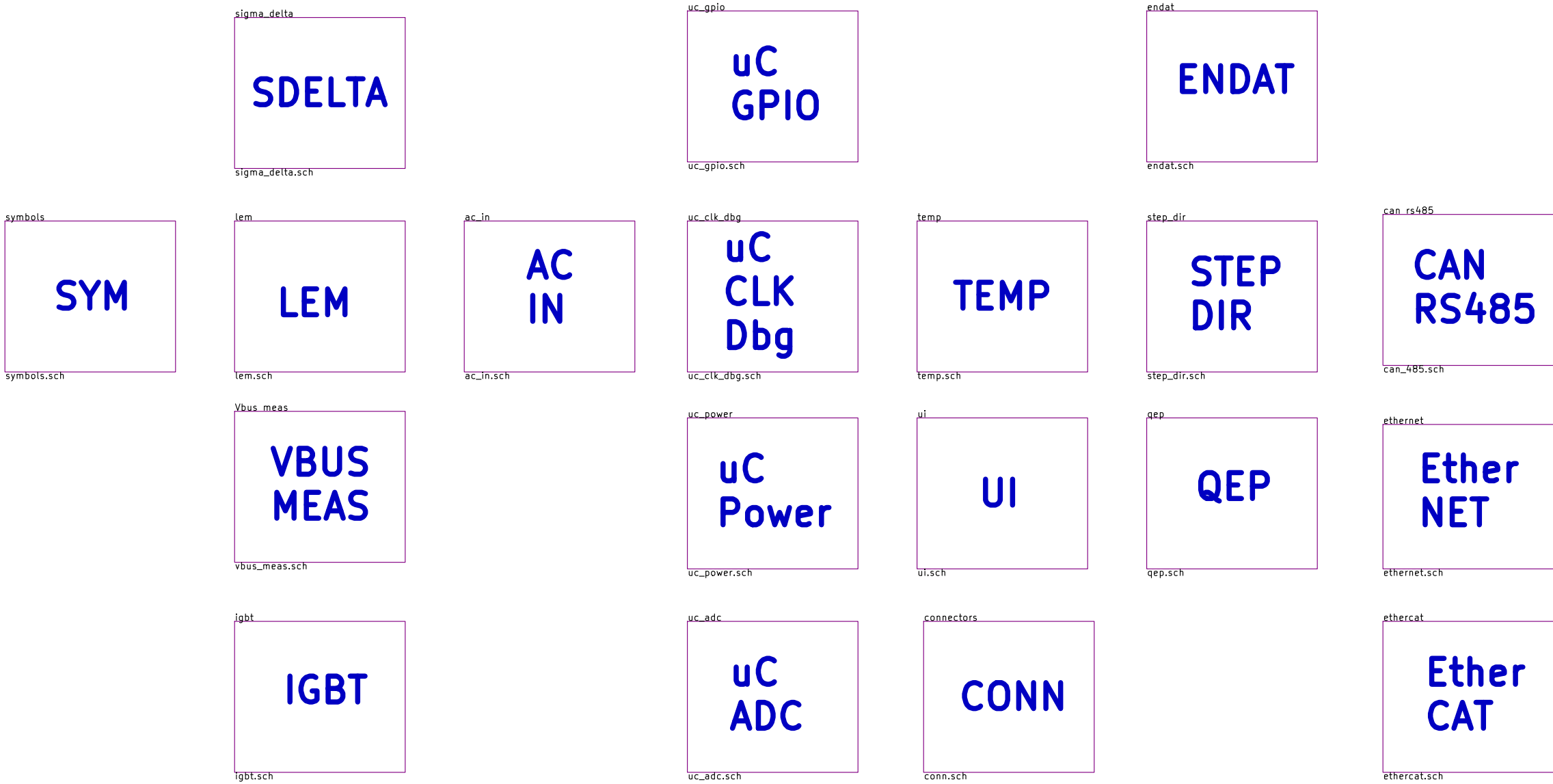


SERVO DRIVER BLOCKS

The project is divided in blocks. Each block into a group of signals and functions depending on the function. The project is divided in three subprojects: hardware, software and documentation. For each subproject, the user can find the files in the project. The user can find the files in the project. The user can find the files in the project. The user can find the files in the project.



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



It's just an USB protection



dc

Sheet: /ethernet/
File: ethernet.sch

Title: ethernet

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

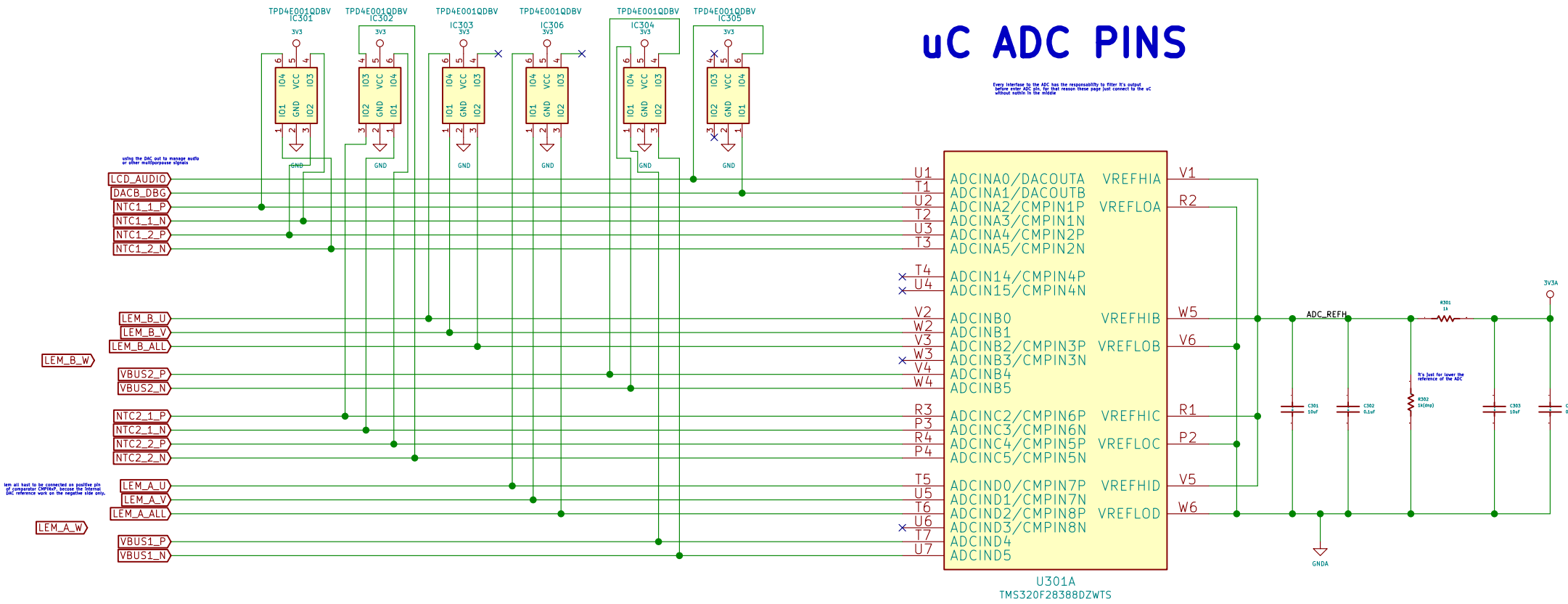
Source: KiCad E.D.A.	Version: 5.0.2+dfsg1-1
----------------------	------------------------

Rev: 1.0

Rev: 1.0
Id: 2/20

uC ADC PINS

From: Interfacing to the ADC has the responsibility to filter it's output.
Before using ADC, the user must ensure these pins are connected to the uC
without noise in the signal.



