

SERVO DRIVER BLOCKS

The project is divided in blocks. Each block into a group of schemas and
circuits depending on the function. The project is divided in three
interdependent interconnection using double arrows. For that reason
the ac is 100Vrms to ac and 200Vrms and the other type is ac and
200Vrms. Not using to connect that PCB to not 220V to just a name
indicating the first iteration stage.

sigma_delta

SDELTA

sigma_delta.sch

uc_gpio

uC
GPIO

uc_gpio.sch

lem

LEM

lem.sch

ac_in

AC
IN

ac_in.sch

uc_clk_dbg

uC
CLK
Dbg

uc_clk_dbg.sch

temp

TEMP

temp.sch

endat

ENDAT

endat.sch

can_rs485

CAN
RS485

can_485.sch

Vbus_meas

VBUS
MEAS

vbus_meas.sch

uc_power

uC
Power

uc_power.sch

ui

UI

ui.sch

qep

QEP

qep.sch

ethernet

Ether
NET

ethernet.sch

igbt

IGBT

igbt.sch

uc_adc

uC
ADC

uc_adc.sch

connectors

CONN

conn.sch

ethernet

Ether
CAT

ethernet.sch

I choose transformer spare part instead of the embedded shielded RJ45 plus leds, cause now I can choose any format connector RJ45 and maybe add PoE then. And are half the price



dci

Title: ethernet

Title: ethernet

Title: ethernet

Date: 2020-01-09

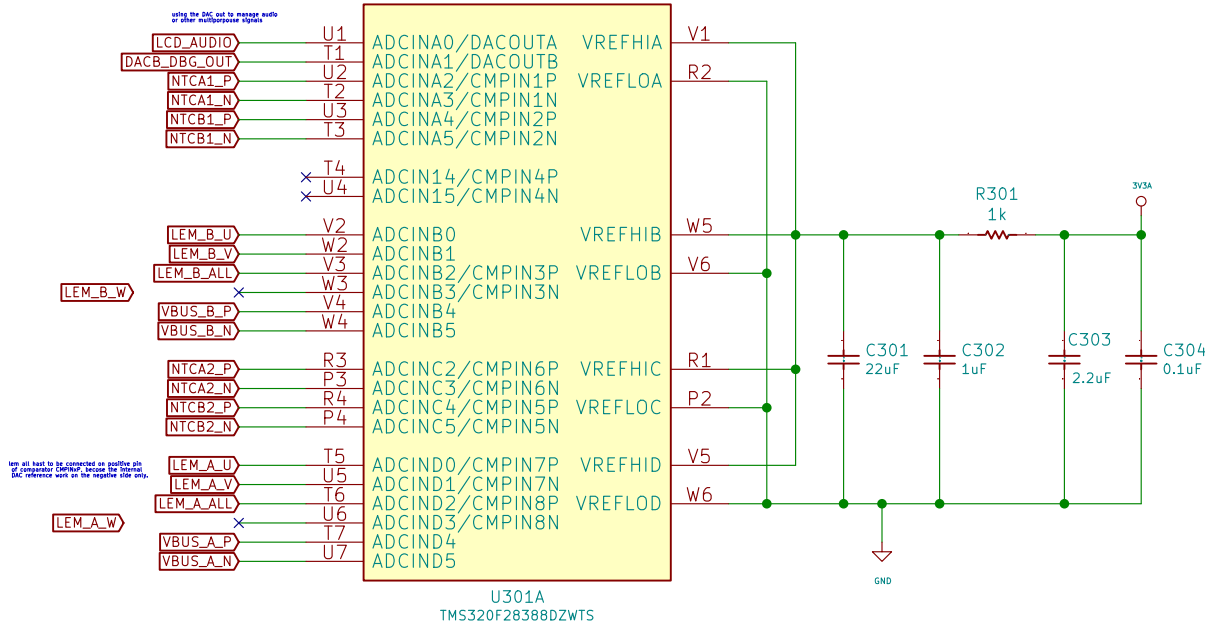
Rev: 1.0

Size: A5	Date: 2020-01
KiCad E.D.A.	kicad 5.0.2+dfsg1-1

Id: 2/18

uC ADC PINS

Every interface to the ADC has the responsibility to filter its output.
Before using ADC pins for that reason these pins just connect to the uC.
Without adding to the noise.



