

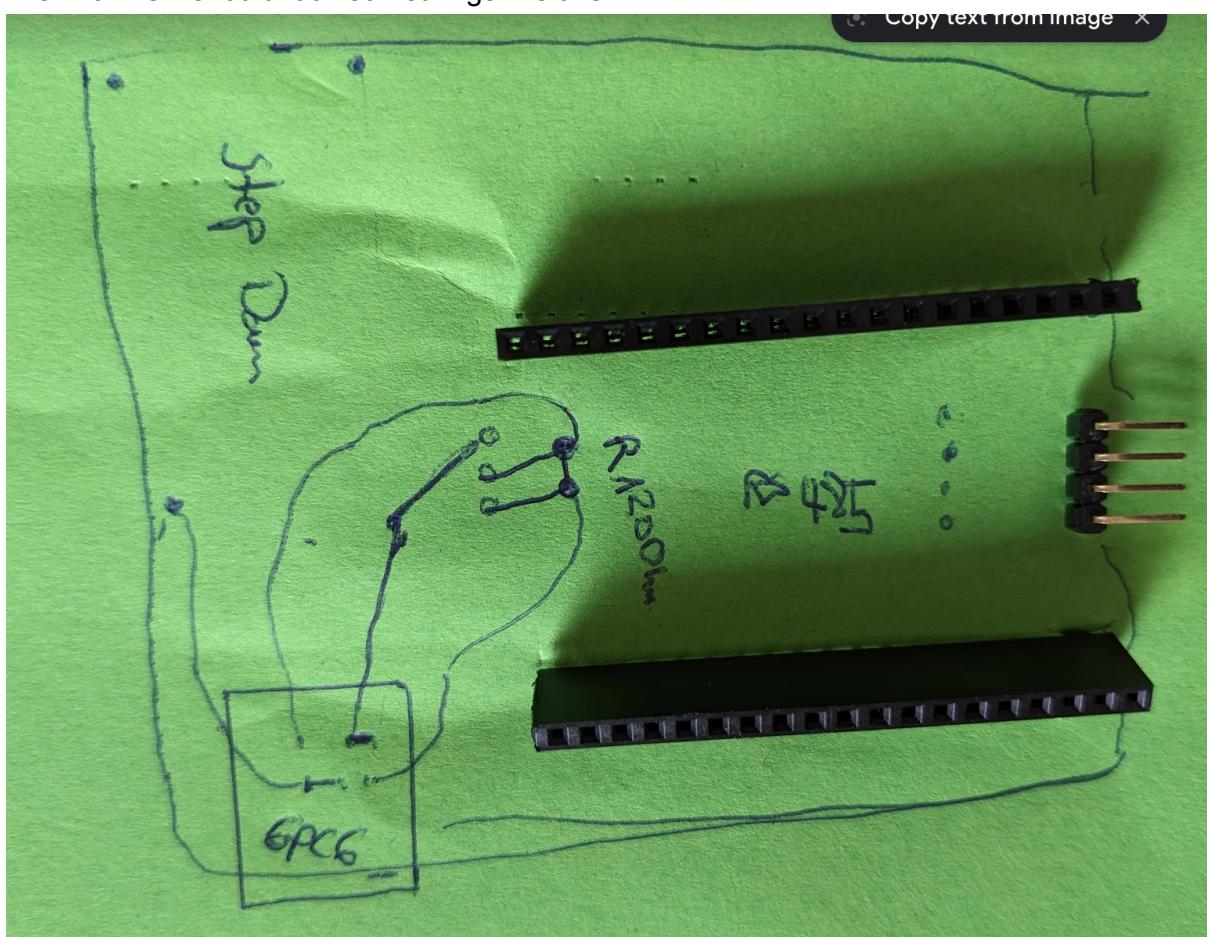
The idea is to create a pcb on which I will put some socket to clip an ESP32-DevKitC-V4, an step-down converter, ttl board, a 6p6c connector and 4 pin header.

In my mind the starting point is the Board of the ESP32 devkitc which is attached and have already the 38 holes where the socket will be to attach clip the esp32 in.

