SYSTEM-IN-PACKAGE

Printed circuit board layout design including the chip and basic components for serial operation and communication

For a better understanding of this hypertext, we recommend reading the contents **Checking the recommended pad design for the chip and sizing information in the datasheet, Test-board prototyping definitions**, available in the System-in-package track, and **Knowing the fundamentals needed for test-board design and manufacturing**, located in the *Evaluation Board* track.

Test-board Schematic Circuit Design

The schematic circuit design will be based on a circuit previously defined in the Fundamentals Track (Test-board Design), in which it was defined that the circuit we will use will be one containing the Hana HTLRBL32L CHIP and that it will be used to make a transmission through LoRa protocol of a pulse and with signal reception to light up an LED in the receiver.

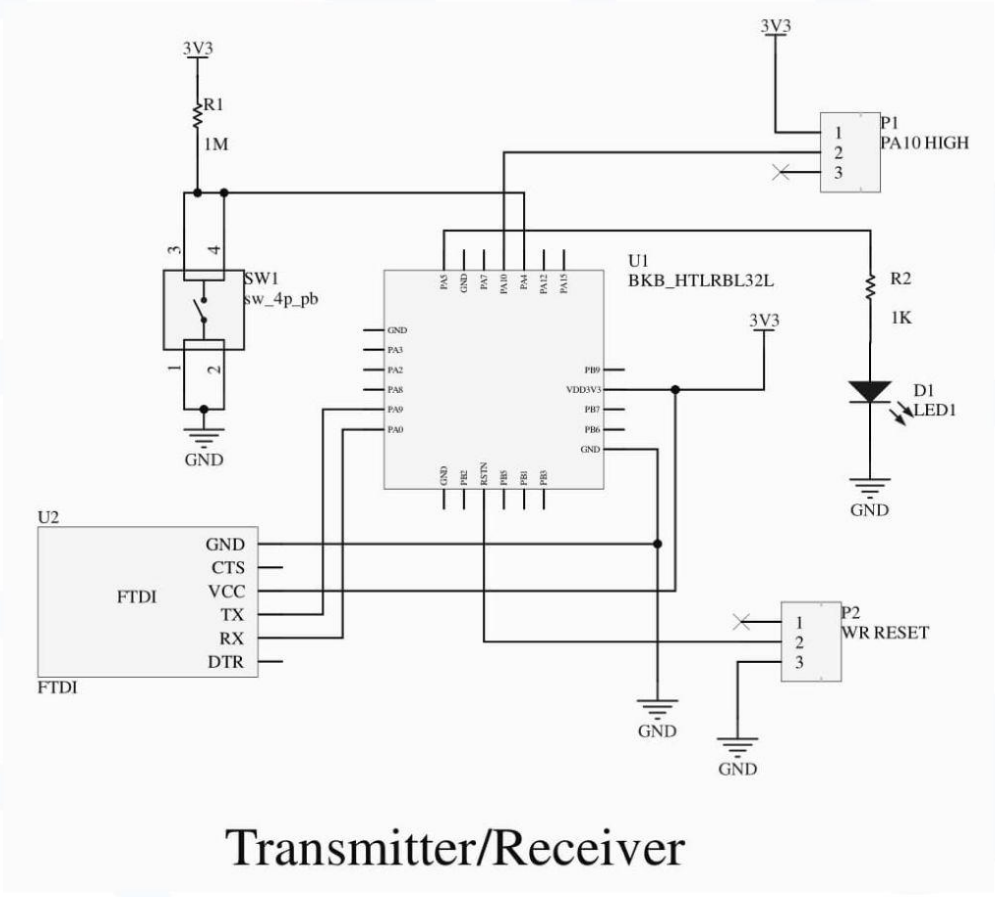

Image 01: Schematic circuit layout. Source: *The Author*.

The ECAD software

ECAD (Electronic Computer-Aided Design) software is a tool for creating circuit layout designs and electronic components on a printed circuit board. In this hypertext, the examples used will be based on the use of **Altium Designer** software.

