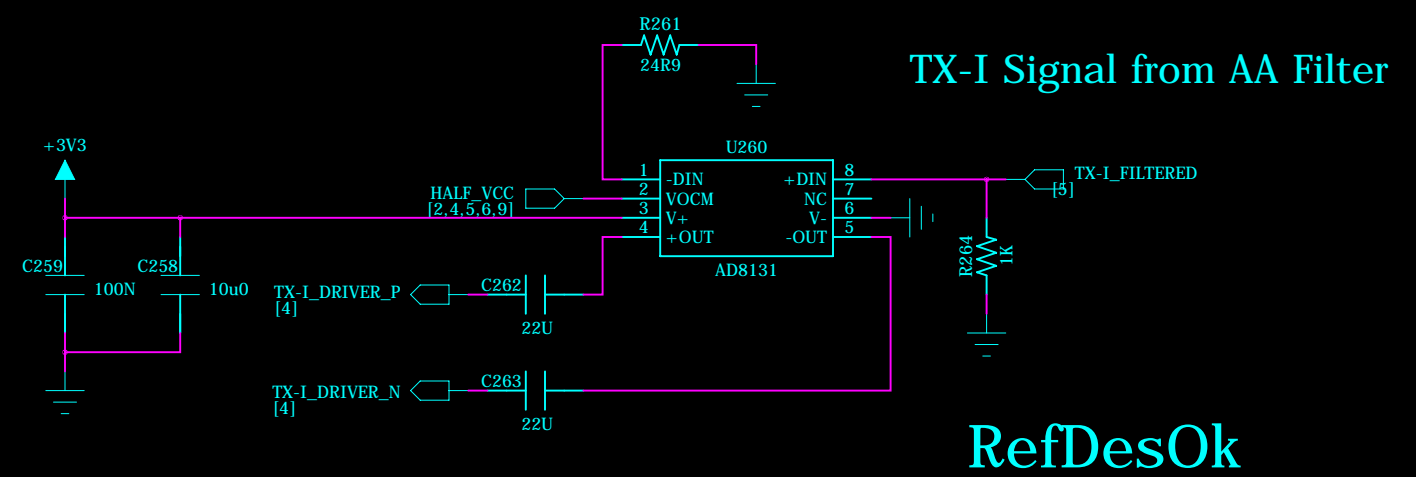
RefDesOk

R37 and R36 (they were 50R) are pulling down too hard!
Removing them is better than having them
what value should they be? Maybe equal to the output impedance of the DACs? Let's use 1K for now.