Pablo Slavkin

dci

Sheet: /uc_adc/

File: uc_adc.sch

Title: ADC

Size: A3

Date: 2020-01-09

Rev: 1.0

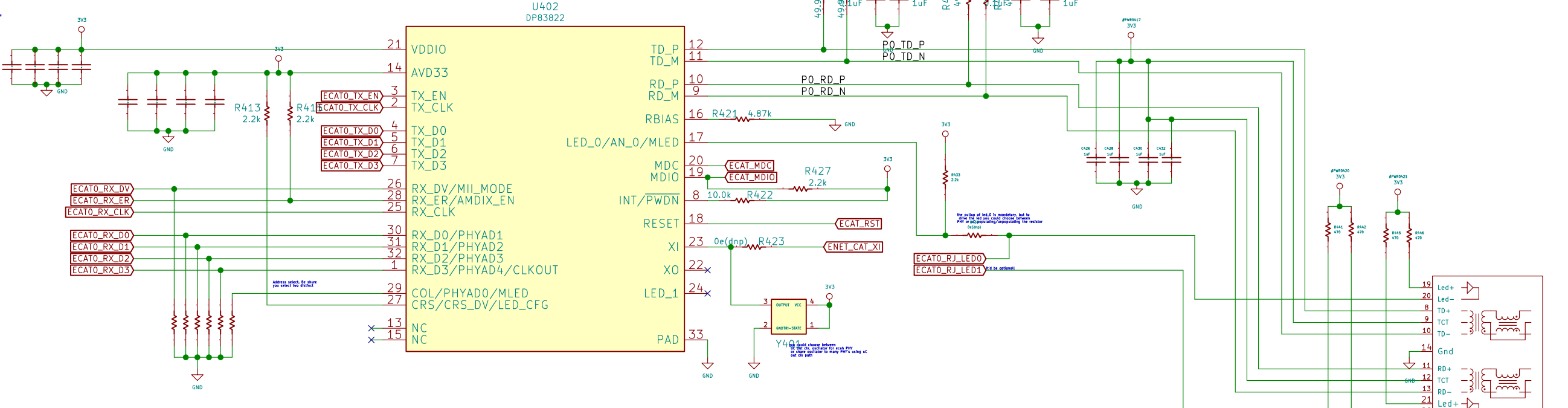
KiCad E.D.A. kicad 5.0.2+dfsg1-1

Id: 3/18

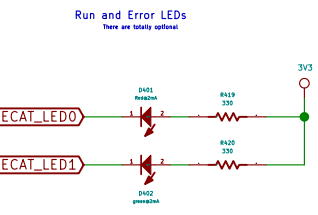
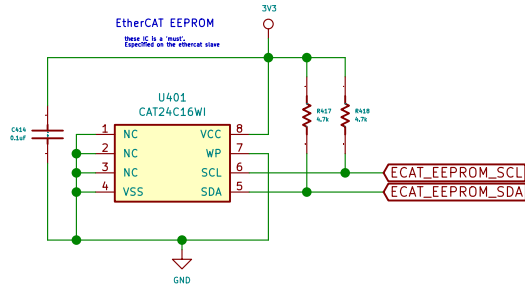
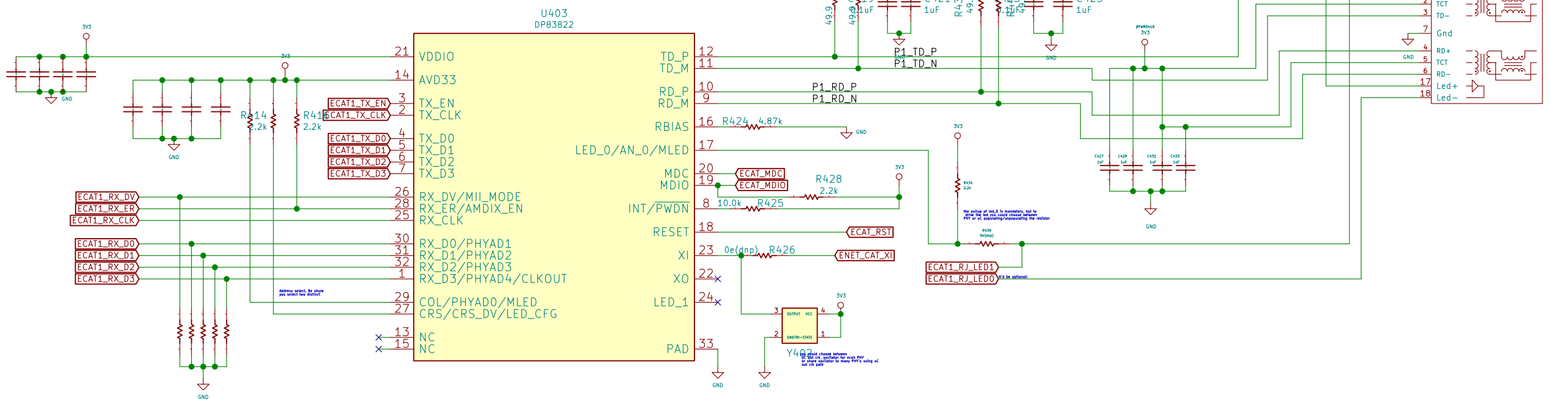
EtherCAT

1 channel transformer spare part instead of the embedded integrated RJ45 ports only, please use 1 pin through any barrel connector RJ45 and make sure RJ45 and are not the RJ45. The diagram is mandatory in order to ESC read save some internal configuration. Lets not opterate, but the PHY's has some pins used as a bootstrap, so you have to respect that.