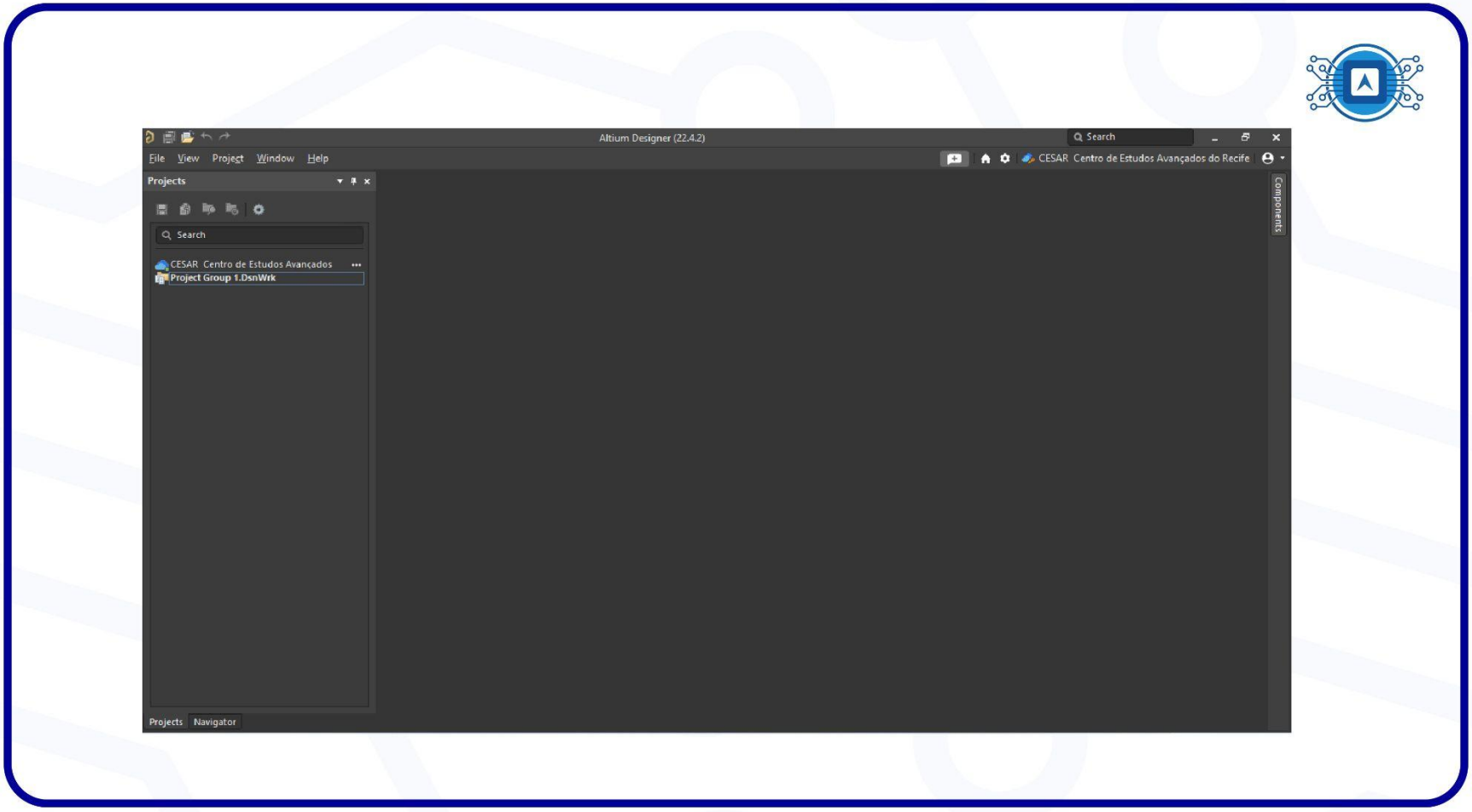

Image 02: Altium Designer software's working area. Source: *Screenshot by the author*.

Circuit design

In Altium Designer, create a new project and then start a schematic. Insert the components as shown in Image 01. If you don't have a drawing of the pads or the chip, you can find the datasheet on the Hana Electronics website and draw the symbol and the footprint of the component.