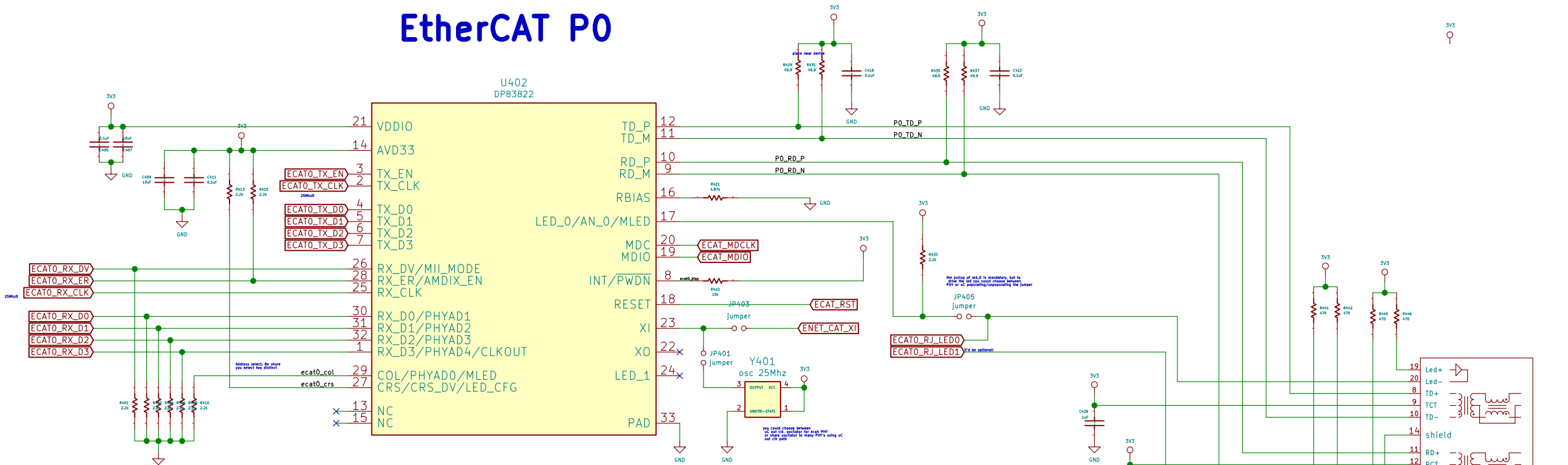
EtherCAT

I chose transformer spare part instead of the embedded shielded RJ45 pins only, as you may I can choose any format connector: RJ45 and maybe also RJ-25, but we will use RJ45.

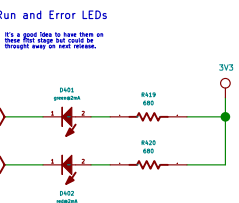
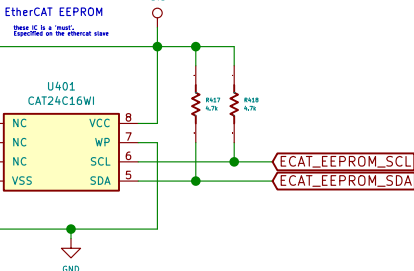
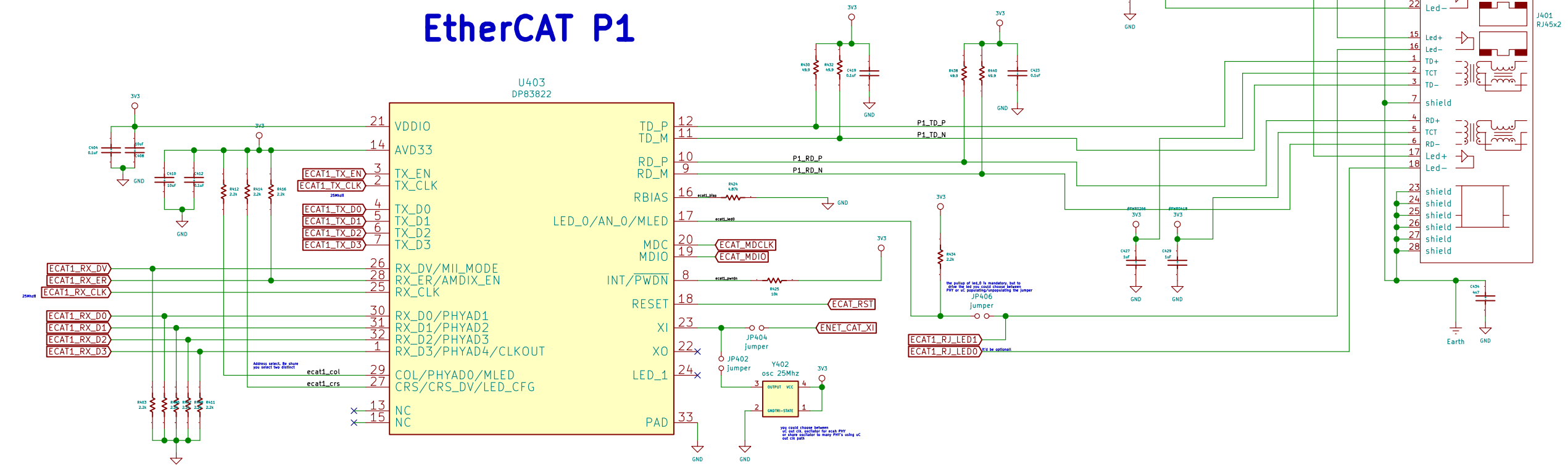
The diagram is mandatory in order to ESD read save some internal configuration.

Let's not forget, but the PHY's has some pins used as a bootstrap, so you have to respect that.