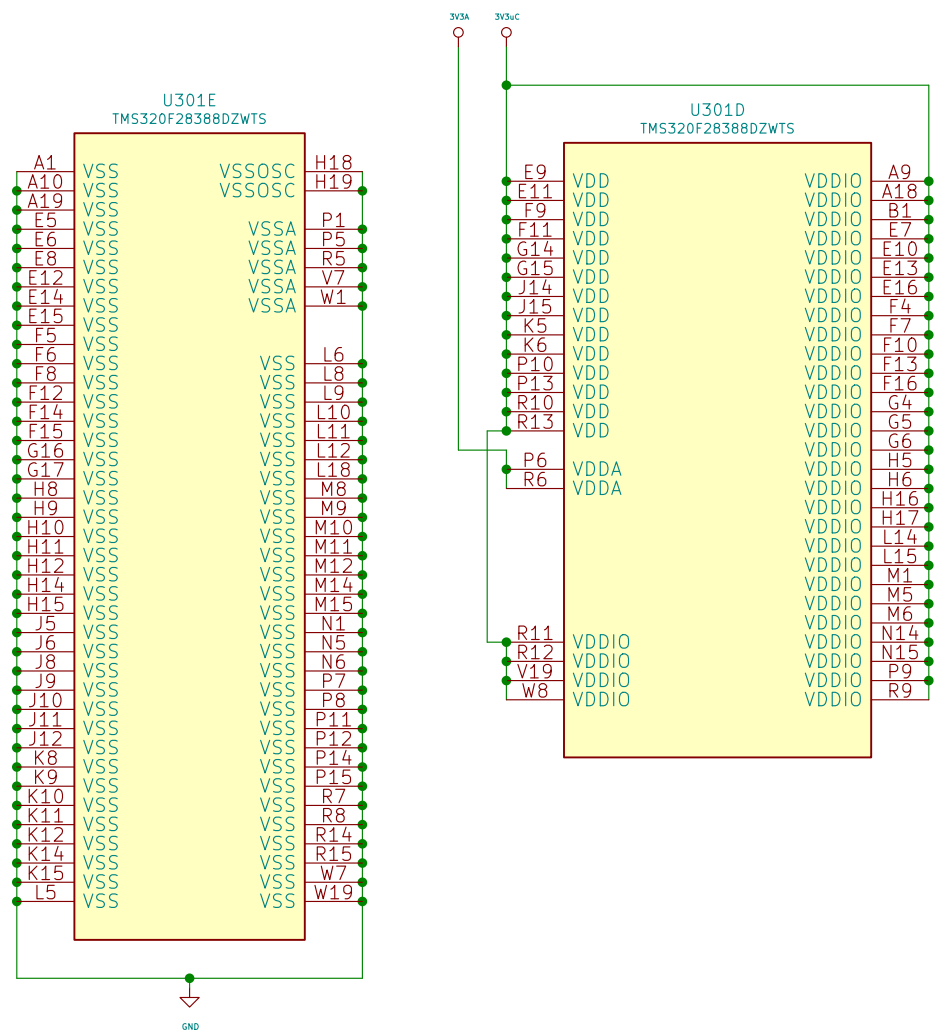
EtherCAT P0



EtherCAT P1

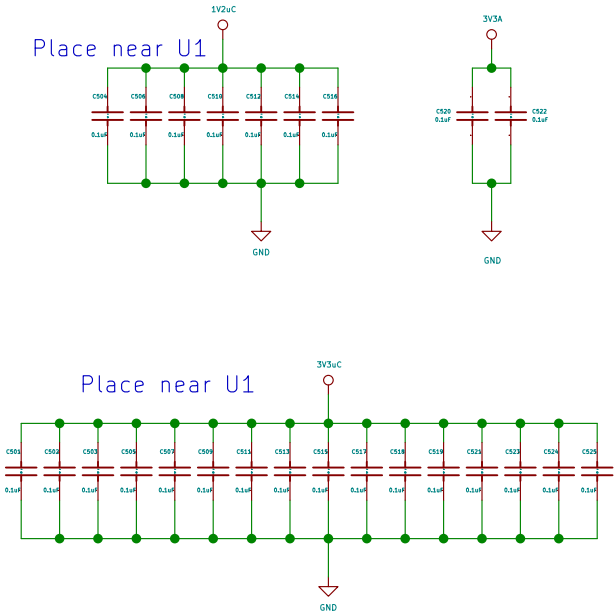


DECOUPLING FILTERS



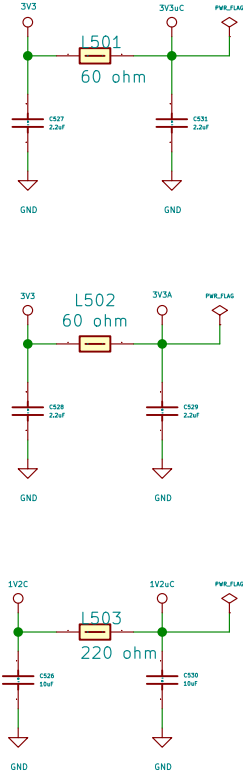
Decoupling Capacitors

Place near U1

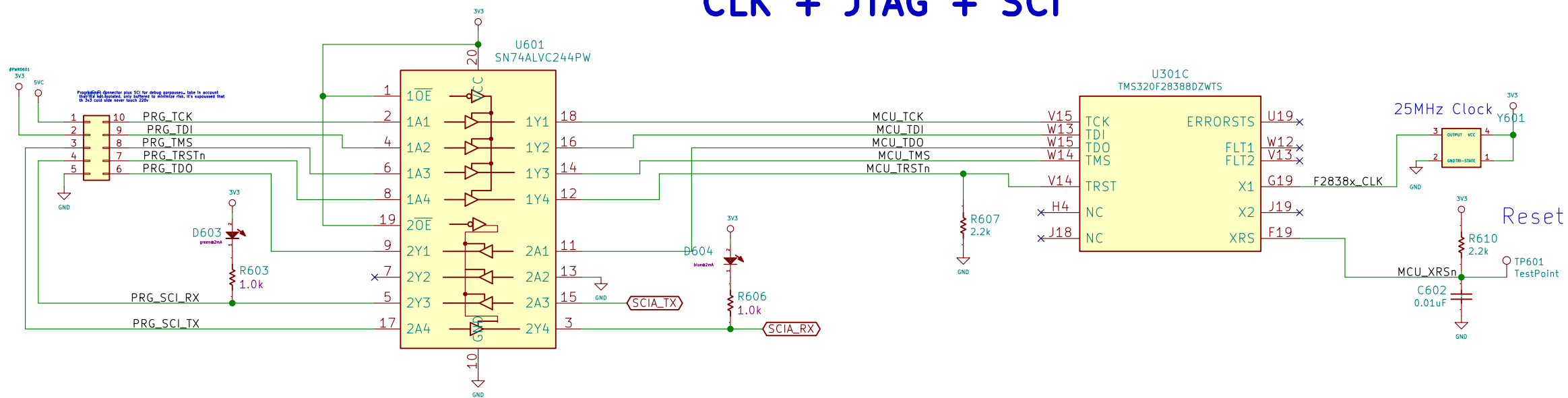


Ferrite Beads

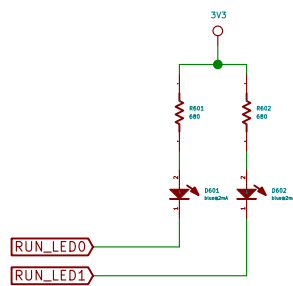
Place near U1



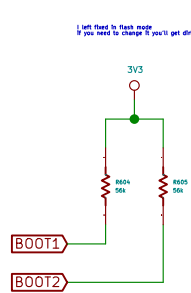
CLK + JTAG + SCI



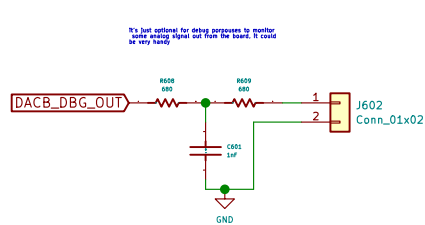
MULTI PORPOUSE LEDS



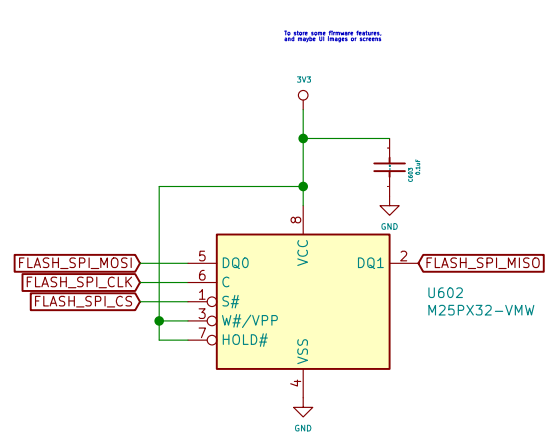
BOOTSRAP R's



DAC DBG OUT



SPI FLASH

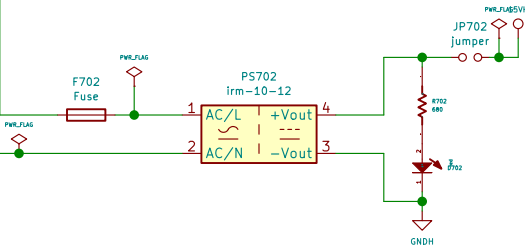
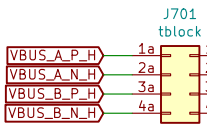
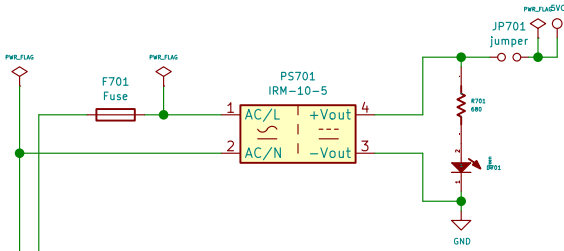


Main Power

In case the control board have to be supplied directly with socket voltage (220V) include these. It's not a good idea cause it has to have right voltage adding these common board, but it's a requirement, so I put it as an option. But you have to be careful with 220V and 240V connectors and

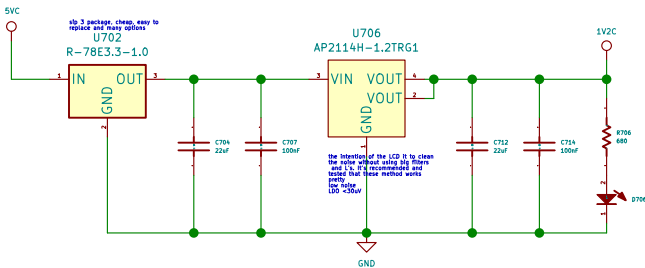
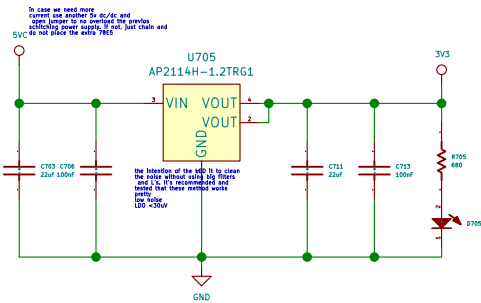
For chosen to add 2 power supply for Cold and Hot sides because it's cheaper than use only one double sided power supply and add a DC/DC isolated converter, and it has the advantage to connect with ease to the AC and to the DC, and the DC/DC can be cold side (more reliable without EMI) or the chosen DC can be cold side (more reliable price to 150uV and to pin-to-pin compatible with 20W, so you could change if you need more power)