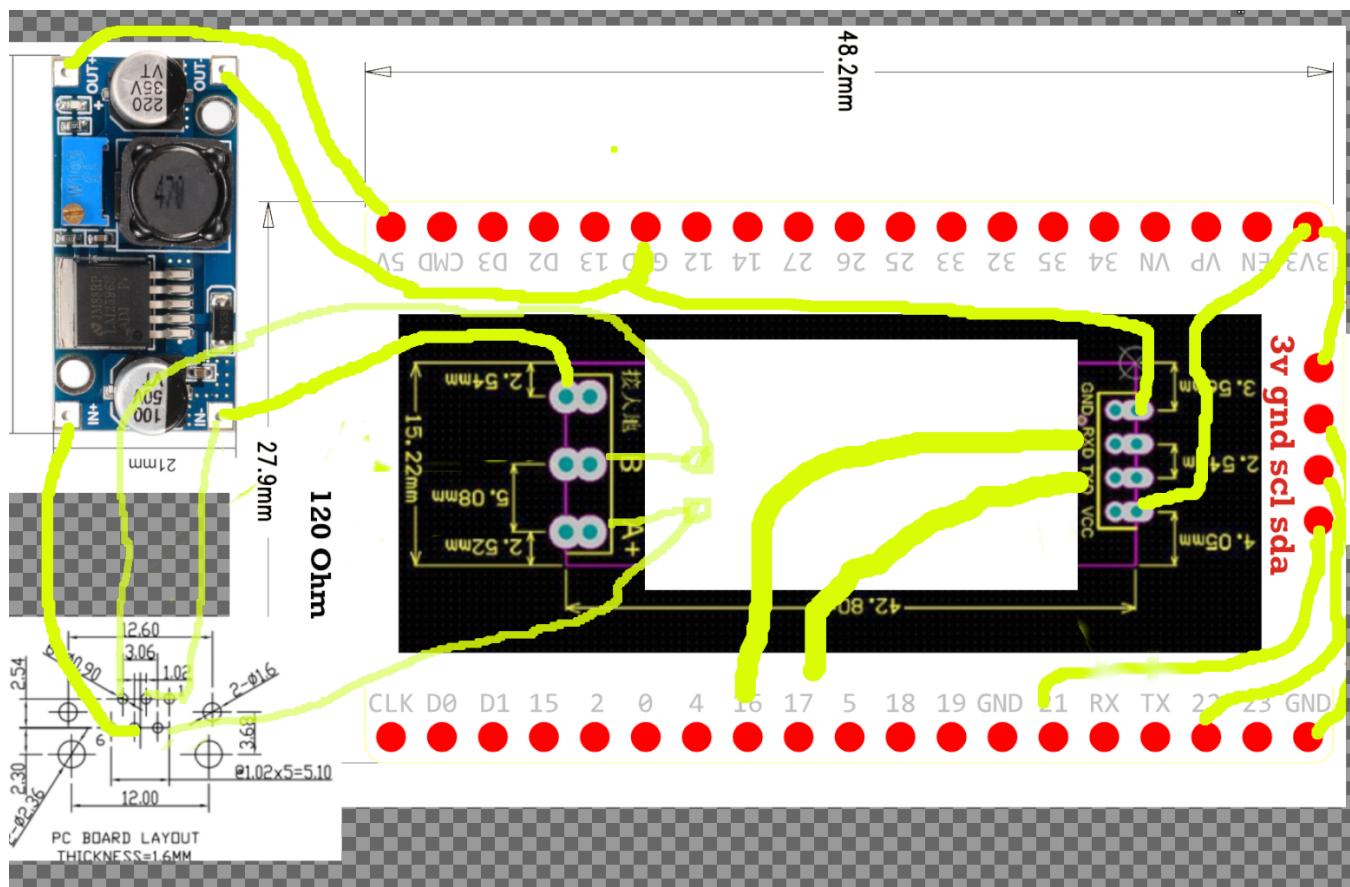
Things to consider

- the pin 5 6 and 1 2 from the 6PC6 connector should be bridged together as they are for the main power supply. Maybe the trace should also be a bit larger.
- Between the pin 3 and 4 I need somewhere to put an 120 Ohm resistance

