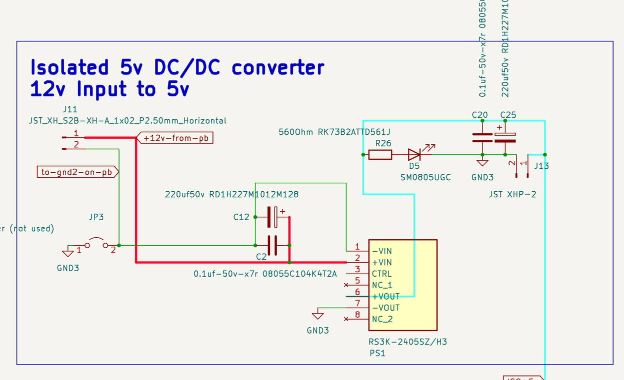
G-EDM

GND1 = ESP GND
GND2 = Pulseboard main GND
GND3 = 5v ISO GND



Important notes and upgrade possibilities.

The two LM358 OpAmps are old tech. Slow and not rail-to-rail.
A possible upgrade would be to replace them with OPA2350UA like the once used on the ESP side.

Old prototype used only OPAs. It seems to work but the single discharge detection is not as reactive as with the initial prototype.

If they are changed against OPA2350UA the current limiting resistors on the HCNR optocouplers need a change too.

R35 & R32 need to be E499 resistors.
The LM358 will swing up to positive rail -1.4v.
Max feedback therefore is 3.6v. The OPA2350UA will go a little above 5v.

Possible that the voltage dividers after the low pass stage (R10&R4 && R28&R30) need a change too for this upgrade. Not sure yet.

R44 connect the isolated 5v GND plane to the pulseboard GND. This is enough to make the analag feedback from the bus work. There is also JP3 that can be used to fully bridge the isolated 5v gnd to the pulseboard gnd with a jumper. It is not used and not needed.

If for some reason JP3 is used then R44 needs to be removed. Only a single path to gnd is allowed or there will be noise issues because of GND loops.

R38 & R39 are EP (experimental)
They normally don't belong there and are by default a zero Ohm resistor. Together with the 100k pulldown next to them they can build a voltage divider that may be useful to enforce a clean 0v output on no-load

R40 & R43 are EP (experimental) and not used
They normally don't belong there and are by default not connected. Together with the current limiting resistor next to them they can build a voltage divider that may be useful to enforce a clean 0v output on no-load

2.5V and 1.65V are only used to terminate unused OpAmp channels

C33, C38, C42, C43 are not used on my board

JP1 connects the PWM return path to the ESP GND plane
By default it is bridged.
If it is not bridged it can be wired to the ESP GND directly.
Was jsut for testing to see if the PWM makes noise on the GND plane. But it doesn't so just bridge it.

C21 & C35 should be tested with lower values.
Old prototype used 47pf. Not sure how they will impact the feedback but maybe a lower value will increase sensitivity for single discharges.

C17 & C30 are unused feedback capacitors.
They have a massive inpact on the feedback.
Even at 100pf the feedback amplitude is reduced heavily.
Feel free to test different caps. Maybe it improves the feedback.
Something below 47pf may work.

Old prototype did not use them and I also don't at the moment.

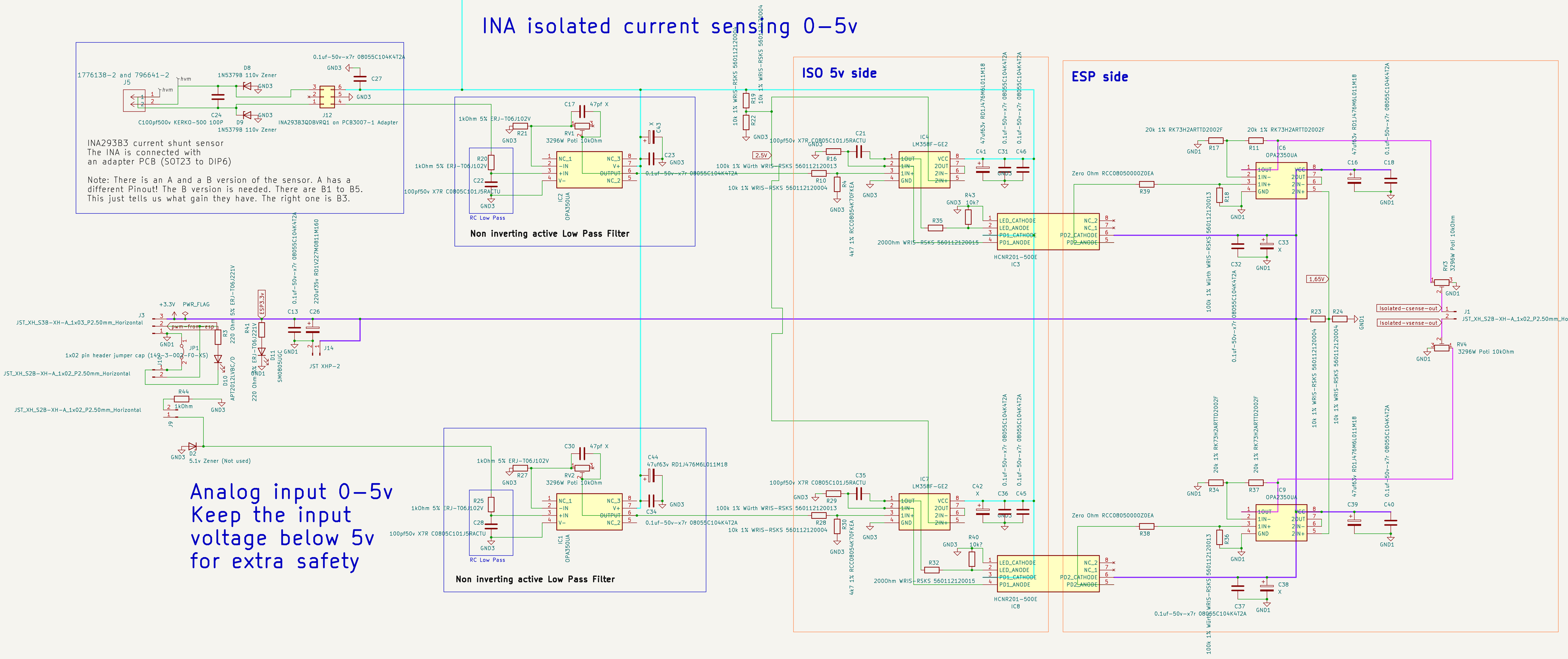
C22 & C28 are part of the RC low pass stage.
With 1k and 100pf the low pass cutoff frequency is something around 1.5MHz.