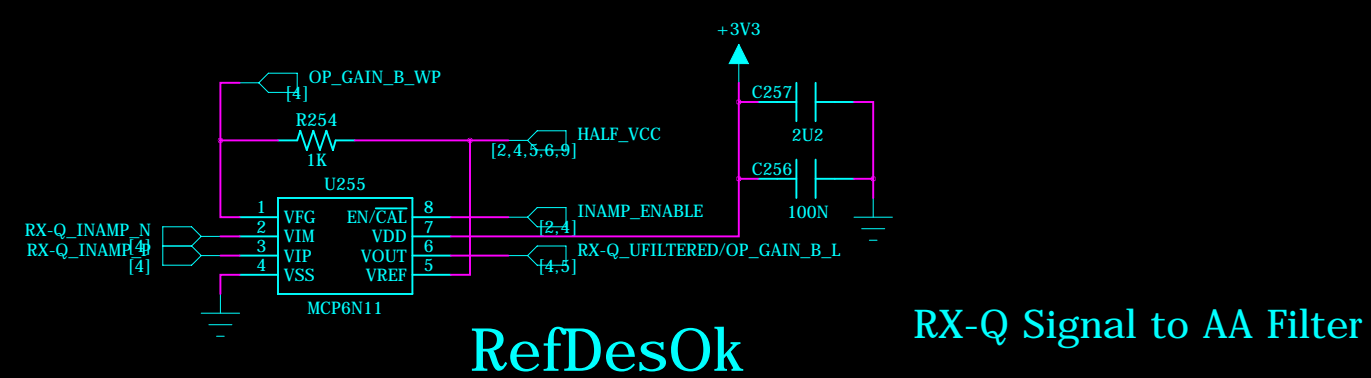


RefDesOk

RX-I Signal to AA Filter

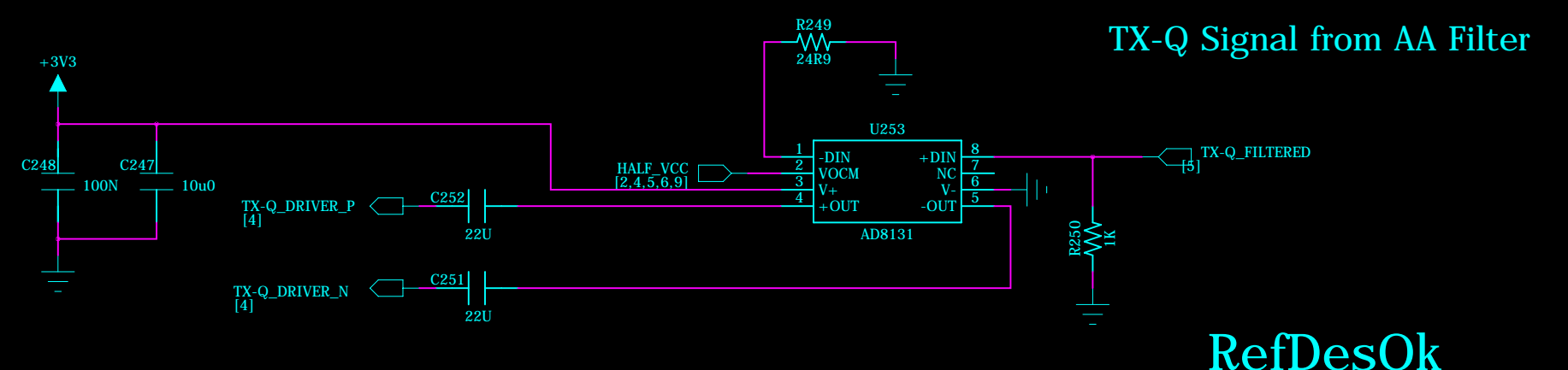


RefDesOk



RefDesOk

RX-Q Signal to AA Filter

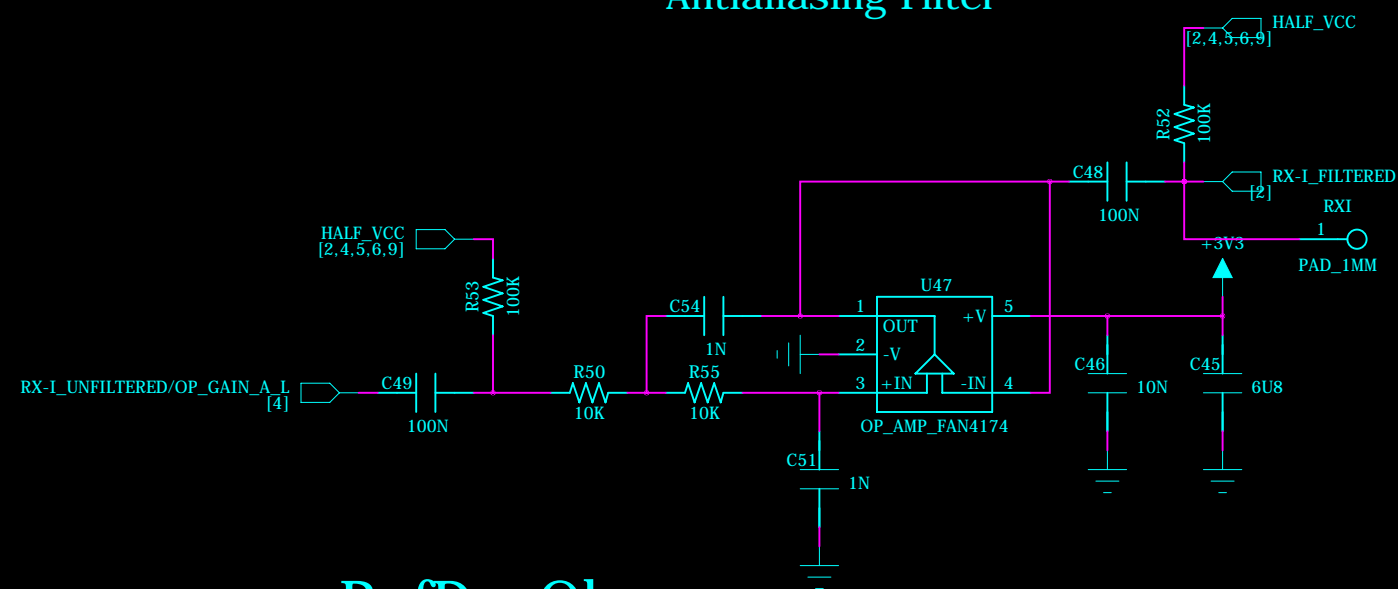


RefDesOk

TX-Q Signal from AA Filter

RX-I

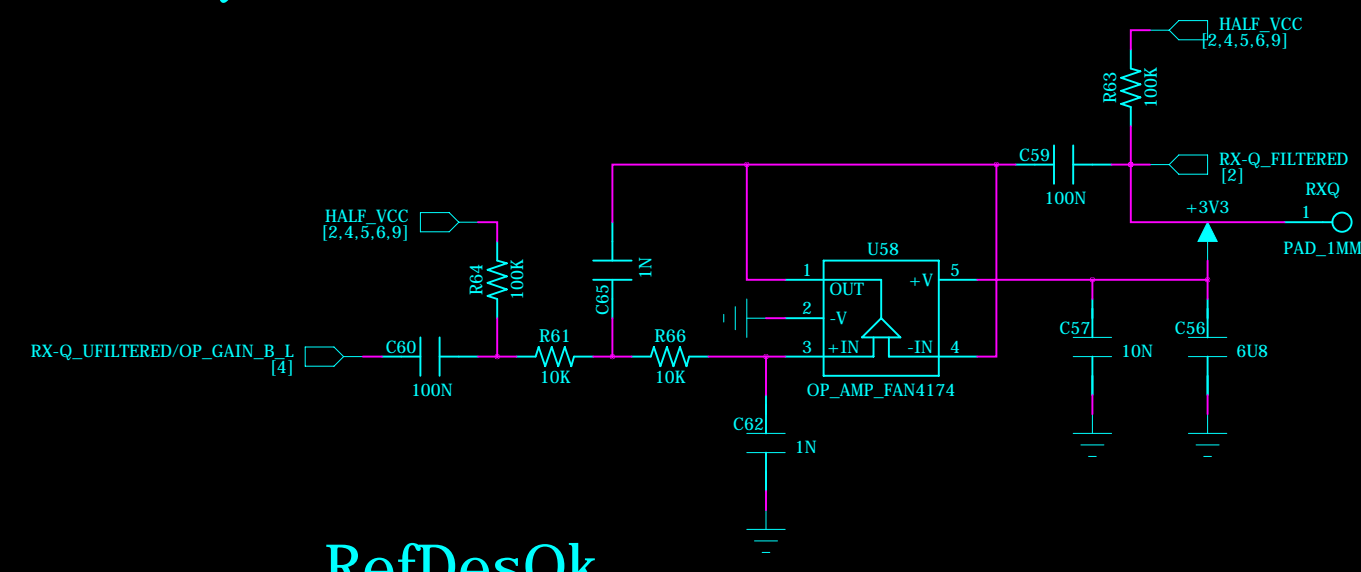