I wrote 32V on the DC voltage side, but it's better to use 32V, but the range could be from 8 to 32V more or less



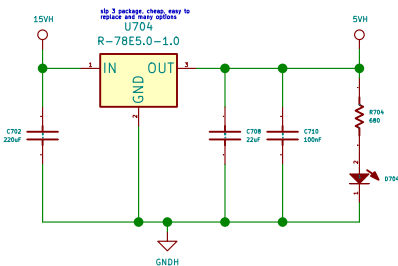
COLD SUPPLY

It is limited to use only 1 input (ONLY inside the control board, none of these outputs will be used for the board, I include every single pin from these supply to be careful, but these is correct)



HOT SUPPLY

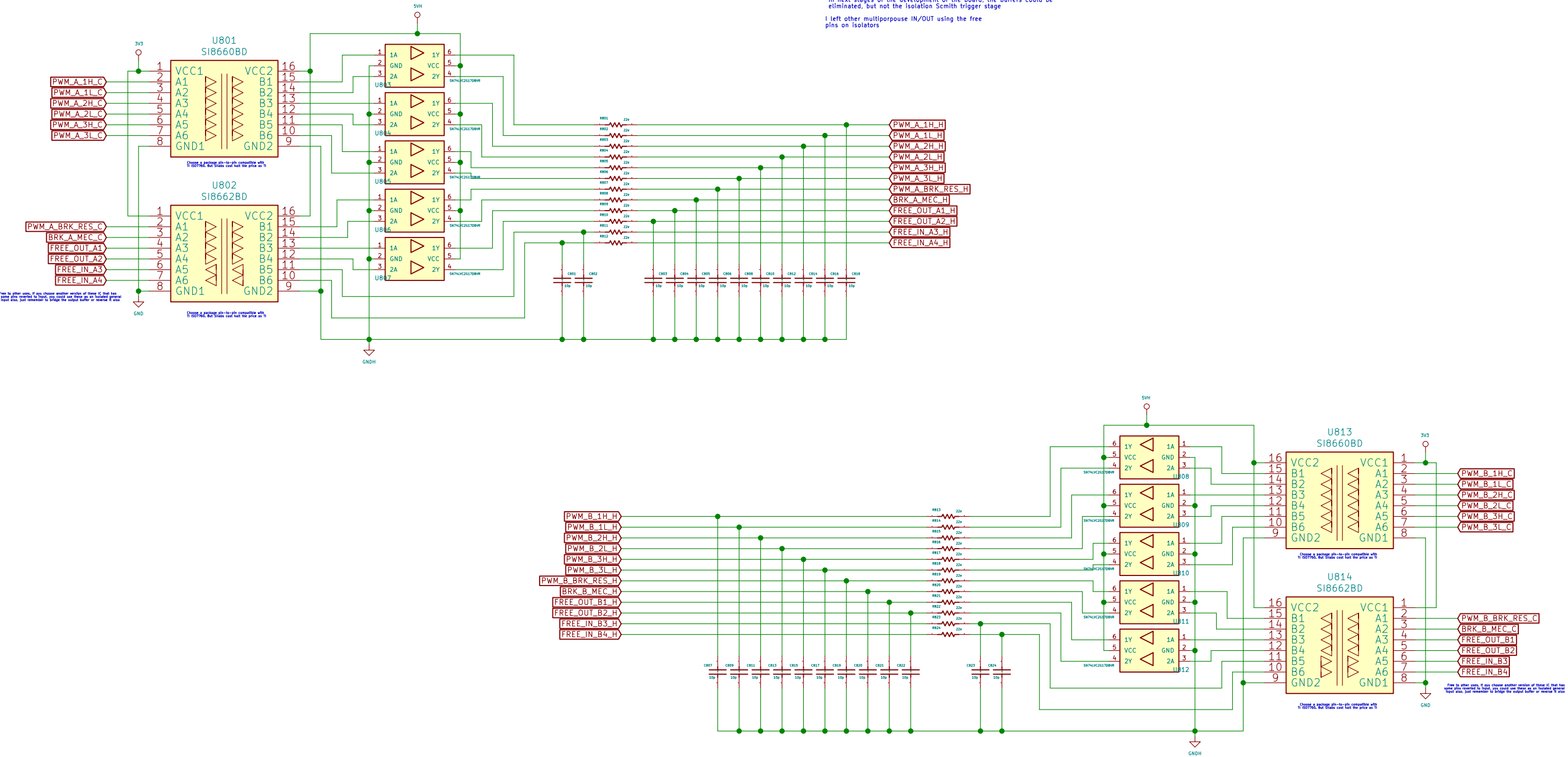
It is important that hot supply, more than it is, is added to high voltage (220V) and that the 220V voltage is connected to ground, so the hot side and ground (which is to make the reference location) is the location where the single reference side



