

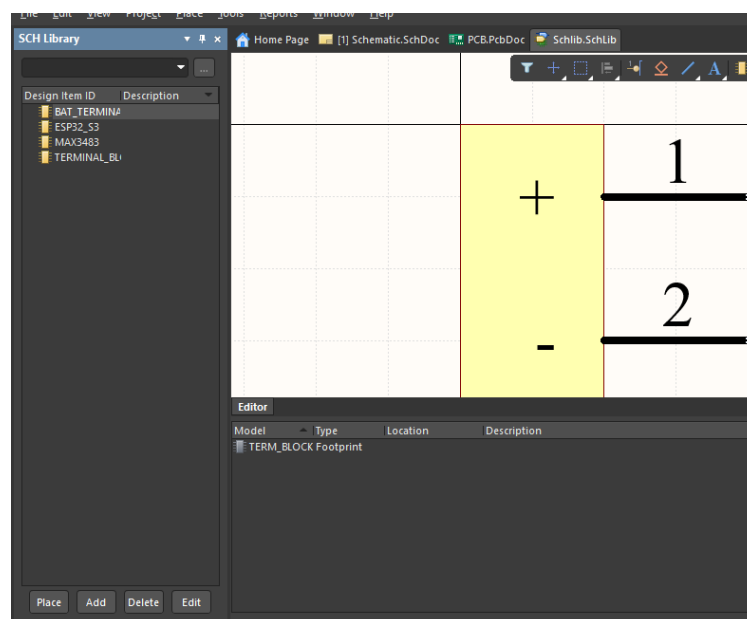
Report – PCB

A PCB will be developed for the 7-in-1 soil quality sensor project. The components that are to be put on the PCB are: the ESP32-S3-WROOM Board, the transceiver MAX3483 and 2 terminal blocks (one with 2 sockets where a 5-6V external battery will be placed in order to power up the circuit and one with 4 sockets where the soil sensor will be connected). The PCB will be developed using Altium Designer environment.

First, the schematics library will be developed ^[1], containing all the 4 schematics for the components, according to the datasheet:

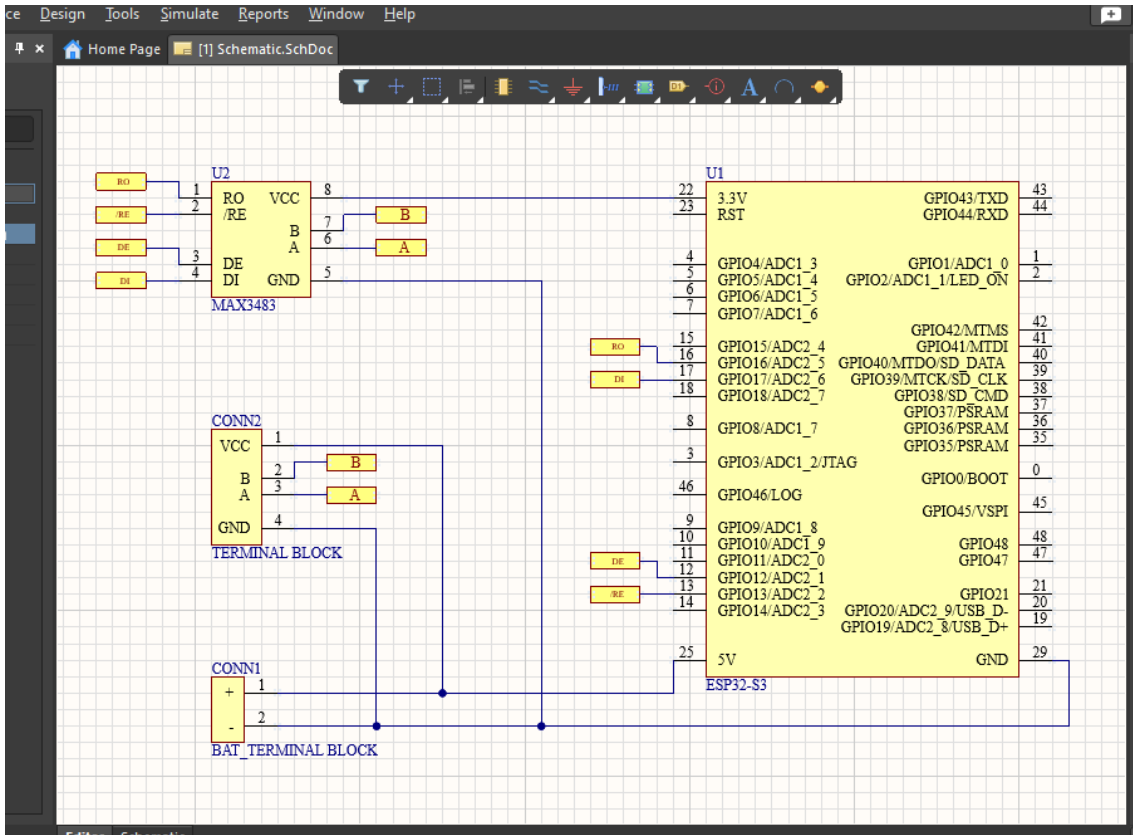
ESP32-S3-WROOM Board	https://github.com/Freenove/Freenove_ESP32_S3_WROOM_Board/blob/main/Datasheet/ESP32-S3%20Pinout.pdf
MAX3483	https://www.digikey.com/en/products/detail/analog-devices-inc-maxim-integrated/MAX3483ESA/1702246
1760510000	https://www.digikey.com/en/products/detail/weidm%C3%BCller/1760510000/459739?s=N4lgTCBcDallwHYBsAGArHFWUgLoF8g
9993920000	https://www.digikey.com/en/products/detail/weidm%C3%BCller/9993920000/269782?s=N4lgTCBcDallwIMxzABnakBdAvkA