Antialiasing Filter



RefDesOk

RX-Q

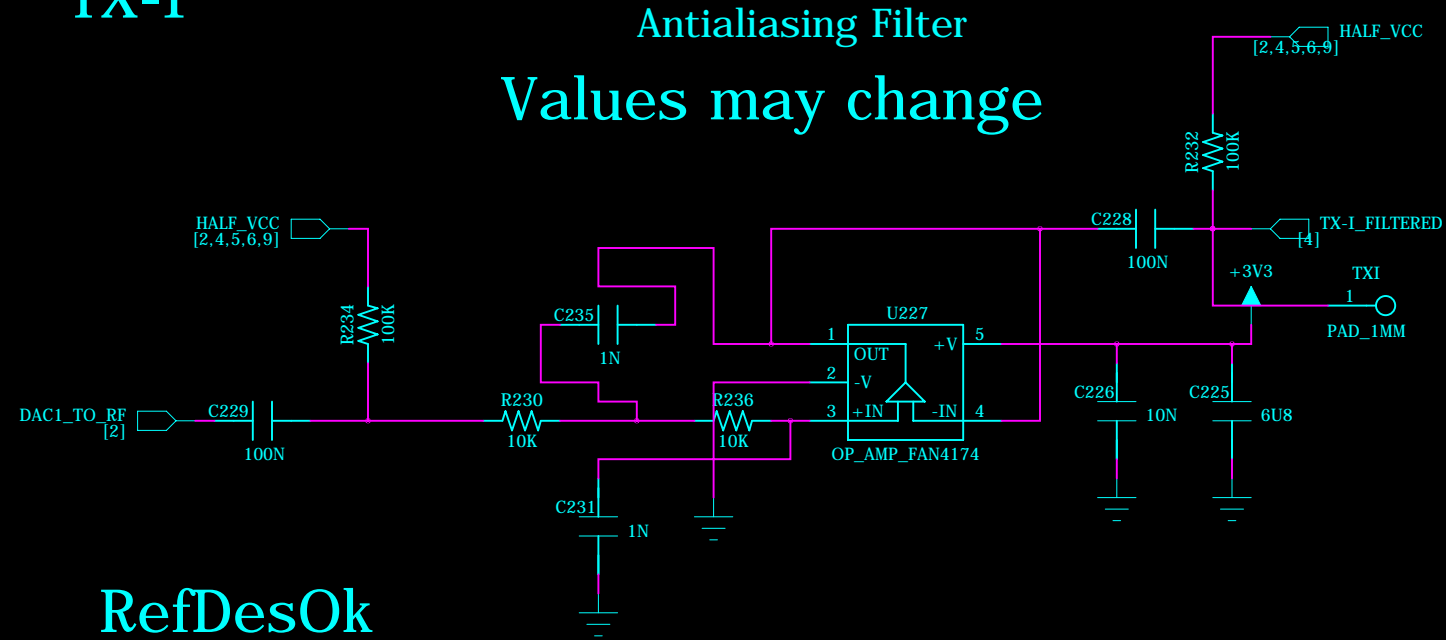
Antialiasing Filter



RefDesOk

TX-I

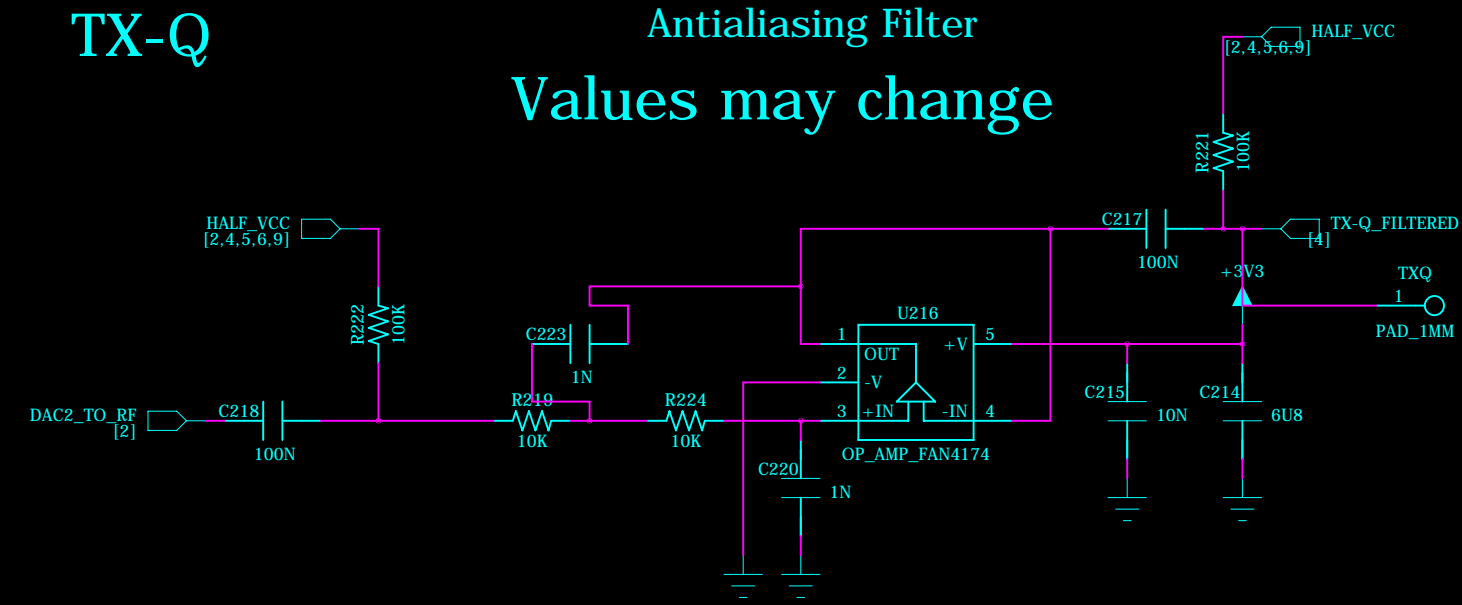
Antialiasing Filter
Values may change



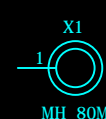