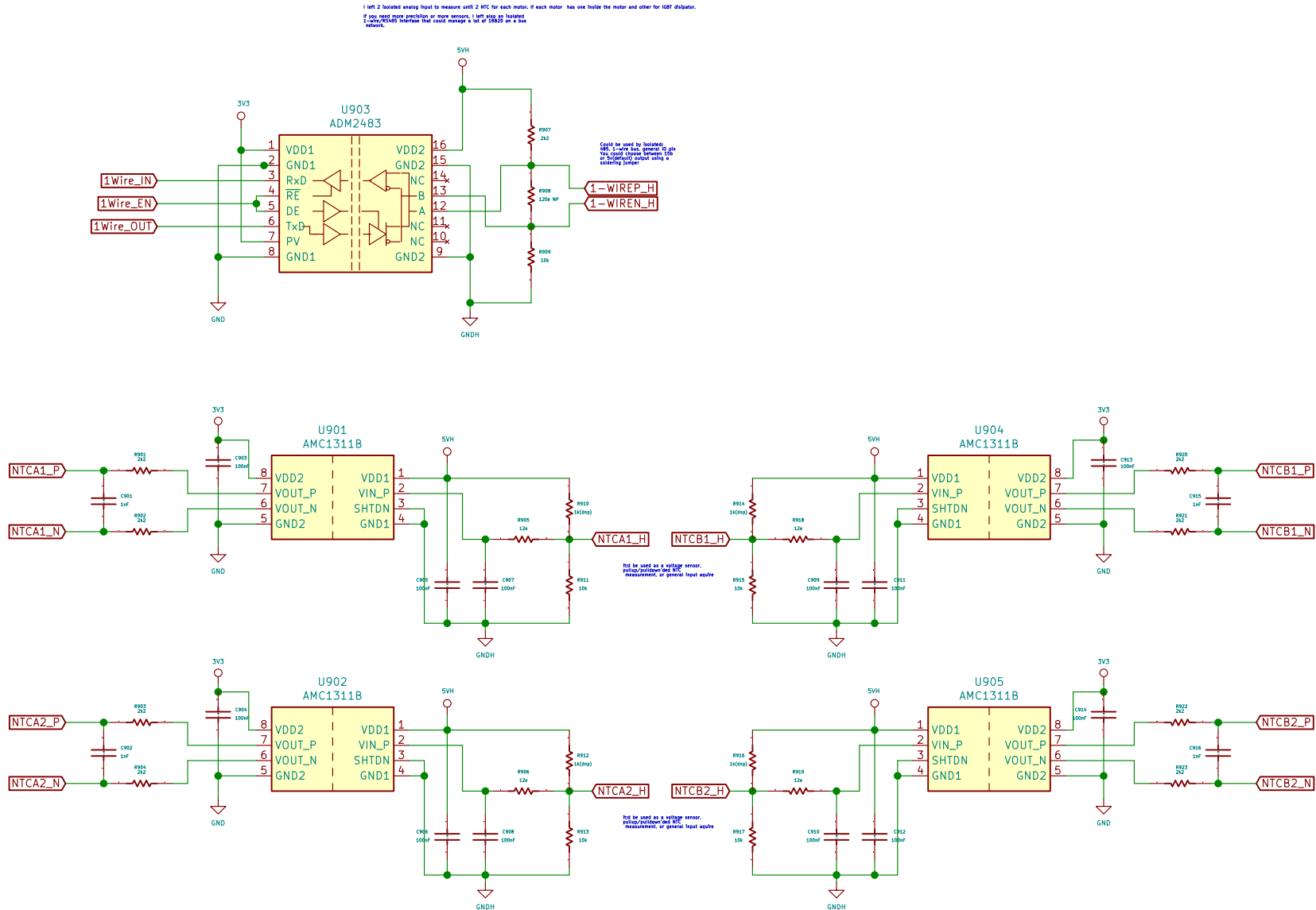
PWM OUT -> ISOLATOR -> BUFFER -> FILTER

these pins has isolation and a buffer. They could be directly connected to a IGBT module for test porpouses or to a power board and another isolation stage for production porpouses to have a reinforced isolation. In next stages of the development of the board, the buffers could be eliminated, but not the isolation Scmith trigger stage

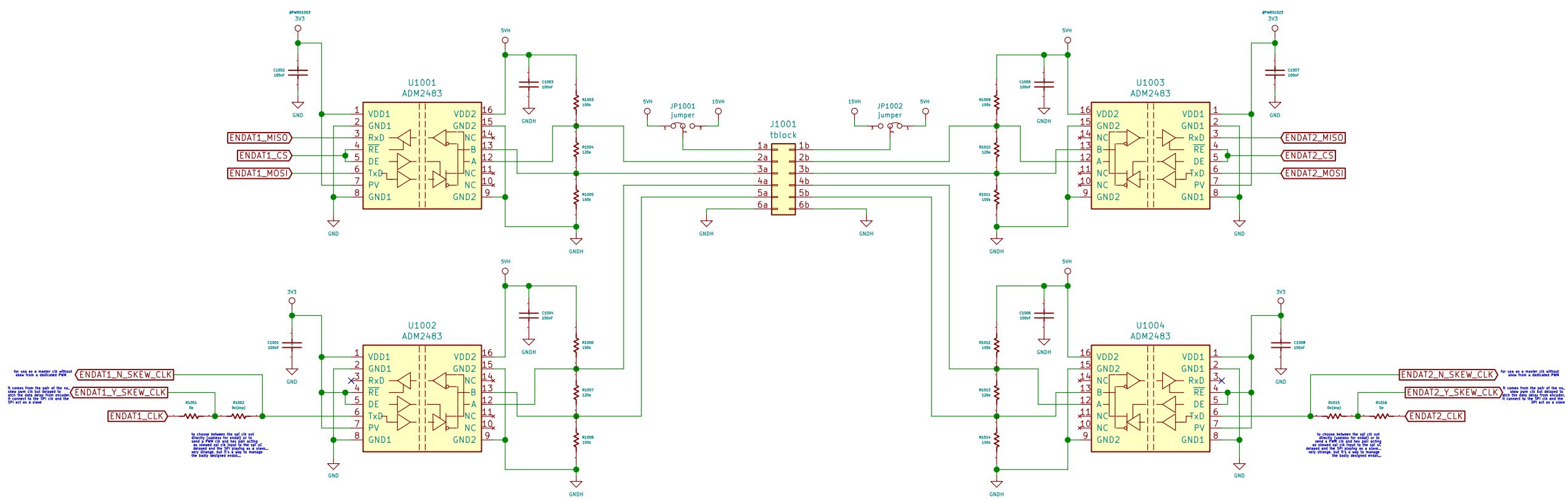
I left other multipurpose IN/OUT using the free pins on isolators



2 isolated NTC interfase + 1 isolated 1-wire/485

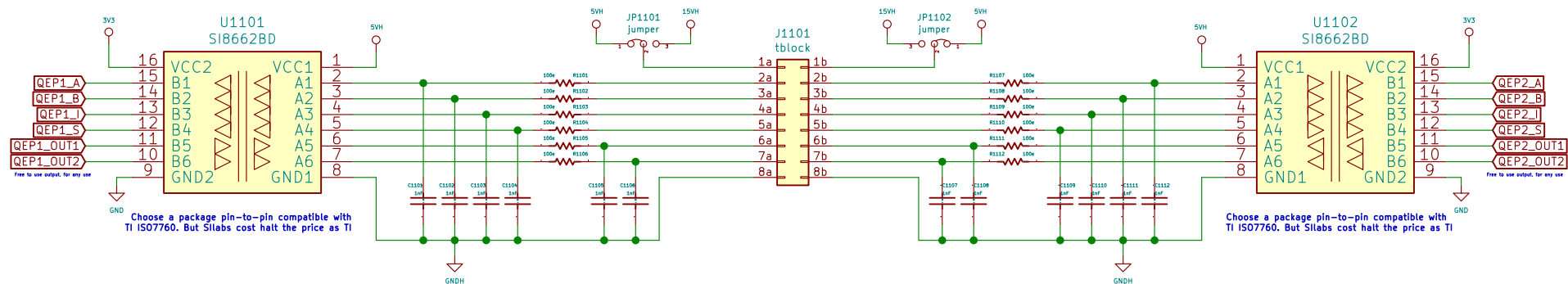


2X Isolated ENDAT interface



2x Isolated Incremental encoder 5v input A-B-I-S

I left the input for two isolated incremental encoders.
From 5V to GND. I think I will give the output for any purpose plus
the ability to choose between 0 or 1VH