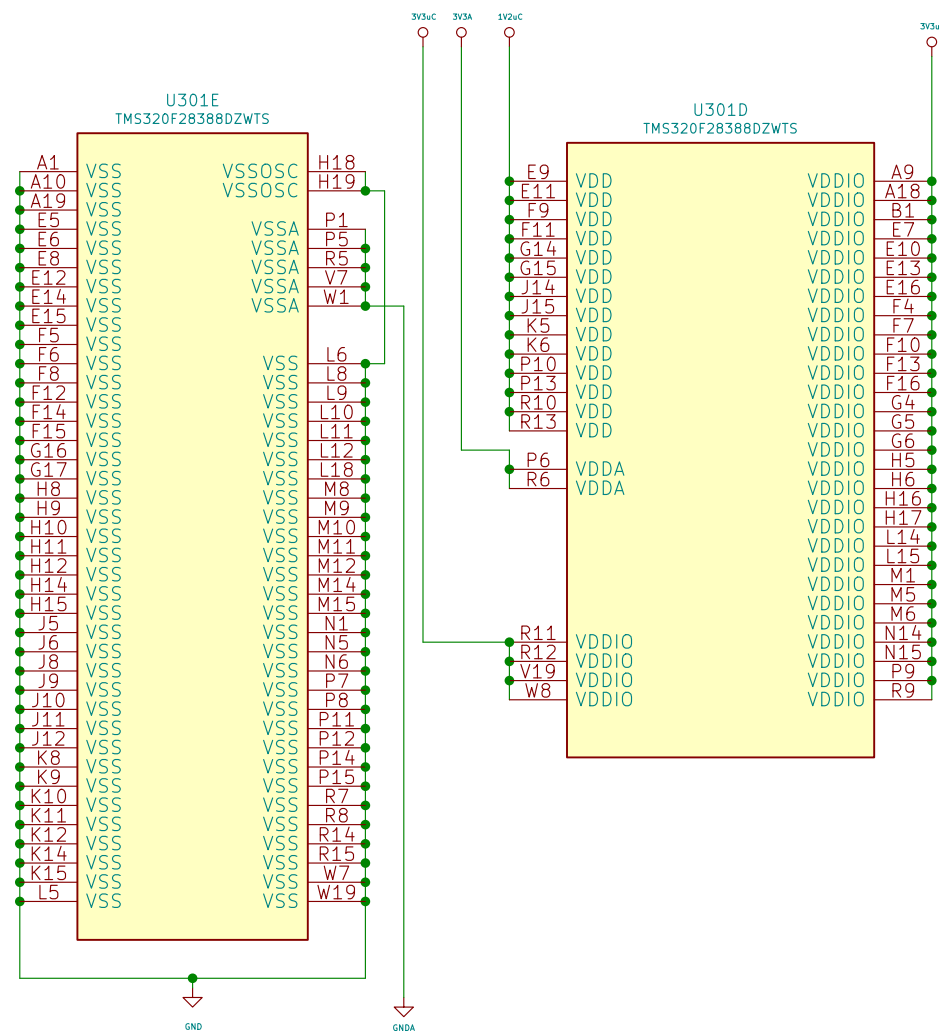
EtherCAT P0



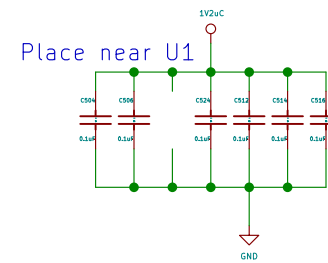
EtherCAT P1



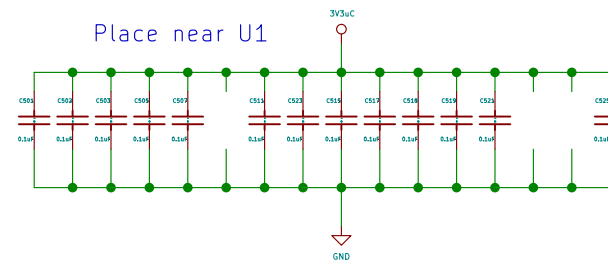
DECOUPLING FILTERS



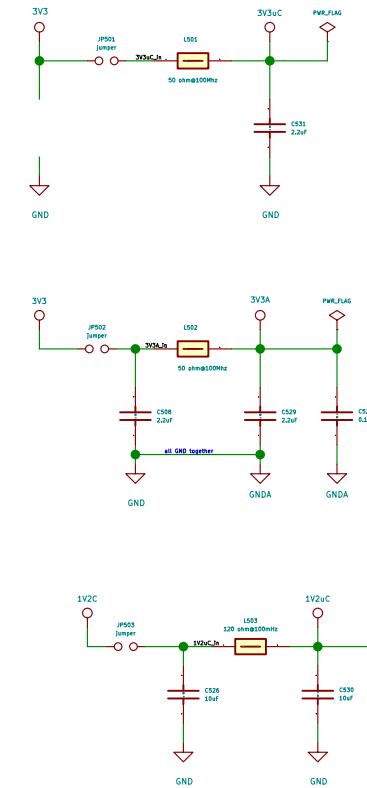
Decoupling Capacitors



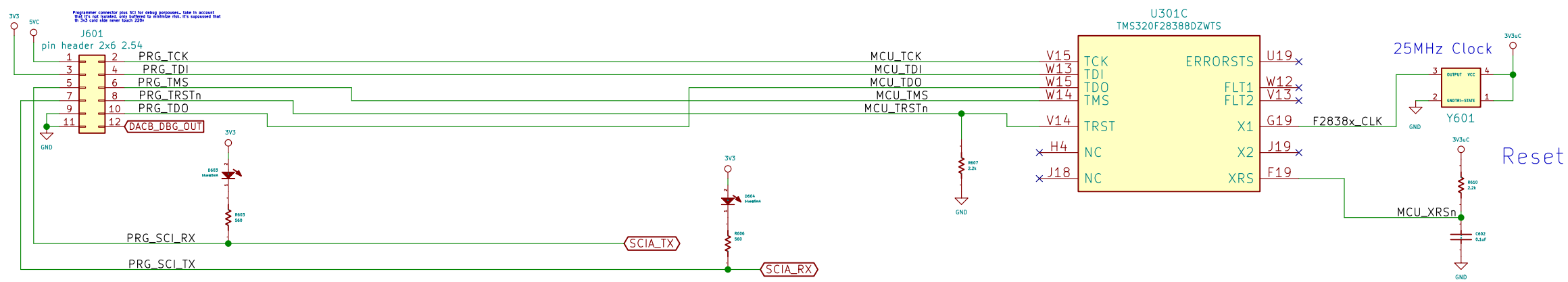
Place near U1



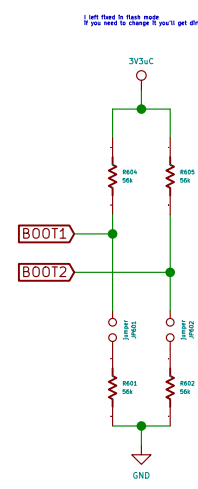
Ferrite Beads
Place near U1



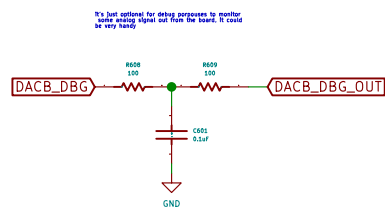
CLK + JTAG + SCI



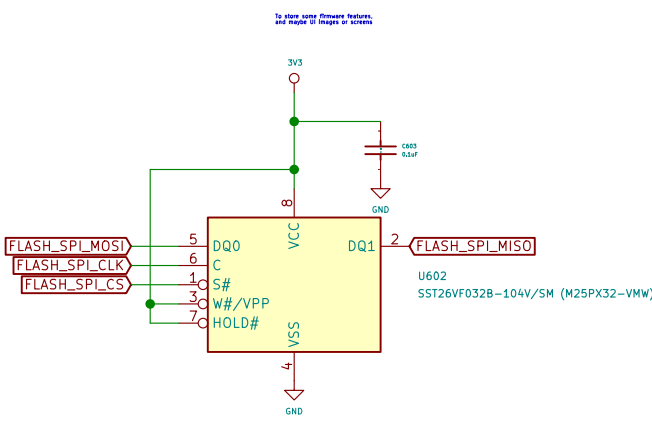
BOOTSRAP R's



ADC/DAC DBG OUT



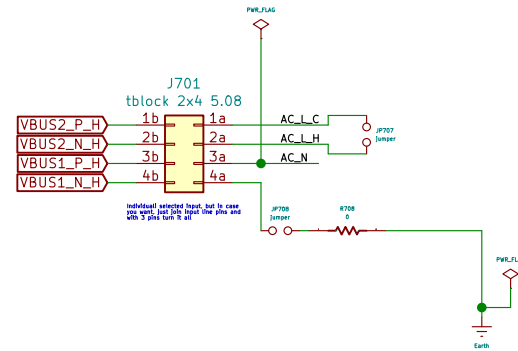
SPI FLASH



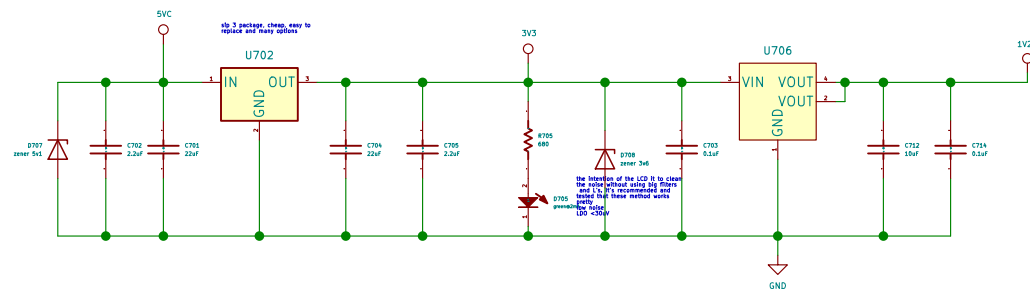
In case the central board has to be supplied directly with socket voltage (220V) population there. It's not a good idea, cause I'll like to keep high voltage outside there. It's a nightmare to get it right, so I let it as an option. But I have the low voltage input 130V and 151V connector slots.

I've chosen to add 2 power supply for Cold and Hot slots because it's cheaper than have only one double sized power supply and add a DC/DC isolated converter, and it has the advantage to choose which side to use and to be able to use the 151V slot. The Cold slot keeps working without affect, I've chosen 15W size boards, has similar price as 10W compatible with 20W, so you could change it if you need more power!