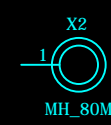
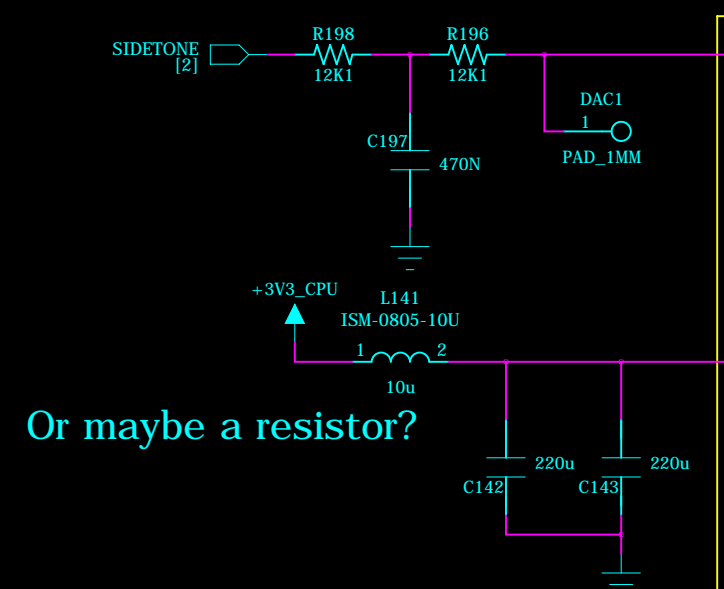
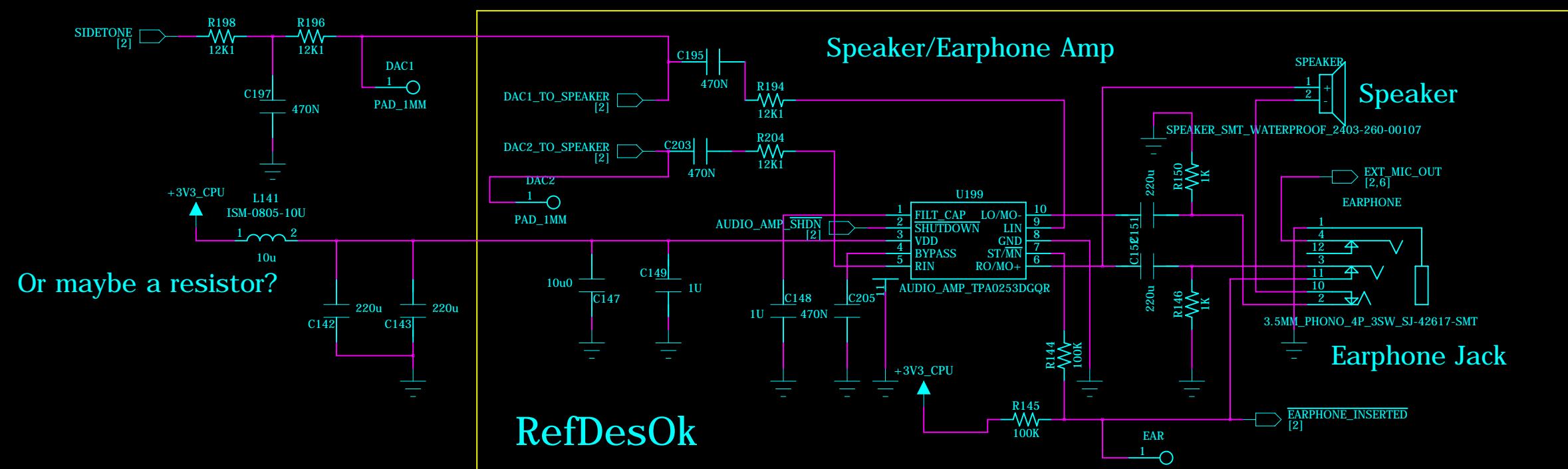
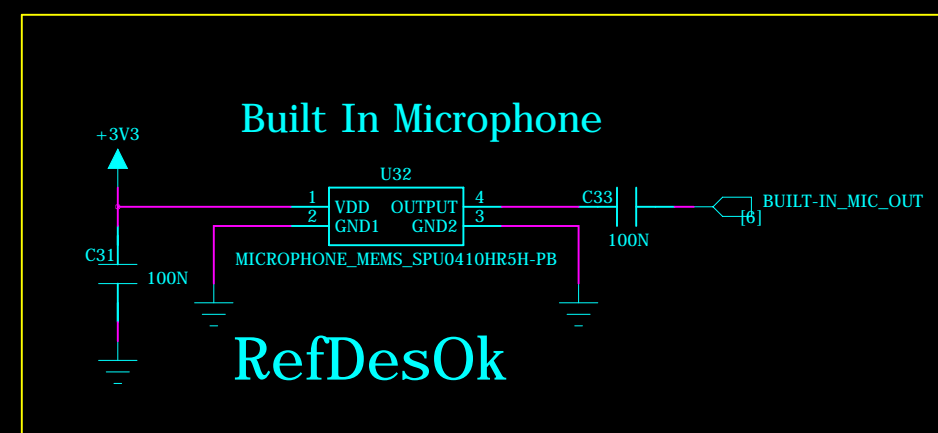
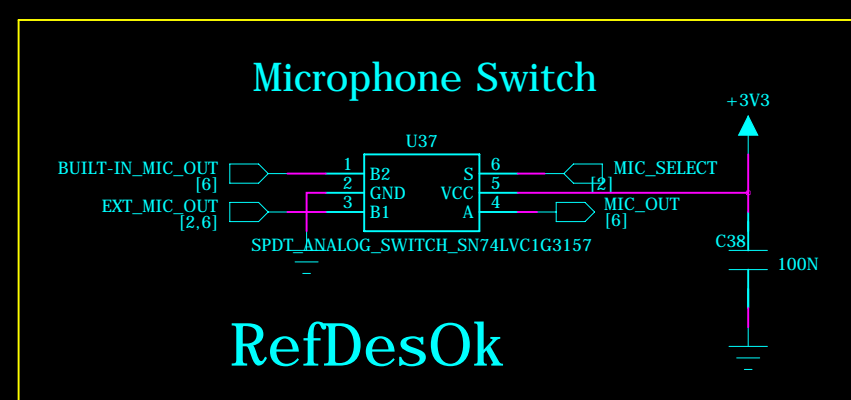
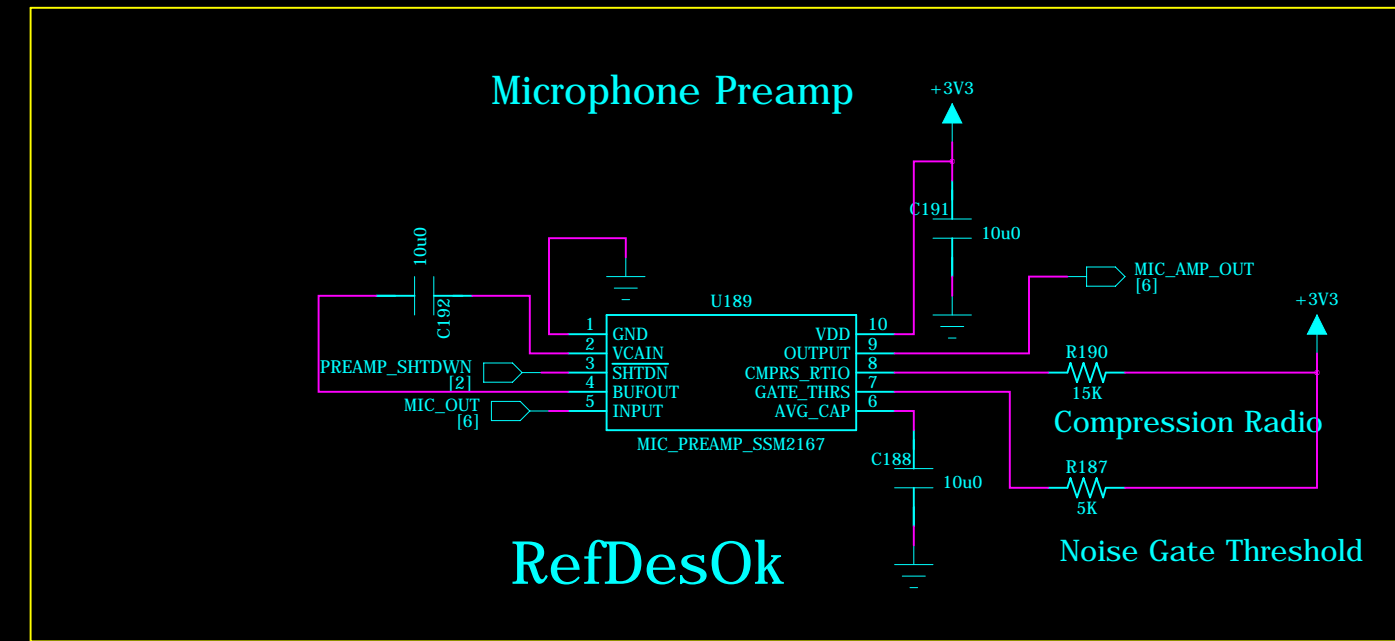
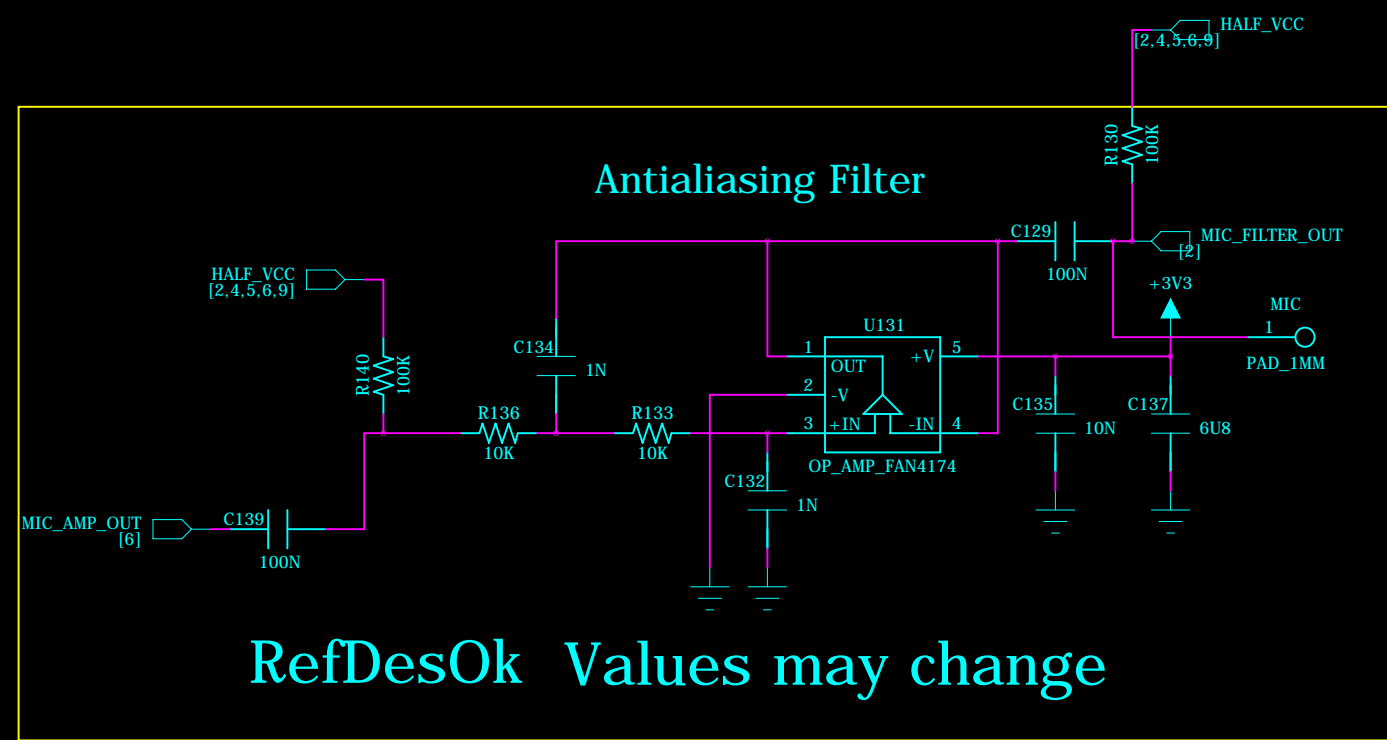
RefDesOk