I did not try it in depth with different values except 1nf. With 1nf it would be a 170khz low pass and this does not work.
Once the DPM switches to maintain current the low pass will mess around.

But maybe a cutoff frequency in between 170khz annd 1.5MHz will work better.

I was to lazy to find out.

Higher capacitor value will reduce the cutoff frequency and lower values will increase it.



Build notes

Add the surface mount SMD components first.

Review soldering

After they are soldered check all OpAmps for solder bridges and remove them if needed.
Adding a little flux makes them easy to remove with a soldering iron.

Test LEDs

Next test all three LEDs with a multimeter in diode mode to ensure they work. All LED current limiting resistors are on the high side.
Put the red probe between resistor and LED and the black to the other side of the LED.
Note: Without jumper the PWM indicator LED is not connected to the GND plane. It is connected to JP1.

Meassure terminal resistance

Then measure the resistance from ISO5+ to ISO GND on J3. It should be 20kOhm. On my boards with 1% resistors it is pretty exact 20kOhm.

Grab the resistance from 3.3v to ESP GND on J3. (Left and right vias, center is PWM)
This should be also around 20kOhm. I measured 19.8kOhm.

Solder Potis and set resistance

Solder all 4 Trimpotis on the board. RV3 and RV4 have the screw poiting to J3 on the right.
RV1 and RV2 have the screw poiting to J9 or the LOGO print.

All 4 potis should have a very low resitance set. This is finetuned after the board is finished but it is good to have a rough setting.

RV3 and RV4 are voltage dividers while RV1 and RV2 are variable resistors.

RV3&4 are used to supress potential noise at 0v feedbacks and produce a clean 0V output.
Turning them clockwise will make the feedback drop down. The basic voltage feedback is inverted and the poti for this channel (RV4) is not critical but for the current sensing we want a hard 0v as no-load condition.

RV4: Connect the multimeter probes to the left pad from J1 and Pin1 from IC9. Turn the poti screw counter clockwise until it reads about 10 Ohms. Doesn't mneed to be perfect and can be 0 Ohm too.

RV3: Connect the probes to the right pad from J1 and PIN1 of IC6. Use the same values.
About 10 Ohms.

Those values are not to critical. On one board I tested 0.1Ohms for the current sensing and 100Ohm on the voltage channel. Both worked. The reistance on the other side is 10k minus the value measured and the voltage drop is minimal.

RV1&RV2 are used as variable feedback resistors to set the gain of the low pass filters.
By default we don't want any gain here. But it may be useful to have some if the machine is only used for low current wire EDM to have a bigger response on low currents.

Also RV2 needs to be almost zero gain if used with the default bus voltage. The bus voltage is 4.8v that drops down depending on the DPM settings. It could use a tiny gain but doesn't matter.

Gain is only interesting if a custom feedback is connected to the voltage channel and for the current sensing on low currents.

Since C30 and C17 are not connected we can use those as probe points.

Turning them counter clockwise will lower the resistance and gain.

RV2:
Touch one pad of C30 with the red probe and the other pad with the black probe and set the resistance to about 10 Ohm again.

RV1:
Touch one padd of C17 with the red probe and the other pad with the black probe and set the resistance to about 10 Ohm again.

Sheet: /	
File: evo-cube-sensor.kicad_sch	
Title:	
Size: A0 Date:	**Rev:**
KiCad E.D.A. kicad 7.0.7+dfsg-1	Id: 1/1