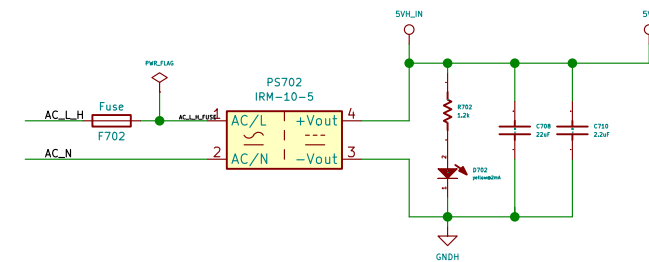
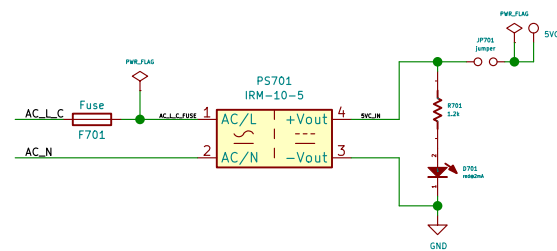
I wrote 15w on the DC voltage side, but it's better to use 12w. But the range could be from 8 to 16 more or less



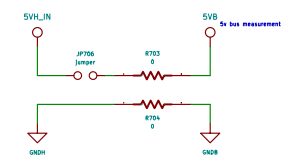
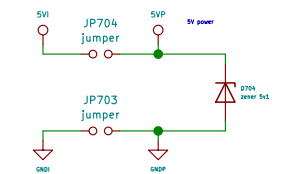
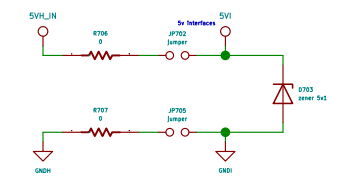
It is intended to use only, I mean ONLY inside the control board, none of these coopers wires has to leave the board. I isolate every single pin from these supply to go outside, take these in account



Take in account that hot doesn't mean that it is referred to high voltage AC socket input path like 220v/380v, it is intended to isolate the uC /logic/user interfase side from accidentally high voltage contact on the hot side and prevent injuries as to make the reinforced isolation possible with one single isolation added



5VH_IN = internal isolated 5V power supply
5VH = for VBUS measurement logic
5VP = for share 5v to the IGBT and power board connectors
5VI = for supply 5V to encoders and external interfaces