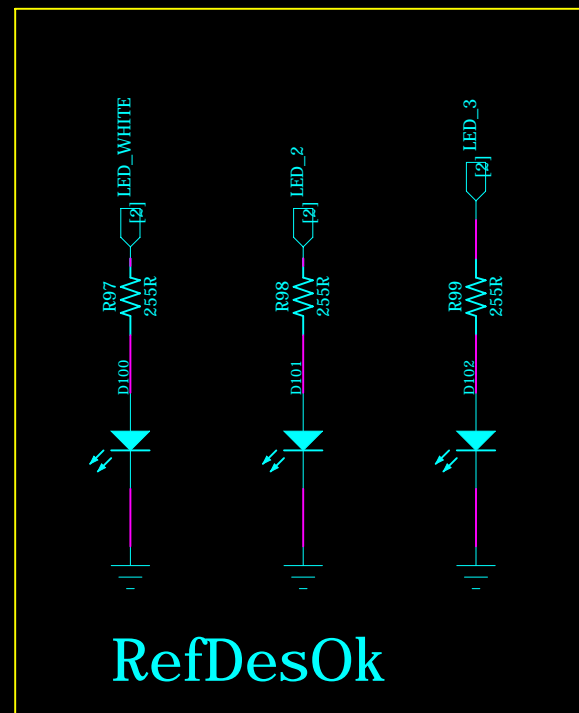
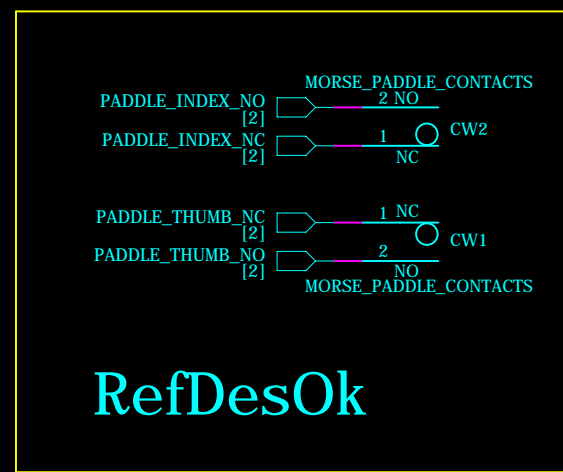
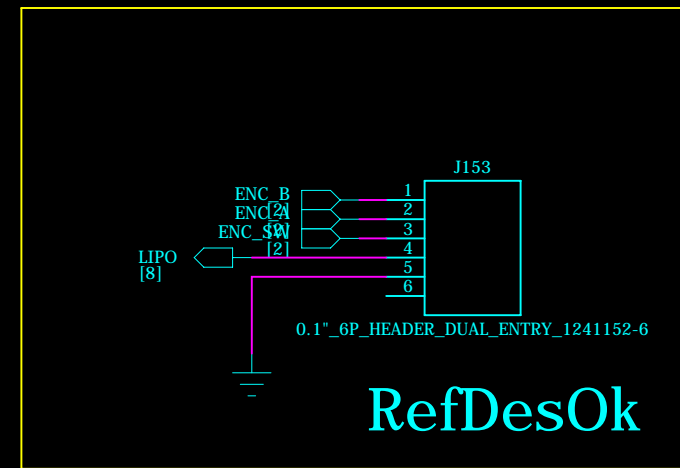
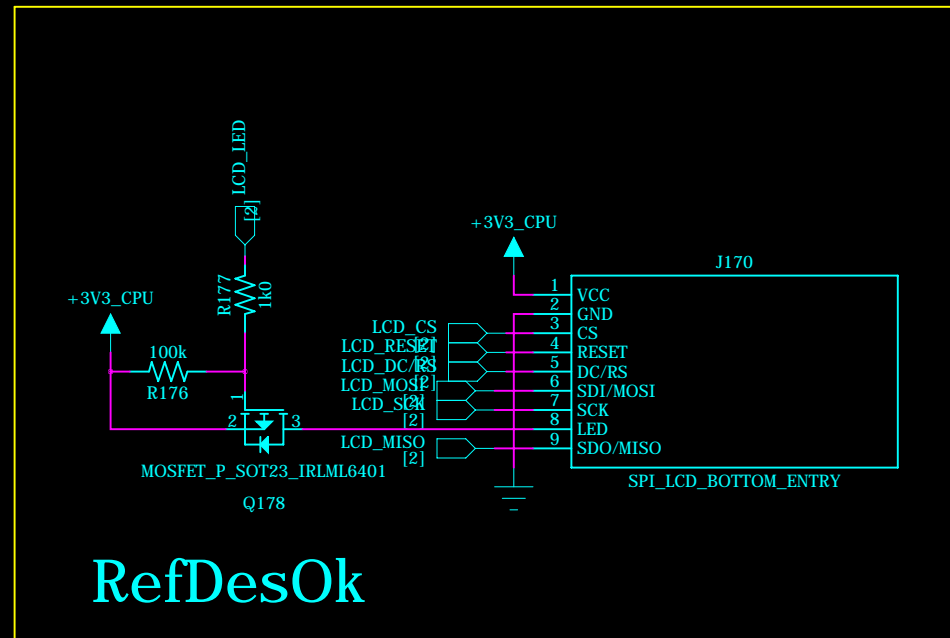
TX-Q