The schematic library with the 4 components:



(figure 1)

The complete schematic with the connections:



(figure 2)

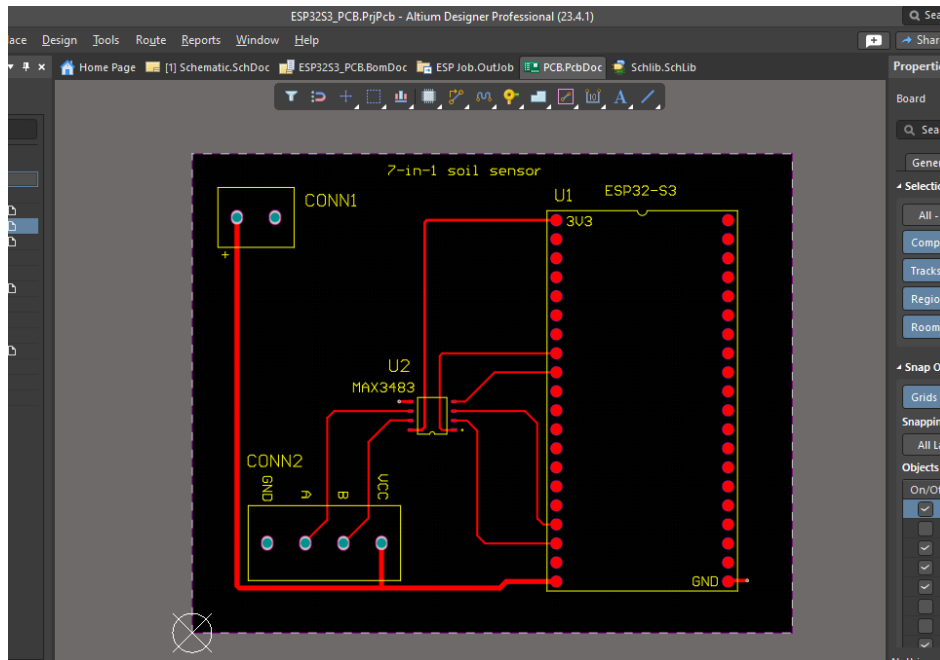
Next, the footprints for the components are developed ^[2], according to the datasheet of each component (footprint library). Also, a check of the footprint is made by importing the 3D model of the components (if these models are found).

For the layout of the board ^[3], the following rules are decided:

- The traces for powering up the microcontroller and the sensor (which need to be supplied with 5V) are of width 0.8mm (meaning max current of 2.8A), enough if a battery of 5-6V is connected.
- Traces through which data is transmitted (DE, RE, DI, RO) are of width 0.4mm. These traces could have been made even thinner because only a small current passes through them, as they connect the pins of the transceiver with the pins of the microcontroller.
- Trace to power up the transceiver (3V3) is of width 0.5mm
- The ground pins will be interconnected on the bottom side where a copper polygon is implemented. The GND pins of the transceiver and of the microcontroller will be connected to the bottom side using vias.
- Board dimensions: 80x64mm

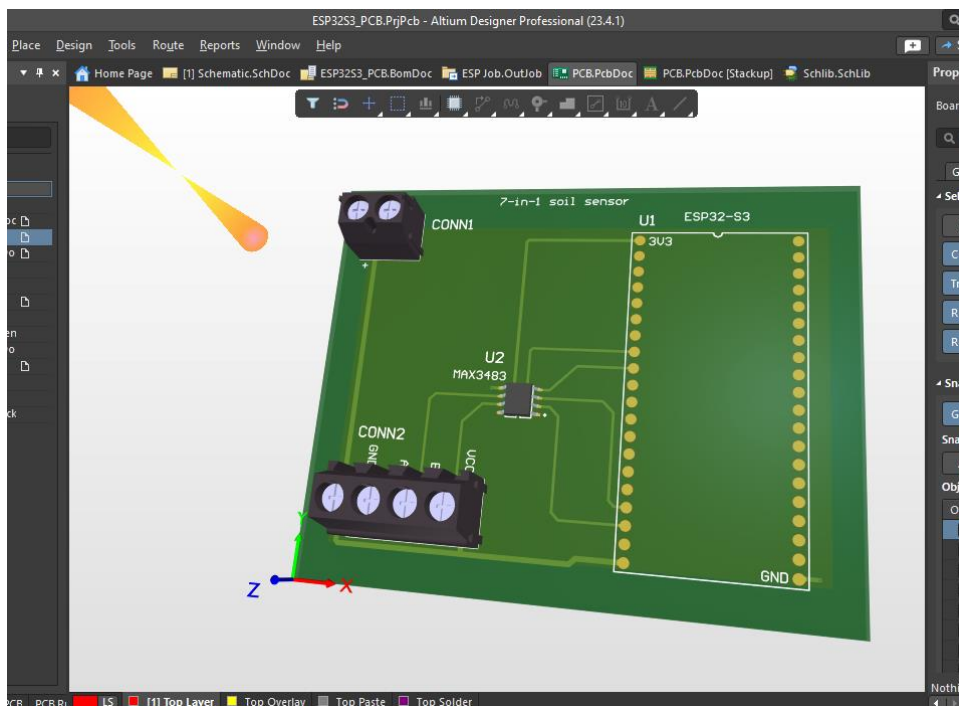
- Dielectric thickness: 1.6mm
- Minimum clearance: 0.3mm (minimum distance between pads, texts, outlines and so on)
- Via dimensions: 0.3mm hole diameter and 0.6mm pad diameter