dci

Sheet: /ac_in/
File: ac_in.sch

Title: AC input

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

Size: A3	Date: 2020-01-
KiCad E.D.A. kicad 5.0.2+dfsg1-1	

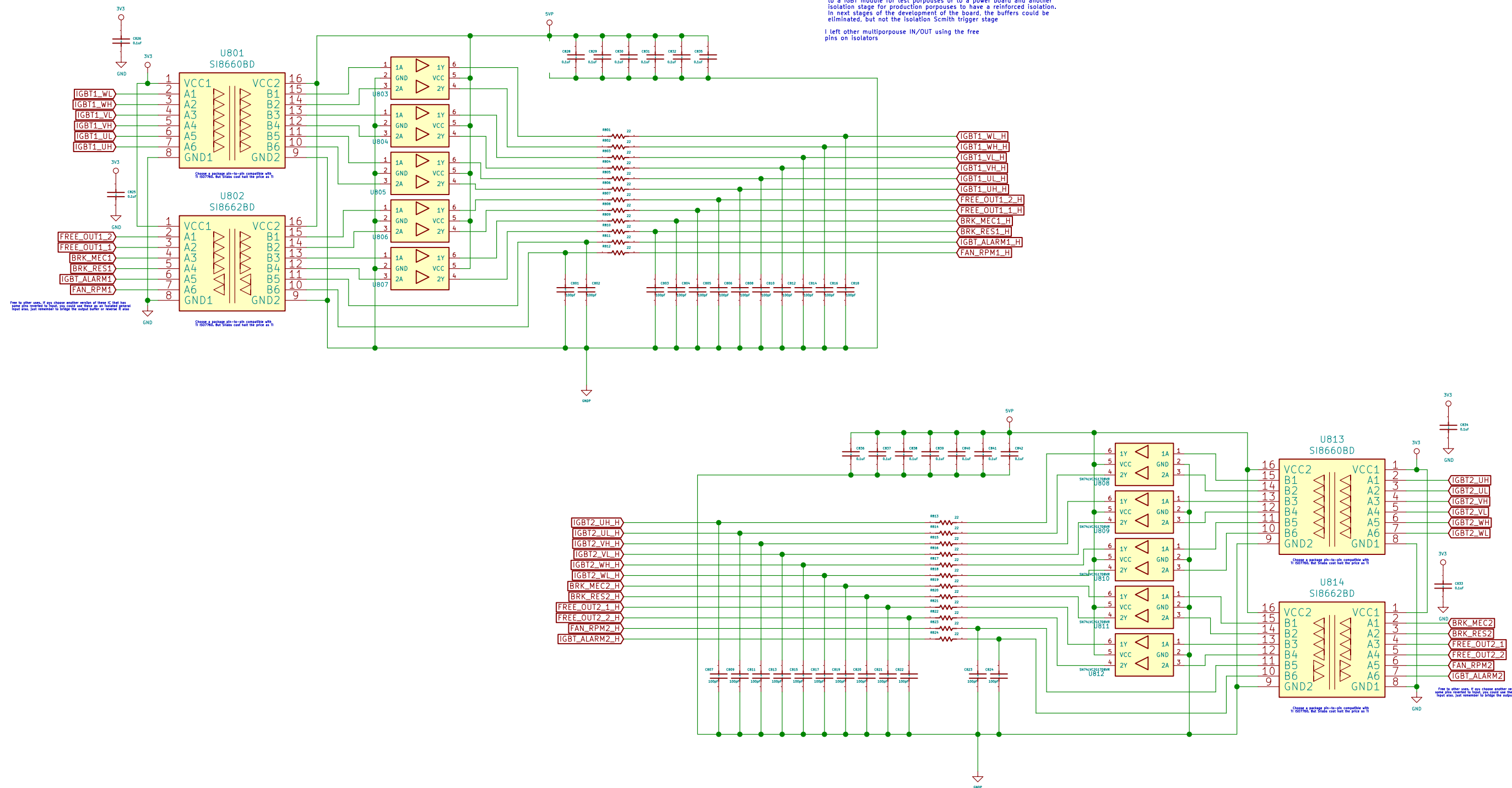
Rev: 1.0

Id: 7/20

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 multiporpose IN/OUT using the free pins on isolators