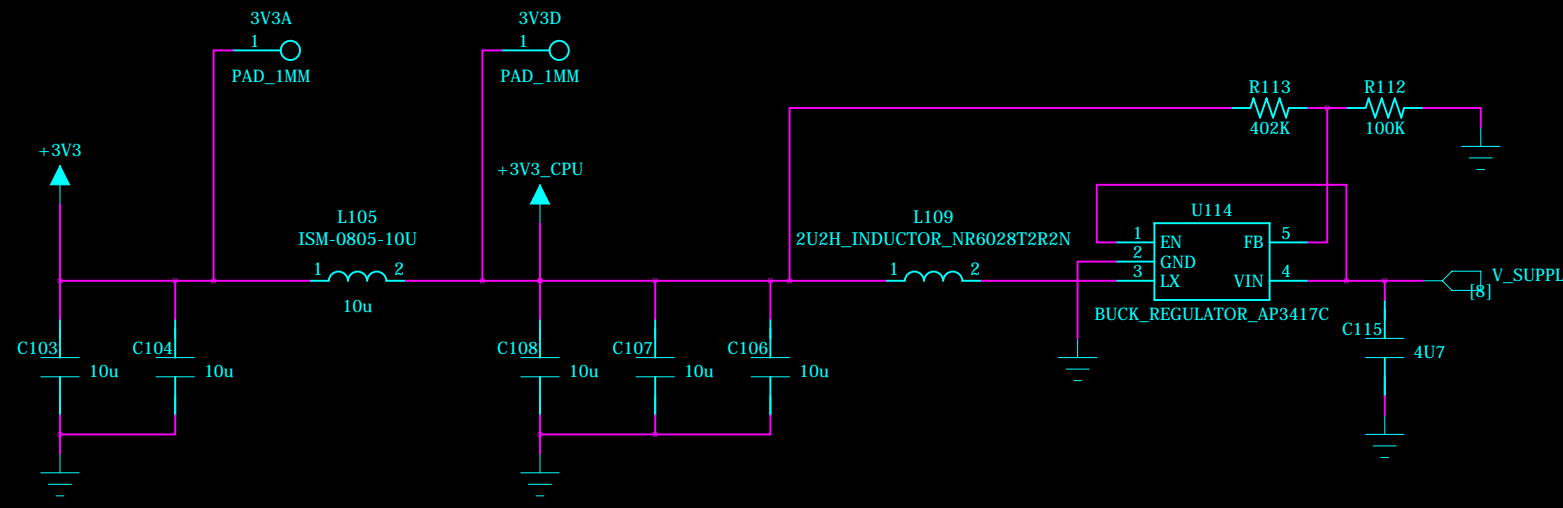
Antialiasing Filter
Values may change



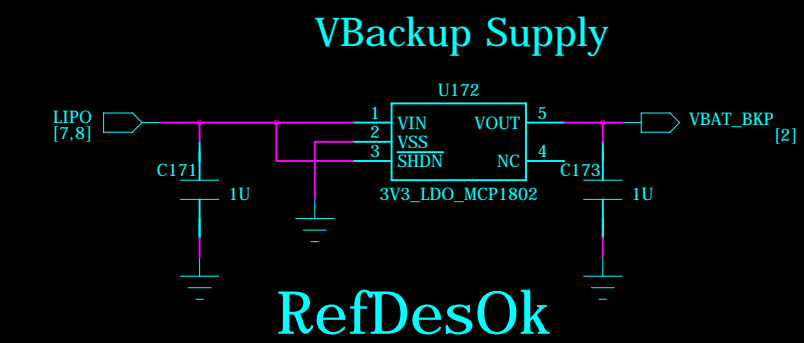
RefDesOk



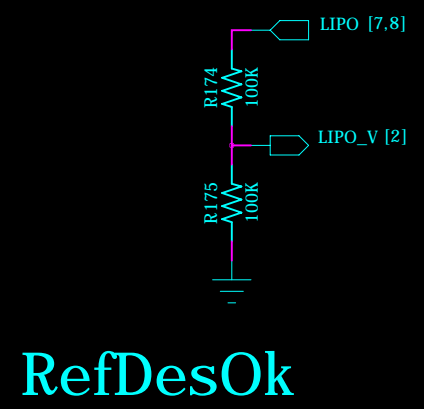




RefDesOk

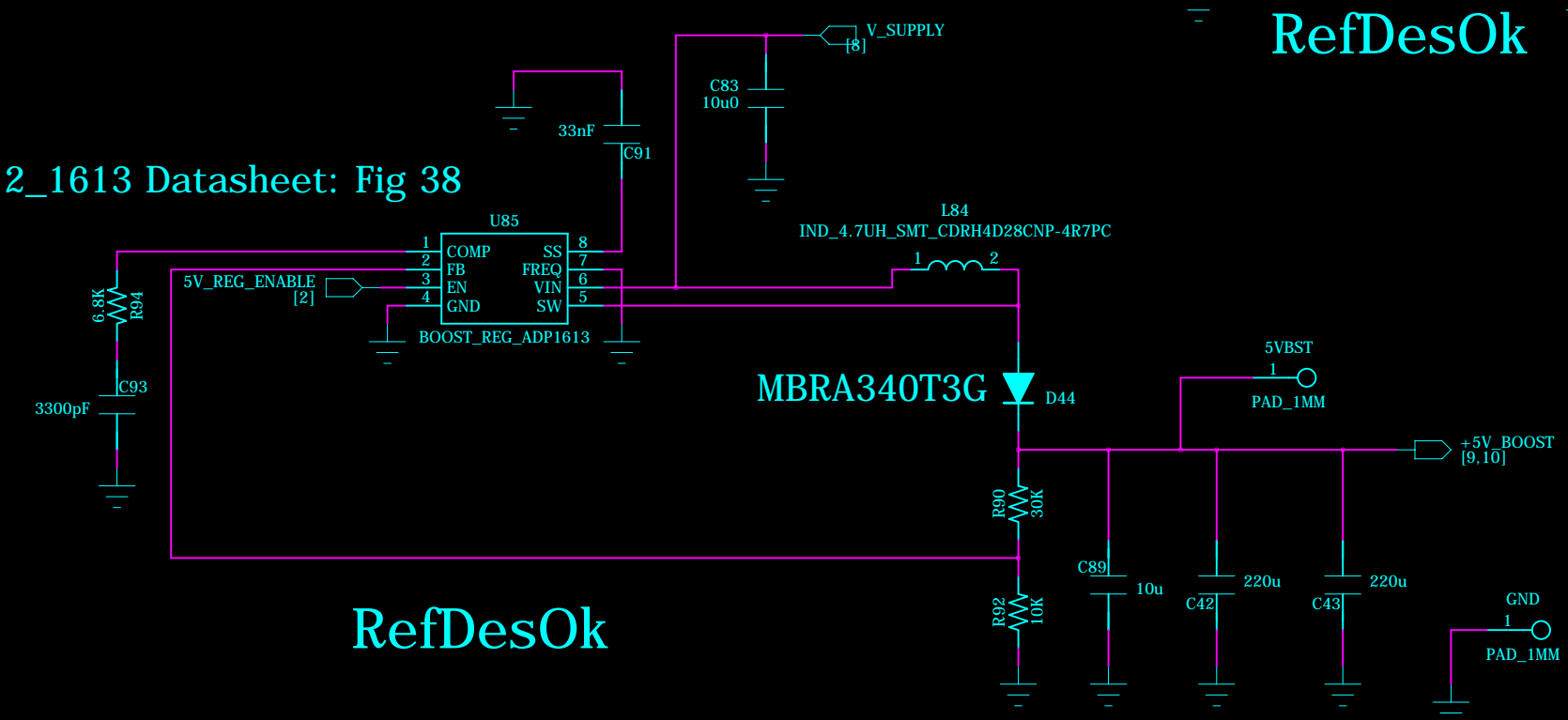


RefDesOk



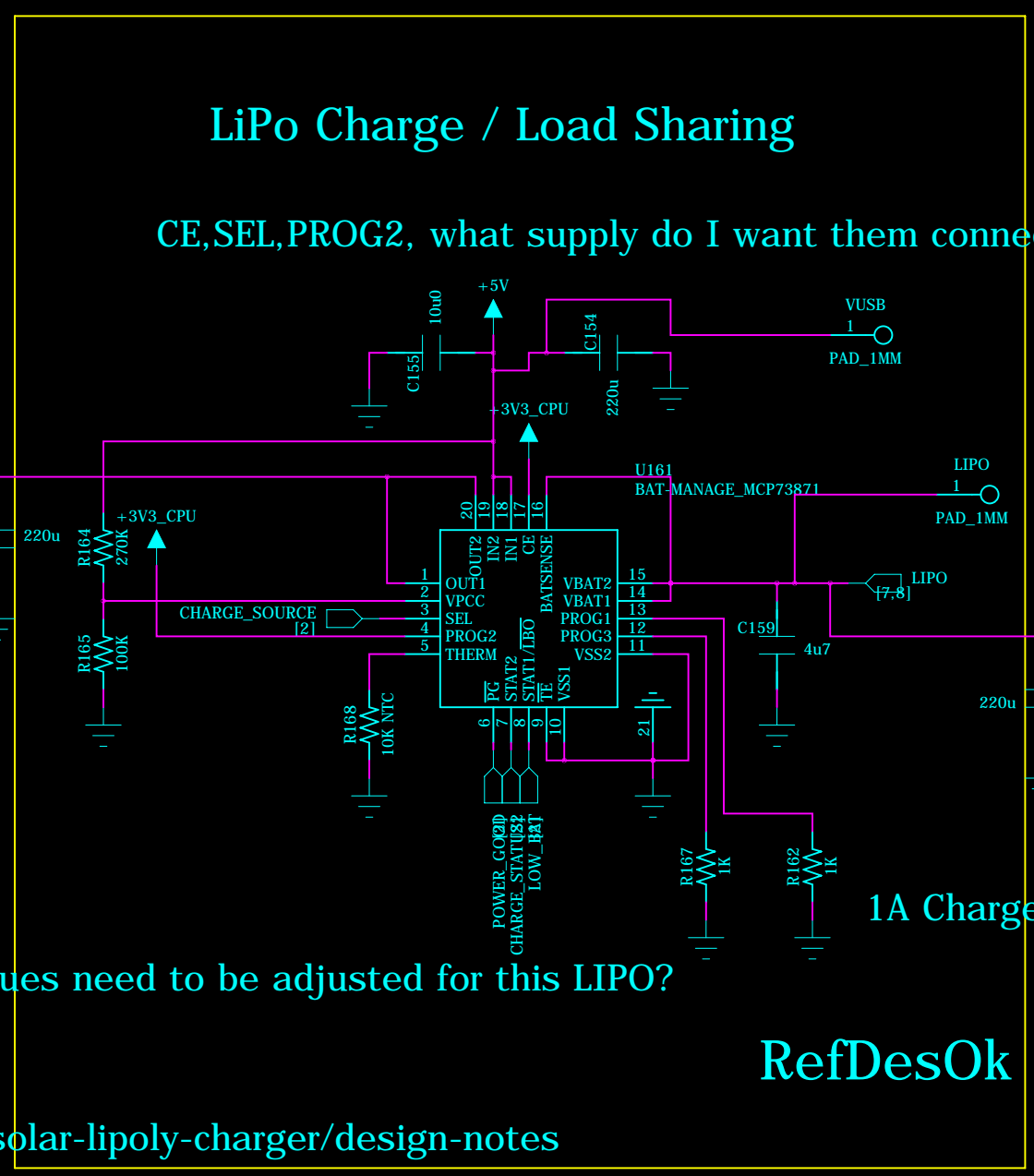
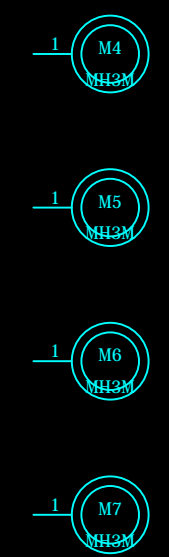
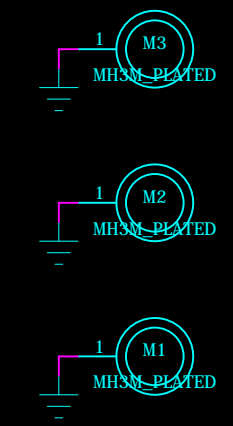
RefDesOk

From ADP1612_1613 Datasheet: Fig 38



RefDesOk

- F1
- FIDUCIAL_1MM
- F2
- FIDUCIAL_1MM
- F3
- FIDUCIAL_1MM
- F4
- FIDUCIAL_1MM
- F5
- FIDUCIAL_1MM
- F6
- FIDUCIAL_1MM



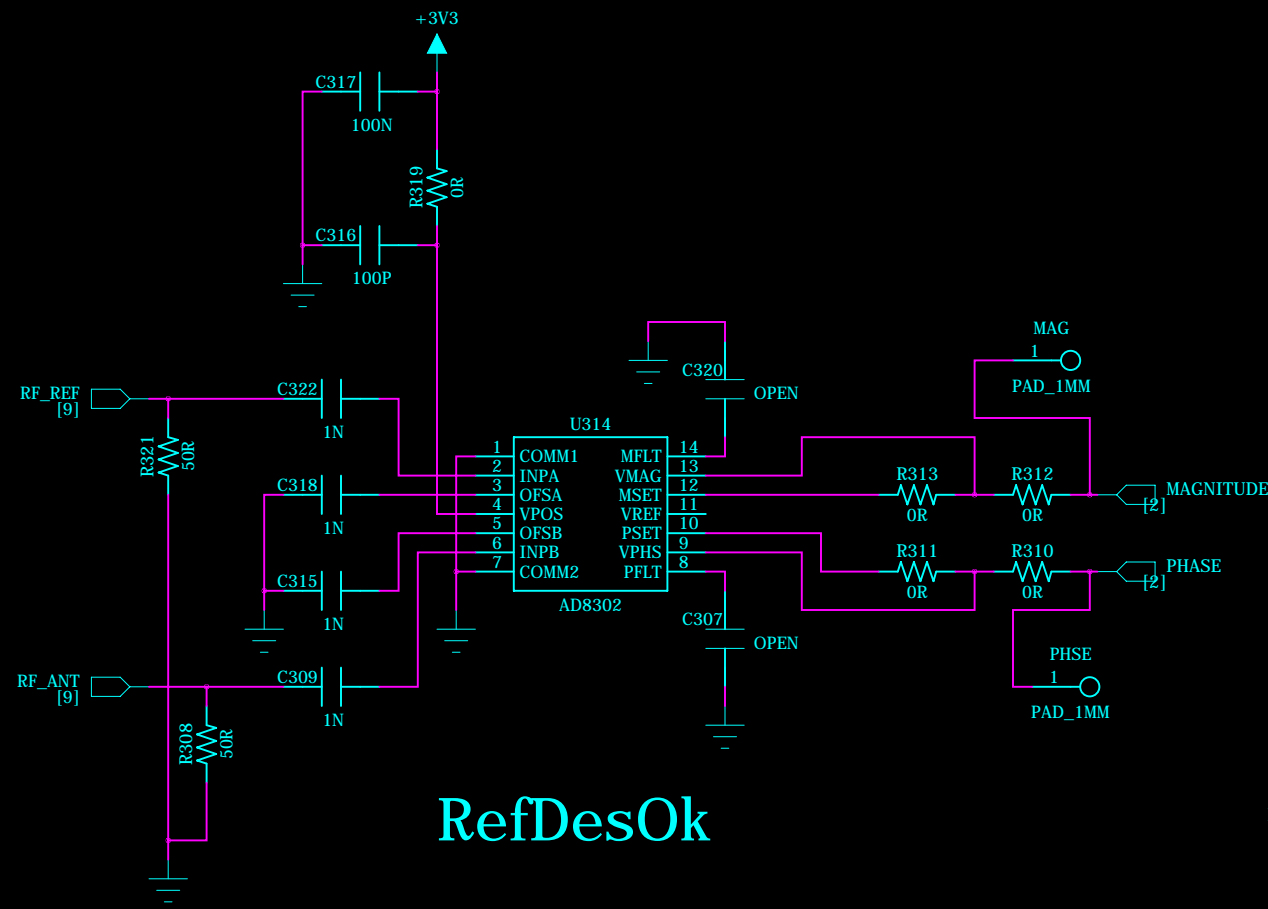
LiPo Charge / Load Sharing

CE,SEL,PROG2, what supply do I want them connected to?