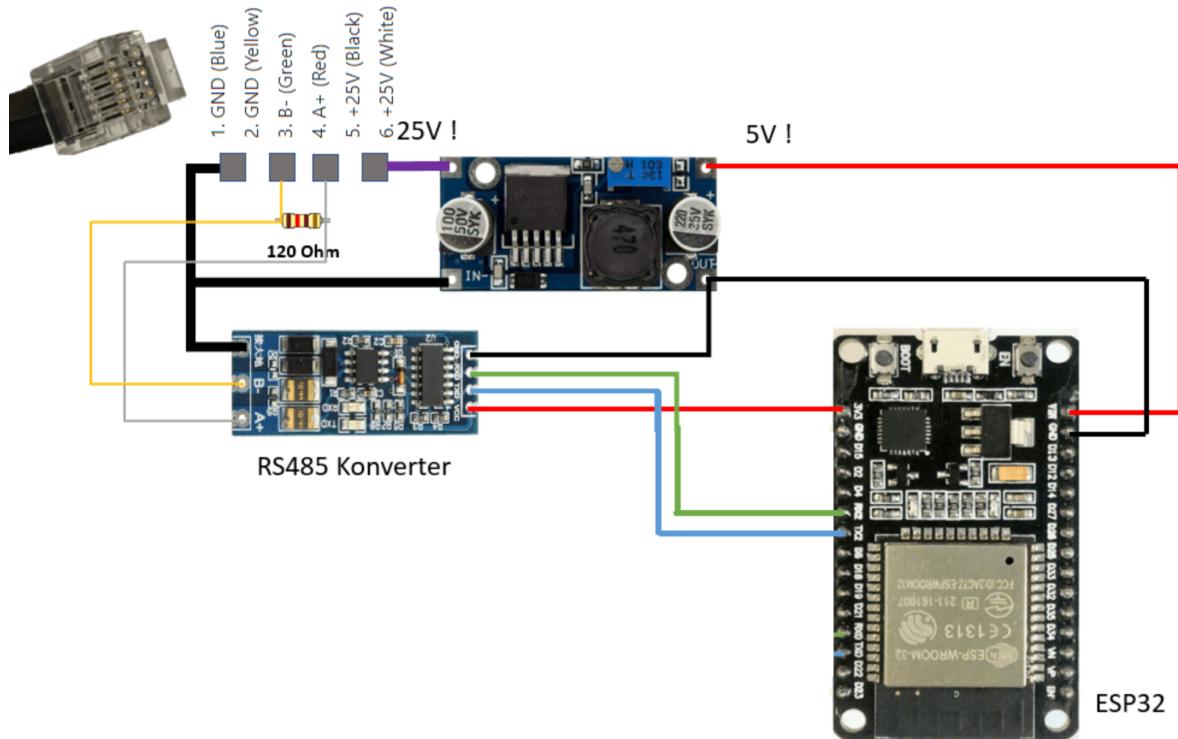
The final PCB should look somethings like this



Here with full trace High resolution attached.

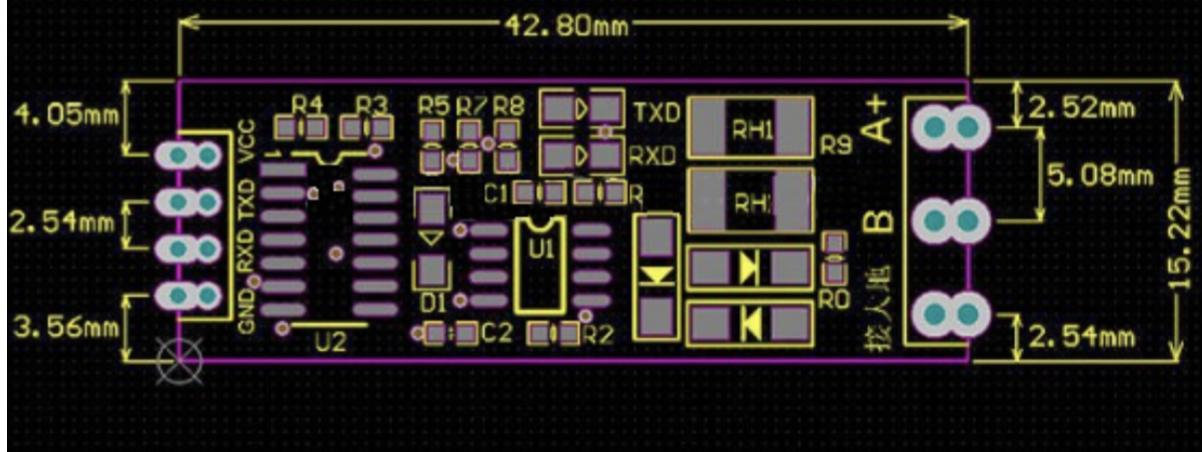


Here the original version of the trace !!Caution!! Pinout on the ESP32 is different (30 pin Version vs 38 pin I will use)



Details and size of the TTL to RS 485

PCB SIZE



I have no details of the Step Down but it measures 40.7 mm between the centre of the holes on length and 17.75mm in the width.

Details of the 6p6C connector