Pablo Slavkin
dci

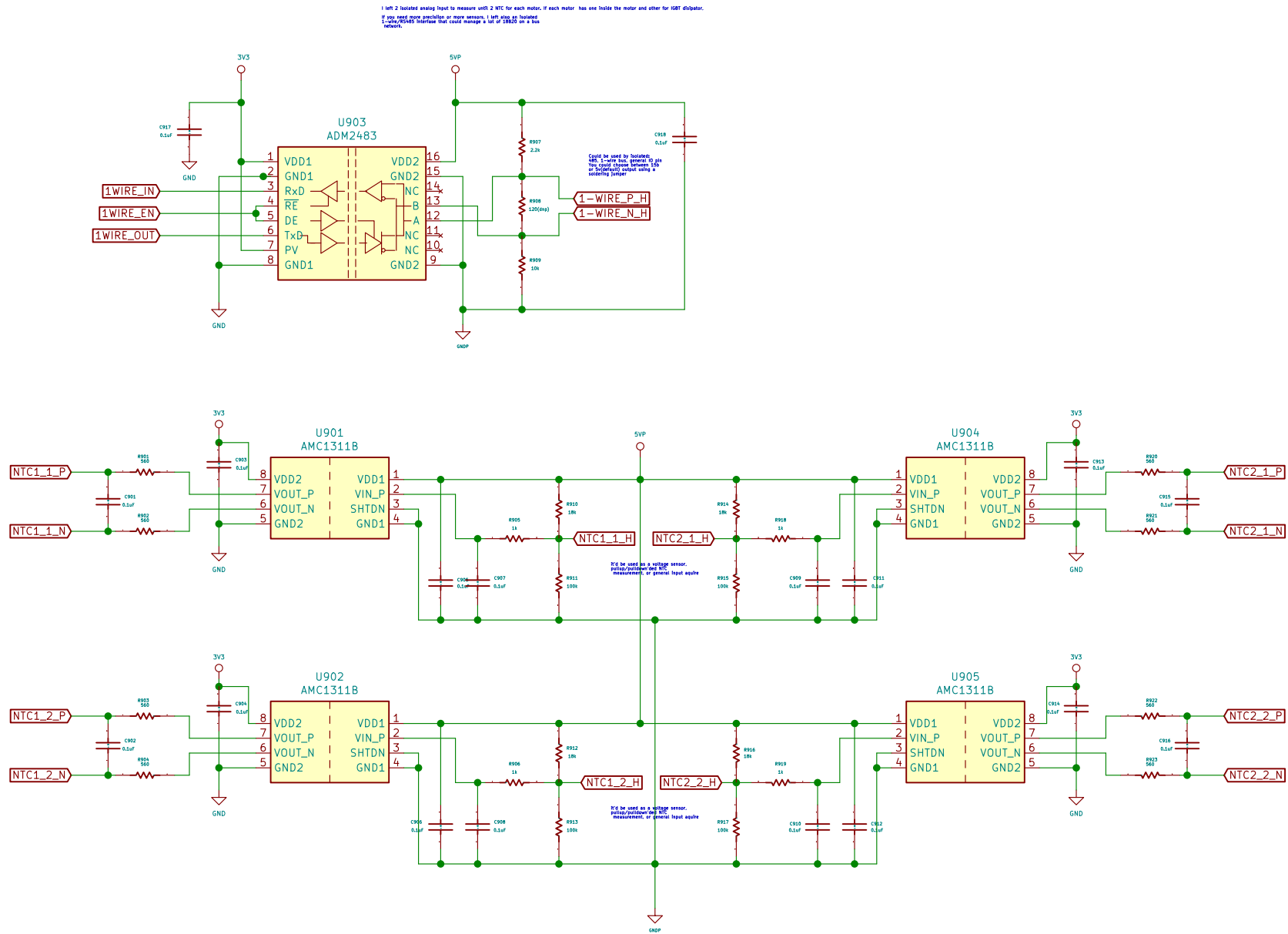
Sheet: /igbt/
File: igbt.sch

Title: Igbt interface

Size: A3 Date: 2020-01-09
KiCad E.D.A. kicad 5.0.2+dfsg1-1

Rev: 1.0
Id: 8/20

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



Pablo Slavkin

dci

Sheet: /temp/

File: temp.sch

Title: gpio

Size: B

Date: 2020-01-09

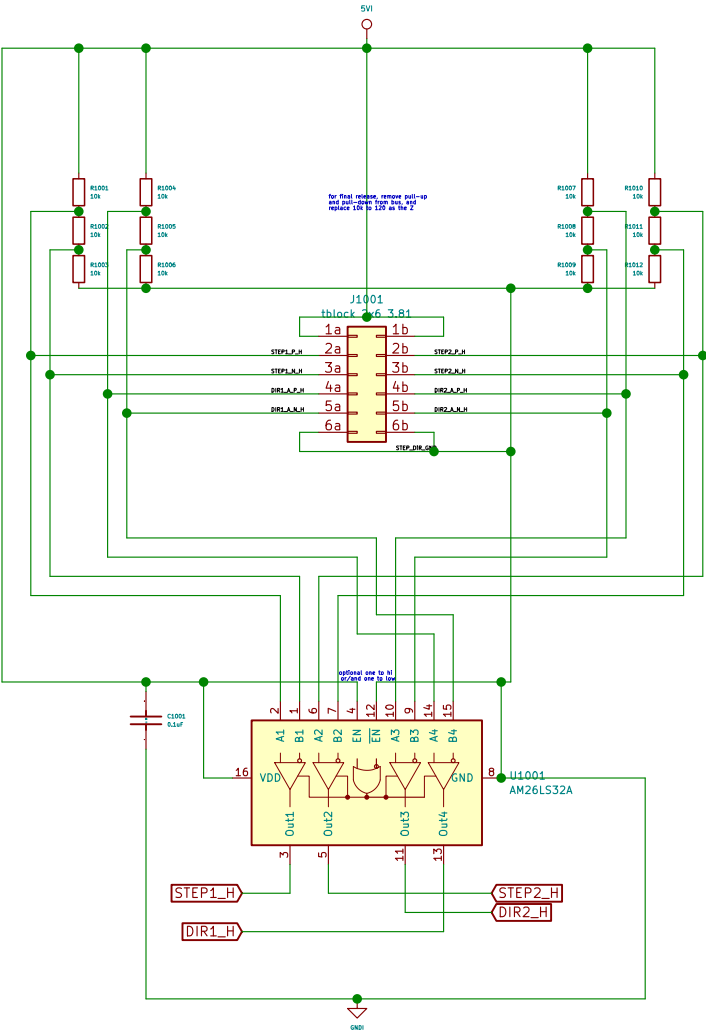
Rev: 1.0

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

Id: 9/20

Differential STEP–DIR input HOT

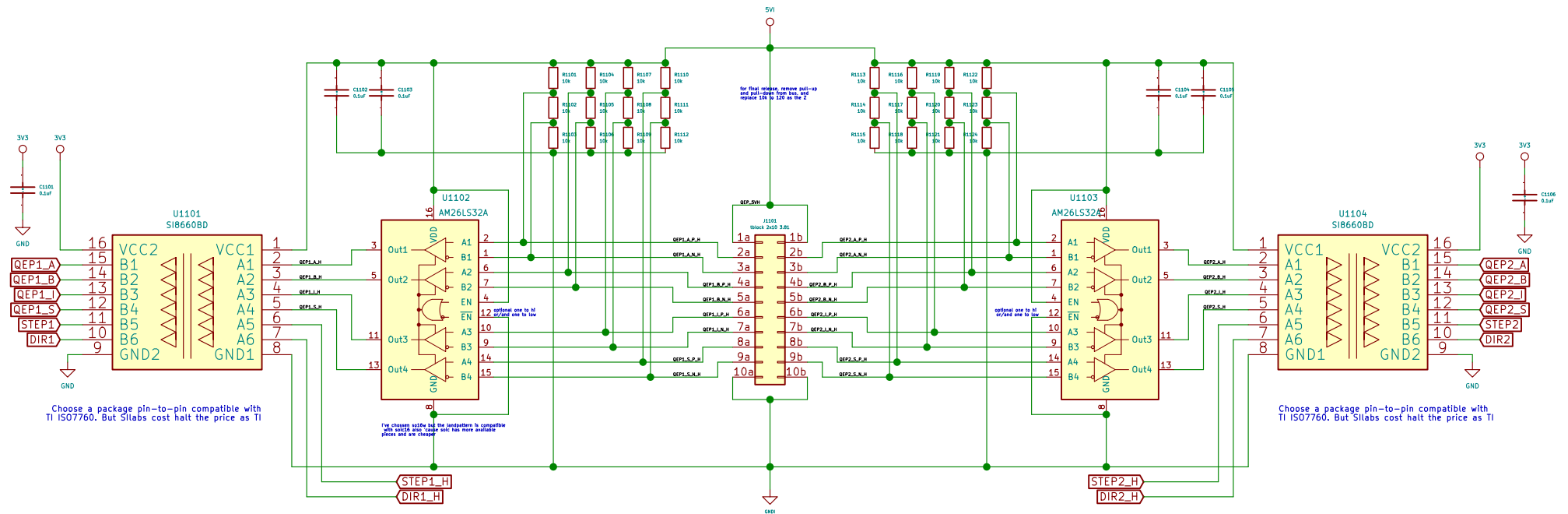
The isolation part is shared with GEP



2x Isolated Idifferential incremental encoder interfase

5v input A-B-I-S

I left the input for two isolated incremental encoders.
I left the 4 signals input plus two auxiliary output for any purpose plus
the ability to choose between 2 or 150V



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

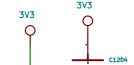
Id: 11/20

A simple CAN driver. It also could be used as a RS485 driver in his speed.. the minimum for CAN is 40kbps, so in RS485 at 9600 could be fine



With these interfaces you could manage isolated RS485 or isolated
GPIO or isolated 1-Wire, using 5M or 10M as a supply

at end of page the Isolator has 4 pins free, so I'd use them plus an
SM501167 to make a RS485 isolated driver cheaper.. The thing is
that I will need to use isolated chips to have 485, what if I need 485
without endat.. and also I need isolated RS485.. so I need a
decision now is to use these IC, that has one only option, and if I
depopulate it, nothing change.. and also has more driver current..



Id: 12/20

Symbols Slots fiducials, and others

I use these sheet to add all manufacturer component including
fiducials, slot holes and things like that