Pablo Slavkin
dci

Sheet: /qep/
File: qep.sch

Title: QEP encoder Interface

Size: A4 Date: 2020-01-09

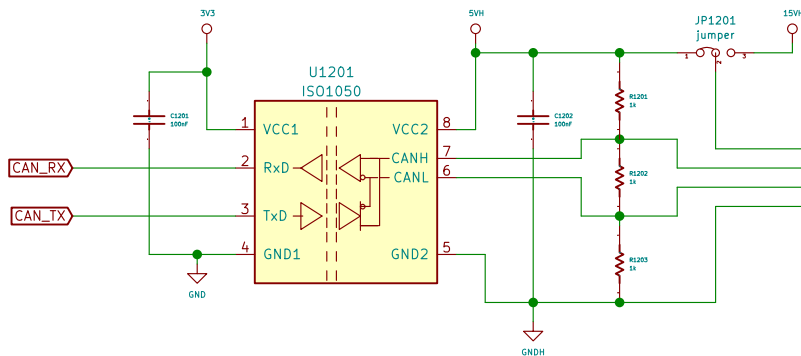
KiCad E.D.A. kicad 5.0.2+dfsg1-1

Rev: 1.0

Id: 11/18

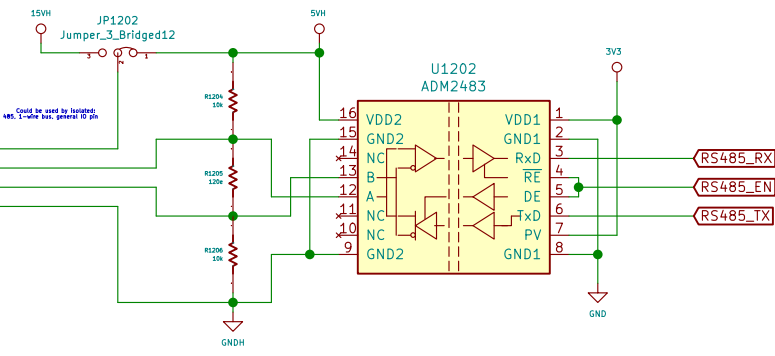
Isolated CAN interfase

A digital CAN device, if used, should be used as a RS485 device in its place. The minimum for CAN is 120ohm, 5k in RS485 or 250k could be fine



Isolated RS485

With these jumper you can't manage isolated RS485 or isolated CAN or isolated L-Net, using 5V or 3.3V as a supply



Pablo Slavkin

dci

Sheet: /can rs485/

File: can_485.sch

Title:

Size: A3 Date: 2020-01-09

Rev: 1.0

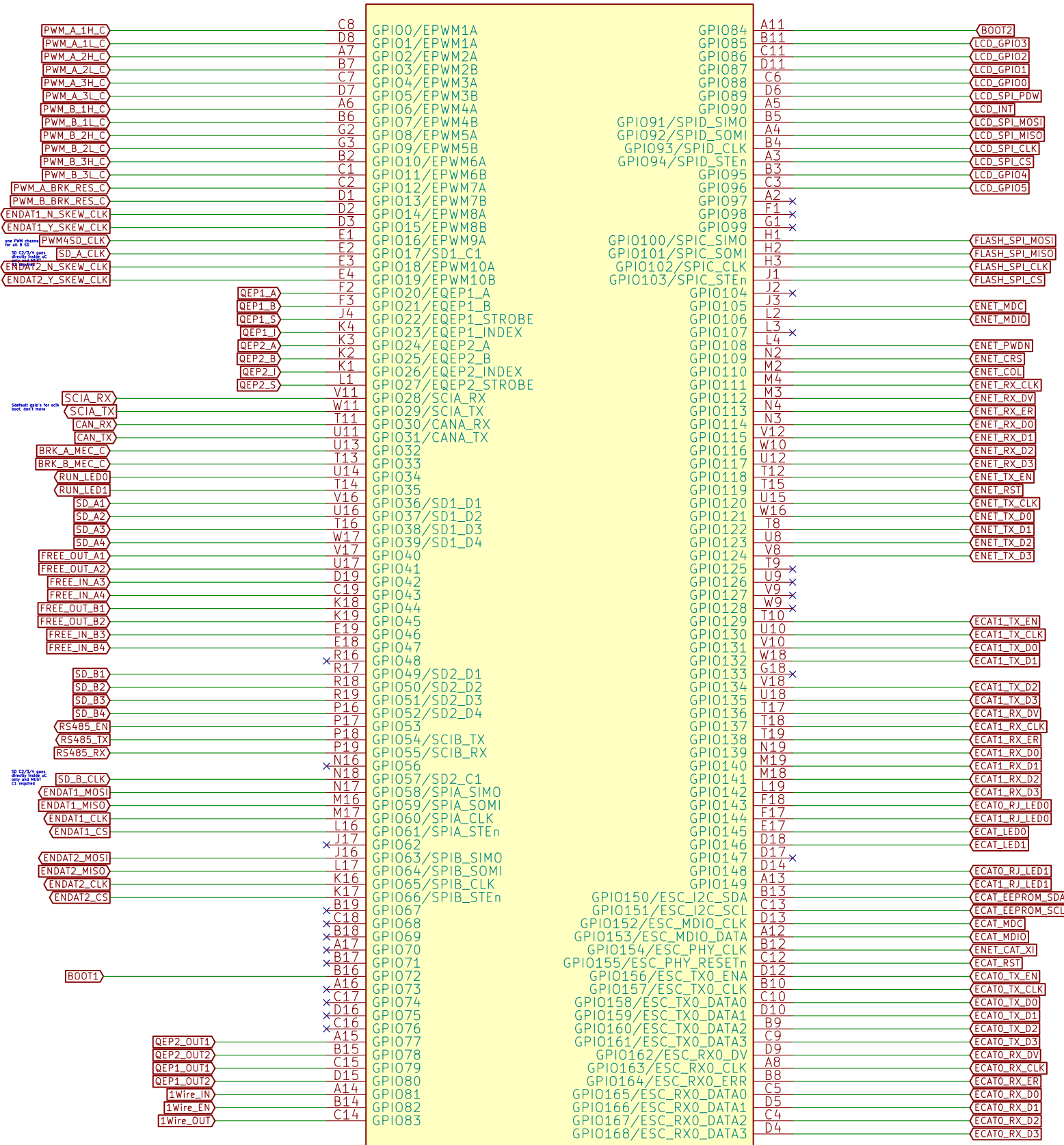
KiCad E.D.A. kicad 5.0.2+dfsg1-1

Id: 12/18

uC GPIO's pins

I've spent hours to choose the GPIO's for each interface trying to not
repeat one to be able to say "this-one is for sensor, that one
is for..."
I've used digital labels connector to get from one page to another
instead of 20 pin connector because it's more flexible to
change, I know it's not too efficient, but it's better and faster for now