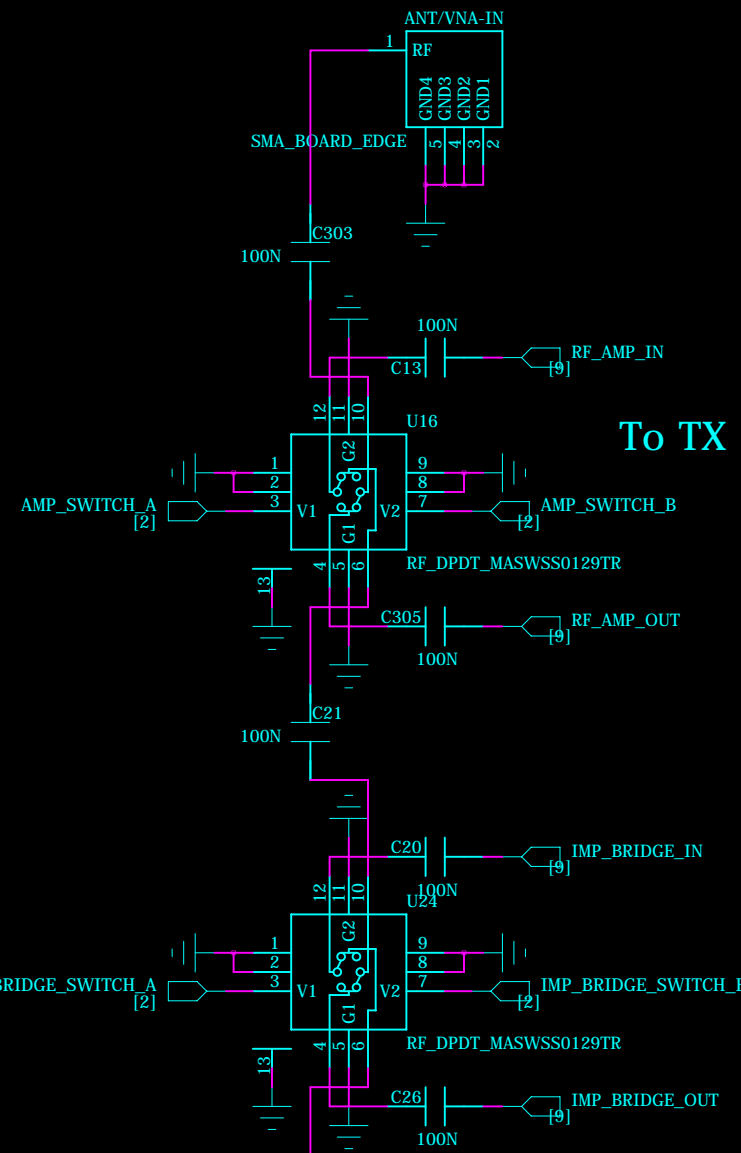
1A Charge is as high as this thing goes :/

Do the values need to be adjusted for this LIPO?

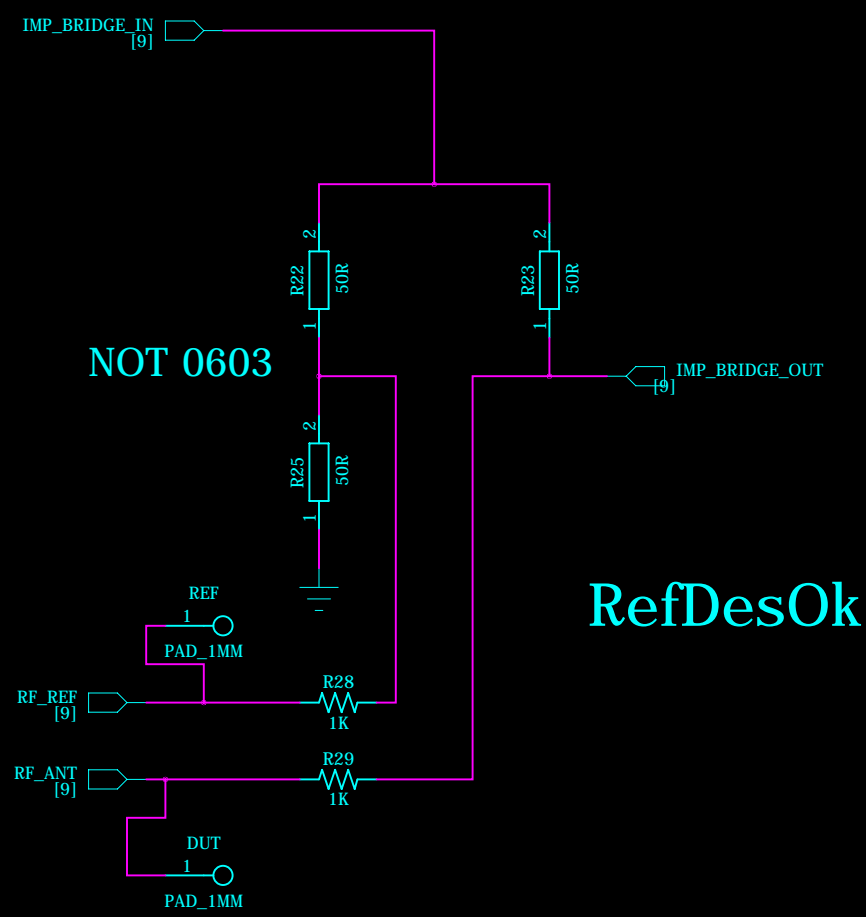
RefDesOk



RefDesOk

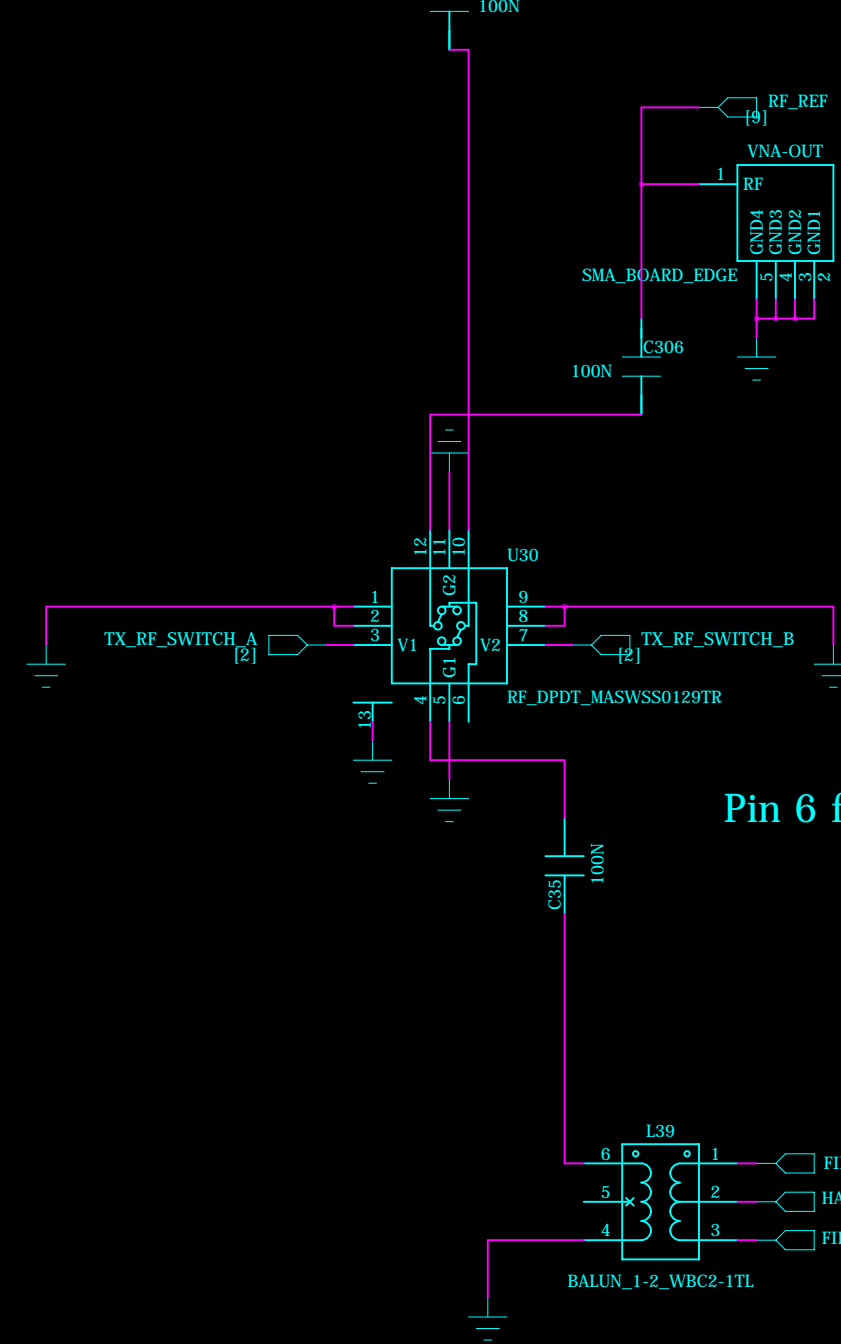


To TX power amp (warning! about 1W max for this switch)