Case



N1301
Housing

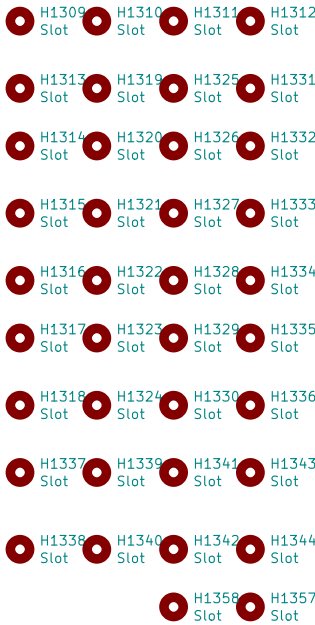
Fiducials TOP



Fiducials Bottom



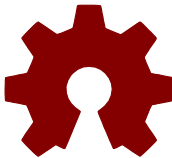
SLOT 'I' anywhere



SLOT V LEMs



mounting holes



Pablo Slavkin

dci

Sheet: /symbols/

File: symbols.sch

Title: gpio

Size: A3

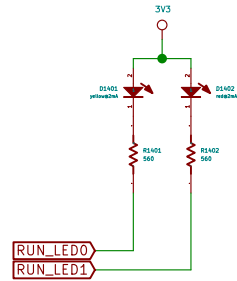
Date: 2020-01-09

Rev: 1.0

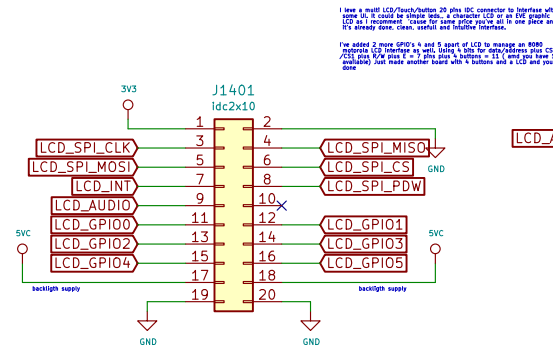
KiCad E.D.A. kicad 5.0.2+dfsg1-1

Id: 13/20

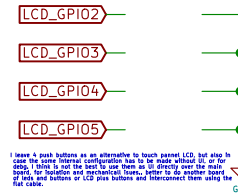
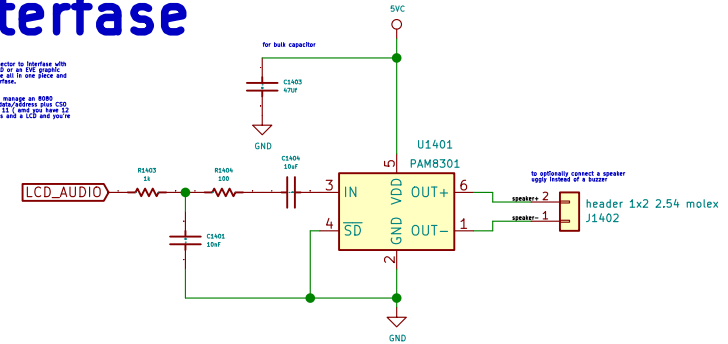
Multipurpose LEDs



LCD UI interfase



Interfase directly using 20 wires flat cable
for NHD-4,3-480272FT-CTXL-T newheaven LCD
or hand wiring
EA eDIP128B-6LWTP
or using any paralel 8080 3v3 interfase bitbanging
the SPI and GPIO pins



I leave a push buttons as an alternative to touch panel LCD, but also in case the same interfase 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 limitation and responsiveness issues, better to do another board or use some buttons or LCD pins buttons and interconnect them using the flat cable.

L1401
NHD-4,3-480272FT-CTXL-T

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: 14/20

8 LEM's current measurement

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

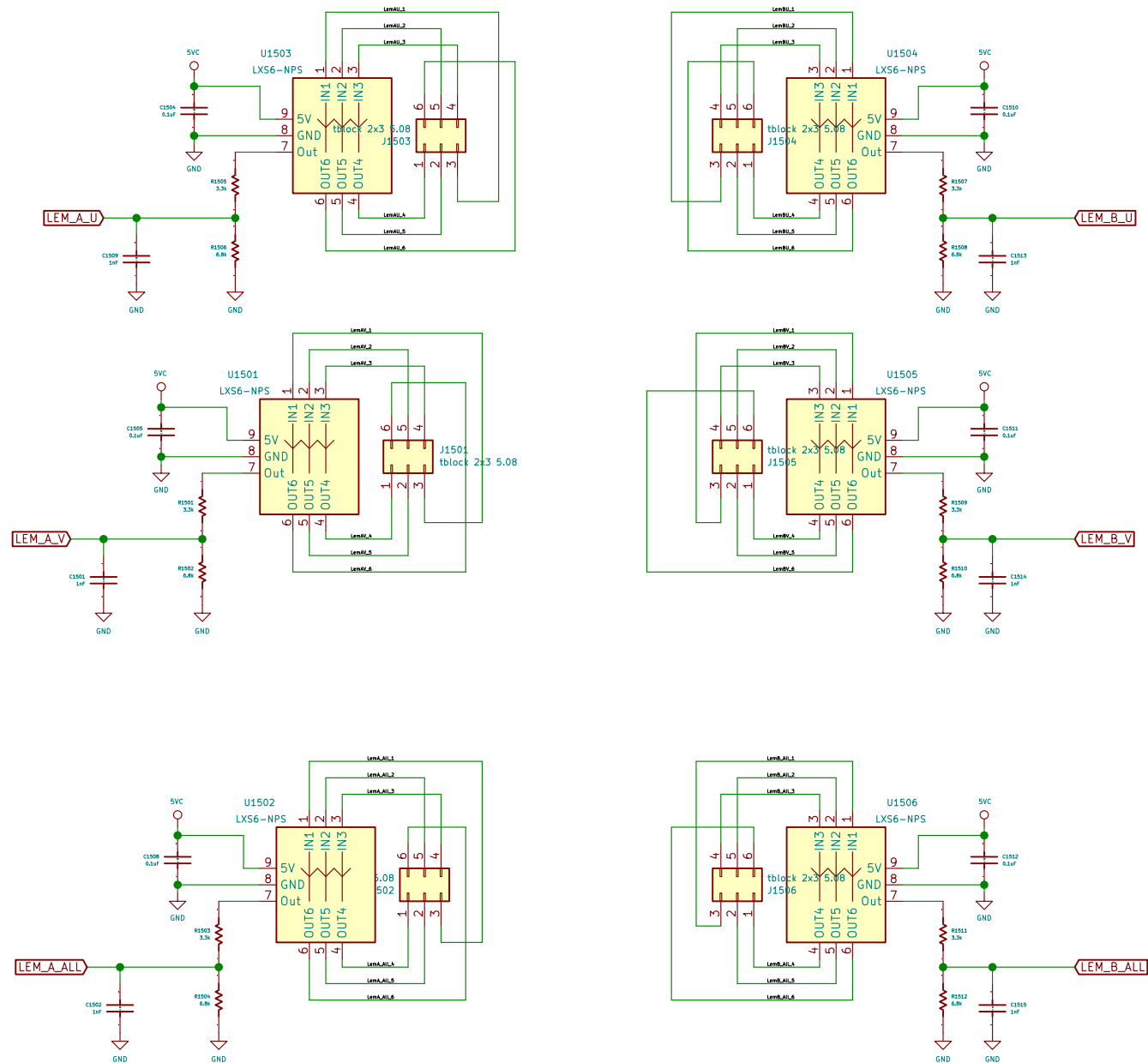
The intention of these terms is to have the capacity on the control board to measure all the current without the need of the Vimp driver. This is a power board's idea. Why? Because I've study the bus topology. I've seen the problem that there are many times that the bus are not connected and the problem that there are many times that the bus are not connected.

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

The intention of these links is to have the capacity on the control board to measure the current, without the need of the signal delta clips (which are not available in the UK). I have used the same board for the L5X and L5S and I am confident that there are very similar, and many of the same, problems. I have also used the same board for the L5S and L5X and I am confident that there are very similar, and many of the same, problems. I have also used the same board for the L5S and L5X and I am confident that there are very similar, and many of the same, problems.