U301B
TMS320F28388DZWT5



Pablo Slavkin

dci

Sheet: /uc_gpio/

File: uc_gpio.sch

Title: gpio

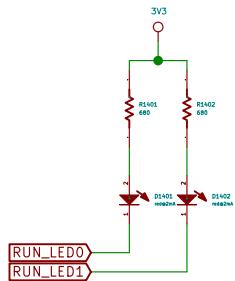
Size: A3 Date: 2020-01-09

Rev: 1.0

KiCad E.D.A. kicad 5.0.2+dfsg1-1

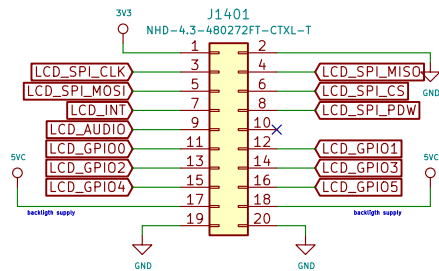
Id: 13/18

Multipropouse LEDs

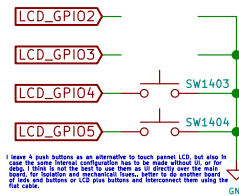
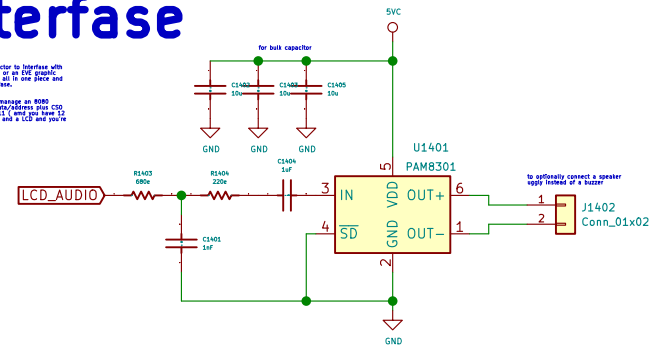


LCD UI interfase

I love a multi LCD/Touch/button 20 pins IDC connector to Interface with some UI. It could be simple, i.e., a character LCD or an EVE graphic LCD as I recommend 'cause for same price you've all in one place and it's already done, clean, usefull and intuitive Interface.



interface directly using f20 wires flat cable
for NHD-4.3-480272FT-CTXL-T newhaven LCD
or hand wiring
EA eDIP128B-6LWTP
or using any parallel 8080 3v3 interface bitbanging
the SPI and GPIO pins



I leave 4 push buttons as an alternative to touch panel LCD, but also in case the some internal configuration has to be made without UI, or for debug, I think is not the best to use them as UI directly over the main board, for isolation and mechanical issues.. better to do another board of leds and buttons or LCD plus buttons and interconnect them using the flat cable.

Pablo Slavkin
dci

Sheet: /ui/
File: ui.sch

Title: clk

Size: A4	Date: 2020-01-09
----------	------------------

KiCad E.D.A. kicad 5.0.2+dfsg1-1

Rev: 1.0

Id: 14/18

8 LEM's current measurement

$U+V+W+ALL \times 2$

The intention of these terms is to have the capacity on the control board to measure all the current without the need of the Vimp driver. This is a power board's idea. Why? Because I've study the bus topology. I've seen the problem that the bus wire is too long and the bus wire is too thin. The problem is that the bus wire is too long and the bus wire is too thin.

with 6 screw connector you could choose
3 range of current measurement 1x, 2x or 3x.

IN 1 BRIDGE 1-2-3	and 4-5-6	OUT 4	-----	X
IN 1 BRIDGE 1-2	and 3-5-6	OUT 4	-----	2X
IN 1 BRIDGE 2-6	and 3-5	OUT 4	-----	3X

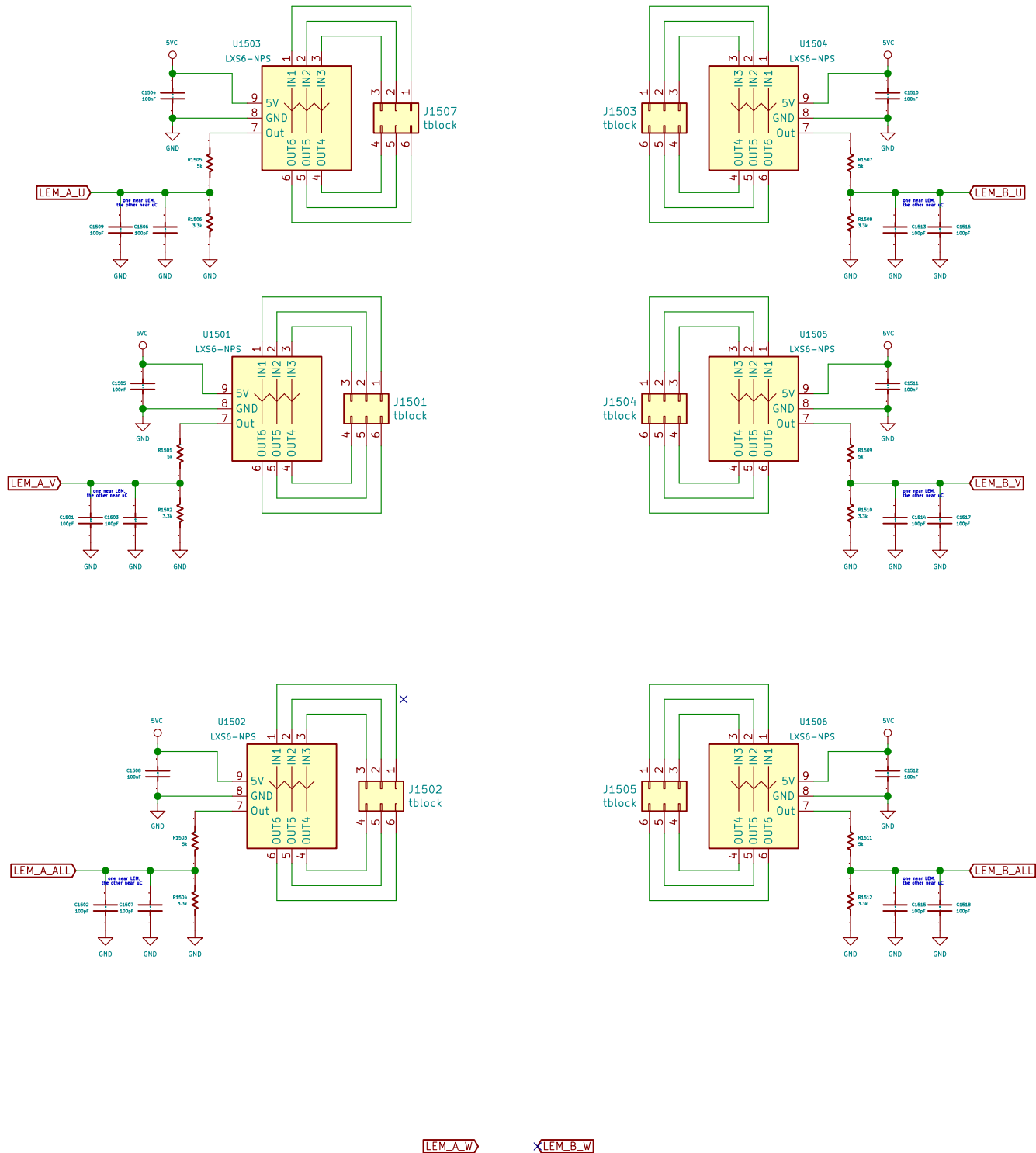
[illegible]

these LEN's version doesn't need a buffer
it's use already included inside LEN

LX56 is LTS and CAS pin to pin compatible, but LX5 is a little better.
I don't use an opamp becous with the external conection I have x2a
and 3y amplifier action without any component

I've decided to make the loop over the LEM on board and if you wanna change it you have to use the soldering iron...because the connector is 2 terminal, not 6

chanlog is I've decided to return to a 2x3 terminal block output 'cause it match the size of the LEM



I've decided to eliminate 1 LEM, you could use 3 in line measurement, or 2 in line plus one for all. The reason is size of board and complexity.