The PCB layout after routing:



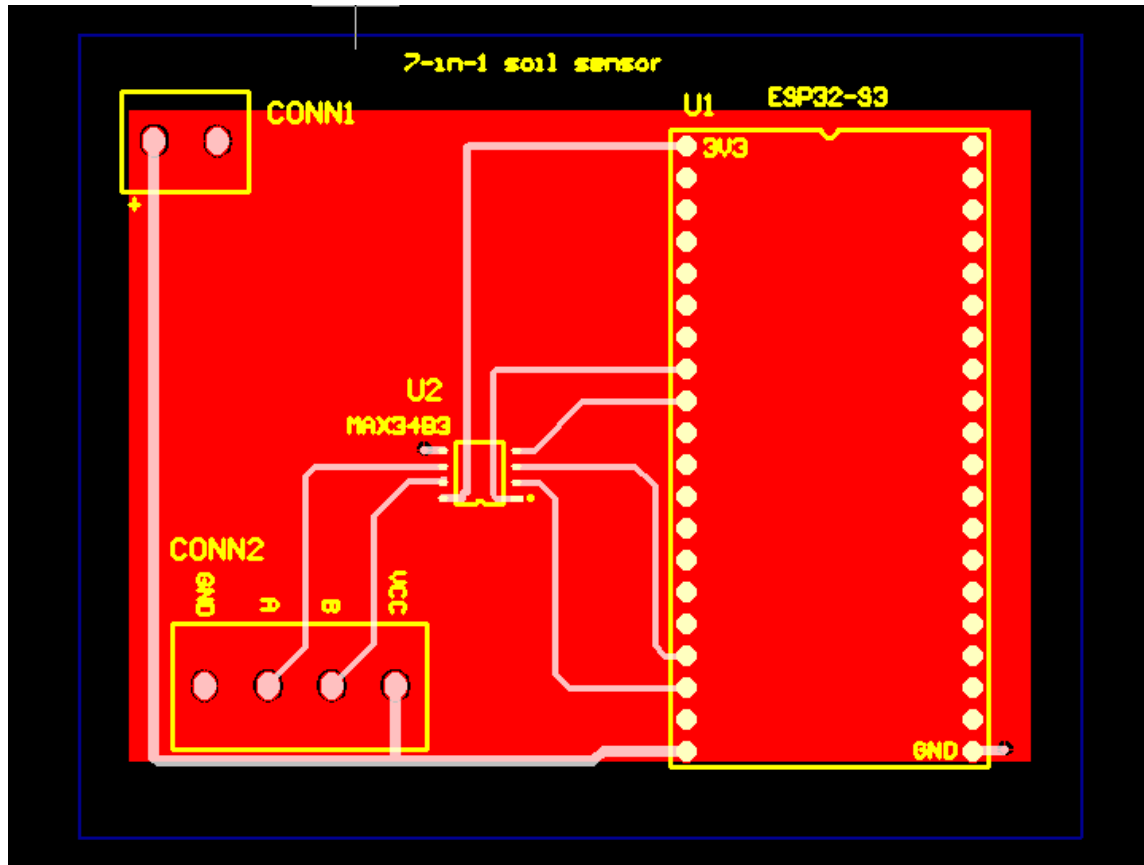
(figure 3)

DRC (Design Rule Check) was also successfully run. The 3D model of the board:



(figure 4)

Finally, the output files are issued ^[4]: schematics (PDF), PDF3D, Layers (PDF), ASM Drawings (PDF), Pick-and-Place file (Text), Gerber, NCDrill, BOM (PDF), Export STEP (STEP). For fabrication the Gerber and NCDrill files were sent:



(figure 5)

References

1. <https://www.youtube.com/watch?v=KpgTud1iQ-4&t=1424s>
2. https://www.youtube.com/watch?v=wxYbIGV9_CY&t=2144s
3. <https://www.youtube.com/watch?v=2I2TX3RLEGM&t=304s>
4. <https://www.youtube.com/watch?v=W21dORx5ceI&t=24s>