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



LEM_A_W ~~LEM_B_W~~

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
--------------	---------------------

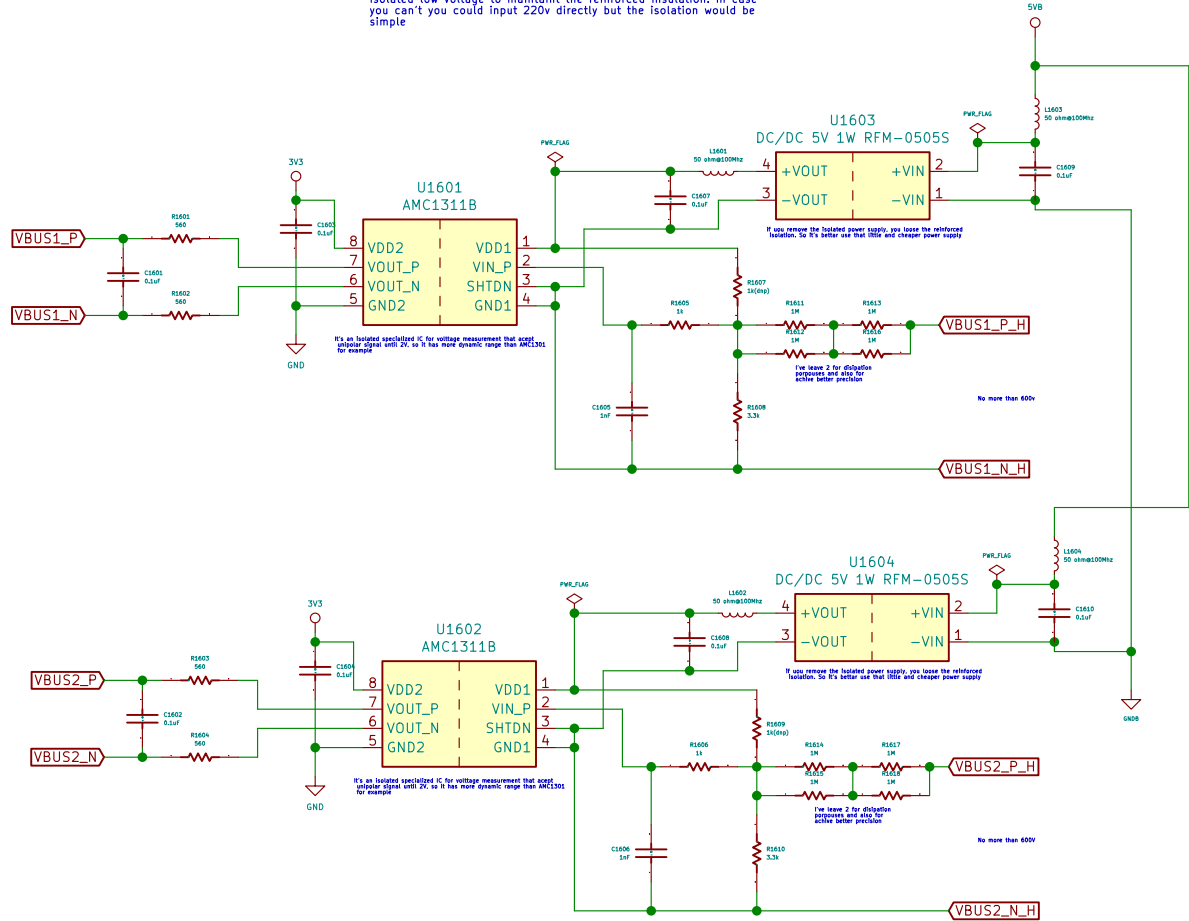
Rev: 1.0

Id: 15/20

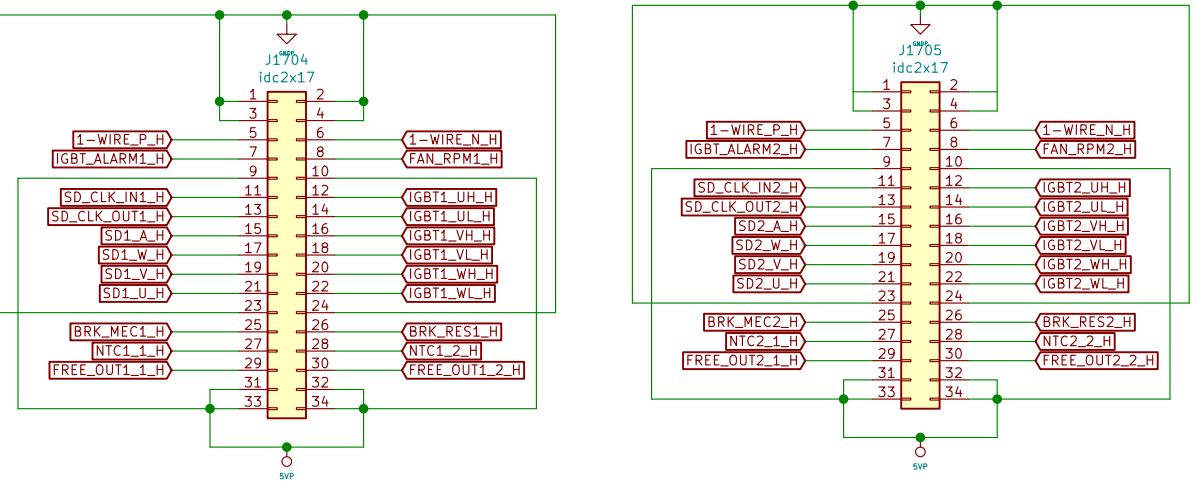
VBUS -> R divider -> ISO ADC -> uC

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

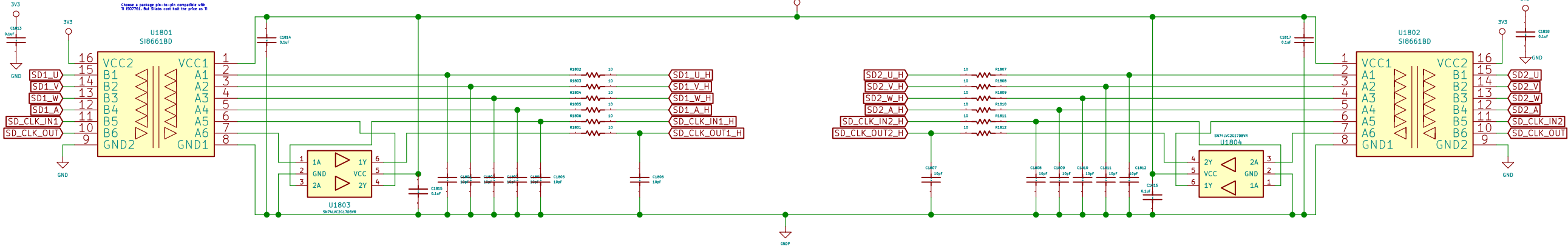


It's intended for power board chiraltry to supply all the signals to control board. The control board will use the 50 filter to acquire the data. All the power part will be at power board.

PAW cili output to powerboard and then come back again to maintain always with respect to 50 data channels. In power board you have to respect traces length between cili and data

I use 50k on power side to be less prone to noise error on the way of connection the two boards (and I've decided to not supply 3v3h too)

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



dci

Title: Shunt Sigma Delta isolated

Rev: 1.0

Id: 18/20

2X Isolated diferential ENDAT interface

dci

Title: *ENDAT/BISS Interface*

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

I've spend hours to choose the GPIO's for each Interface trying to not crash one to the other, just pay attention if you wanna move some pieces...

I've used global labels connector to go from one page to another instead the off-page connector because it's more grounse to errors.. i know that is not too orthodox.. but it's better and fastly for now



dci

Sheet: /uc_gpio/
File: uc_gpio.sch

Title: gpio

Size: A3

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

Rev: 1.0

KiCad E.D.A. kicad 5.0.2+dfsq1-1

KiCad E.D.A. kicad 5.0.2+dfsg1-1 Id: 20/20