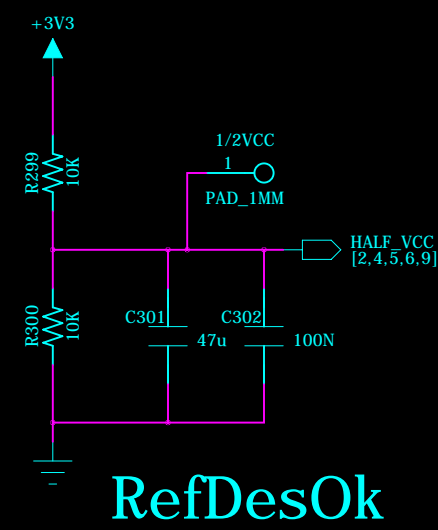
NOT 0603

RefDesOk

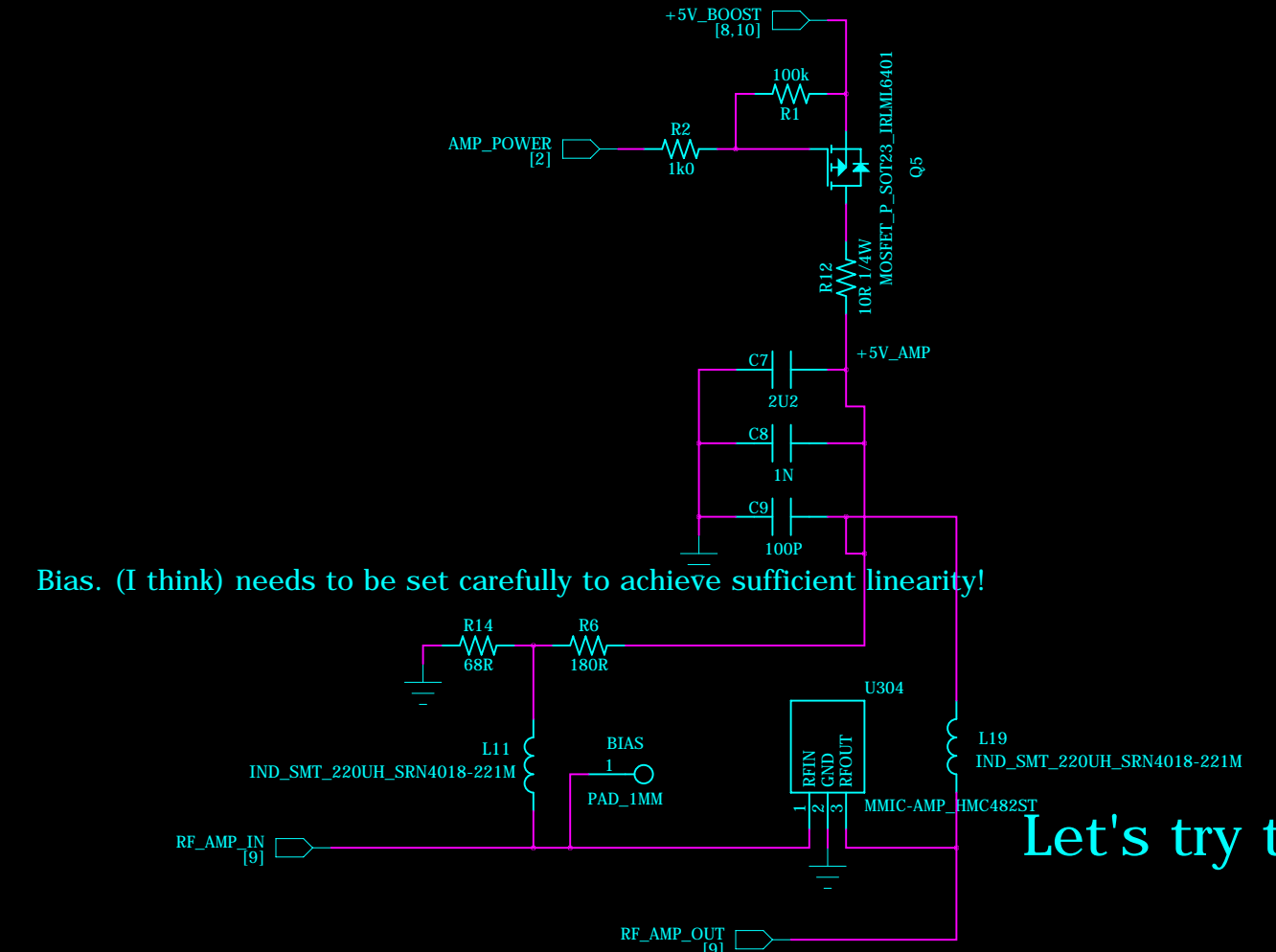


Pin 6 floated so that Impedance Bridge isn't grounded

RefDesOk



RefDesOk



Bias. (I think) needs to be set carefully to achieve sufficient linearity!

Let's try this one

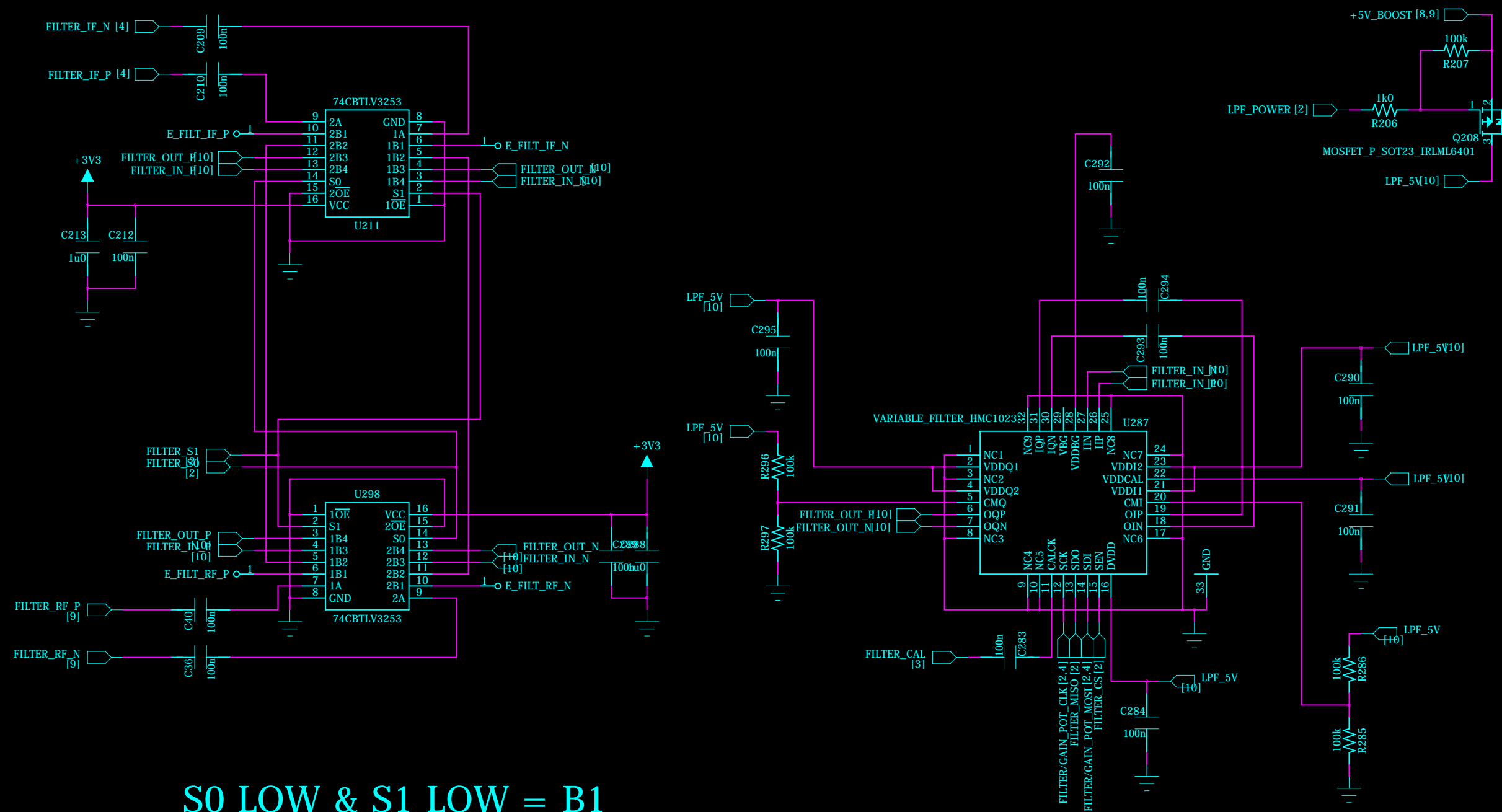
Actually going to use SXB2089Z here instead

RefDesOk

Consider alternatives. This part doesn't have a specified bandwidth.
 The SN74CB3Q3253DBQR is rated upto 500MHz

I think some of these decoupling caps are unnessesary depending on the signal path

Do I need to bias some of these lines?



S0 LOW & S1 LOW = B1
 S0 HIGH & S1 LOW = B2
 S0 LOW & S1 HIGH = B3
 S0 HIGH & S1 HIGH = B4

RefDesOk