I've decided to eliminate 1 LEM, you could use 3 in line measurement, or 2 in line plus one for all. The reason is size of board and complexity.

Pablo Slavkin

dci

Sheet: /lem/

File: lem.sch

Title: LEM currente measurement

Size: A3	Date: 2020-01-09
----------	------------------

KiCad E.D.A.	kicad 5.0.2+dfsg1-1
--------------	---------------------

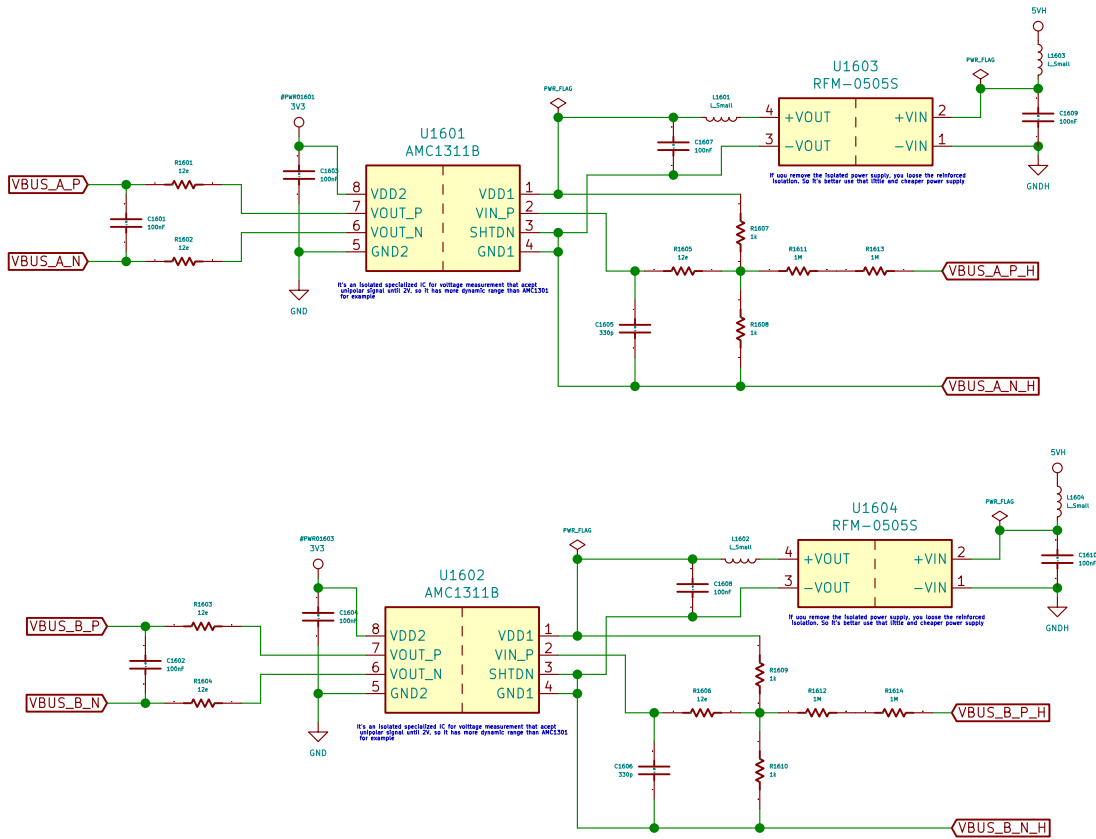
Rev: 1.0

id: 15/18

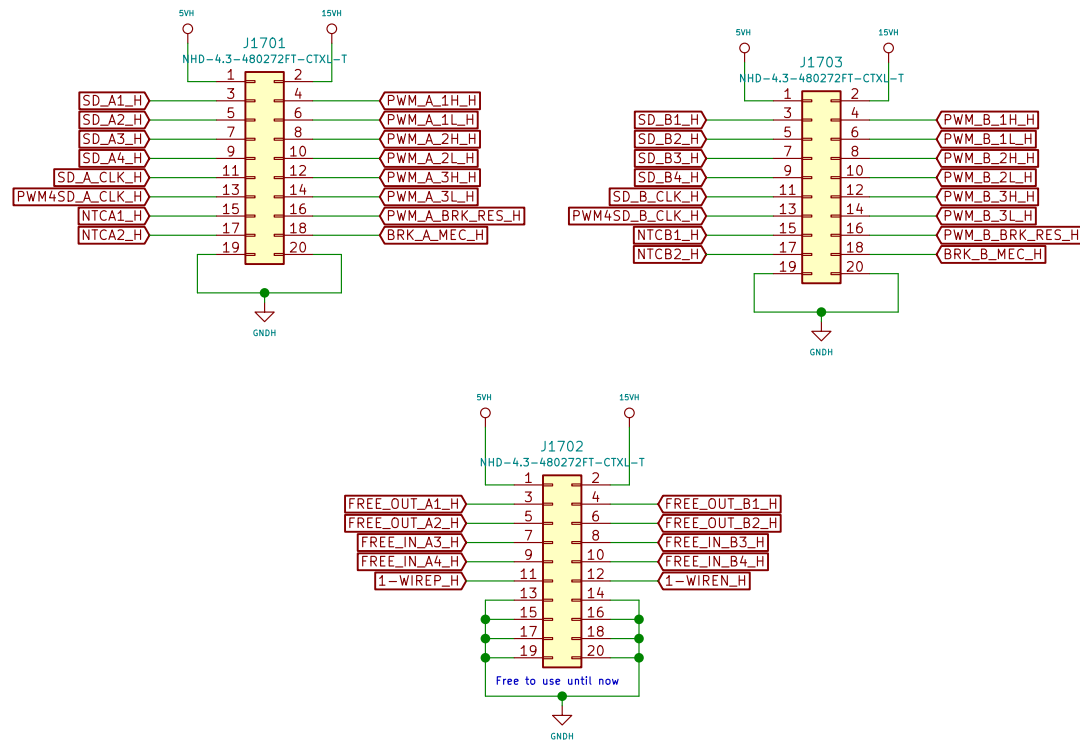
VBUS -> R divider -> ISO ADC -> uC

It's intended to measure the Vbus, one per motor, but they could be joined if both motor share same VBus. The Vbus info will be used by the control algorithm and to drive the brake resistor PWM to protect the rise of the Vbus more than a threshold

The Input is expected not to be 220v or 380v. It's supposed to be a isolated low voltage to maintain the reinforced insulation. In case you can't you could input 220v directly but the isolation would be simple



Common Connections



It's intended for power board circuitry to supply all the signals to control board. The control board will use the SD filter to acquire the data. All the power part will be at power board.

PWM clk output to powerboard and then come back again to maintain delays with respect to SD data channels. On power board you have to respect traces length between clk and data

I use 5VH on power side to be less prone to noise error on the way or connection the two boards (and I've decided to not supply 3x3m hio)

I've added small filter to input/output lines to minimize noise



Size: A5	Date: 2020-01
KiCad E.D.A.	kicad 5.0.2+dfsg1-1