Start by taking the components and bringing them to the sheet, starting with the already drawn symbol of the breakout board we will place in the center of the sheet as illustrated in image 03.

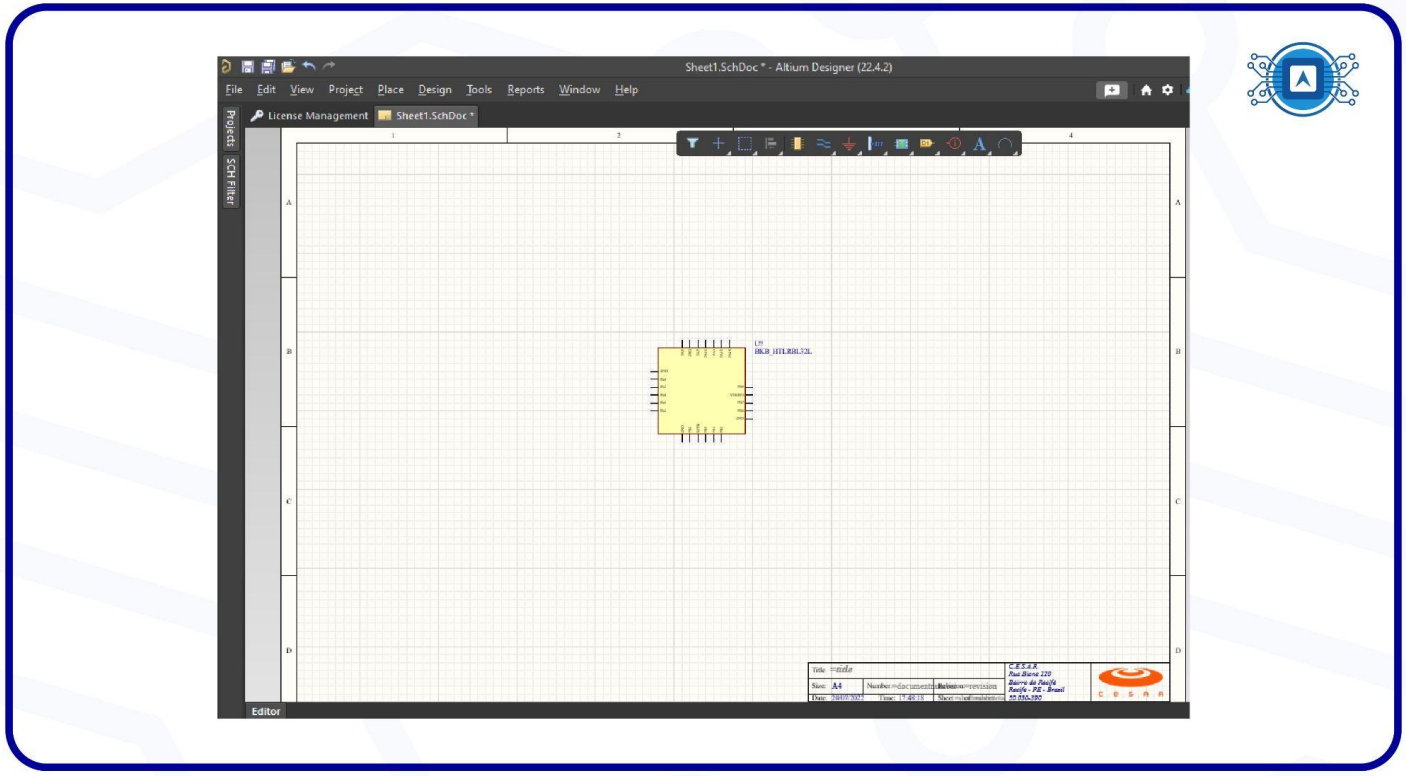

Image 03: Component insertion. Source: *Screenshot by the Author.*

Then add the other components as shown in image 04. Now insert a push-button switch (SW1) and a 1M resistor that will be in series, as seen in the image below, and then position all the other components according to the drawing and the connection needs for a better reading of the schematic.

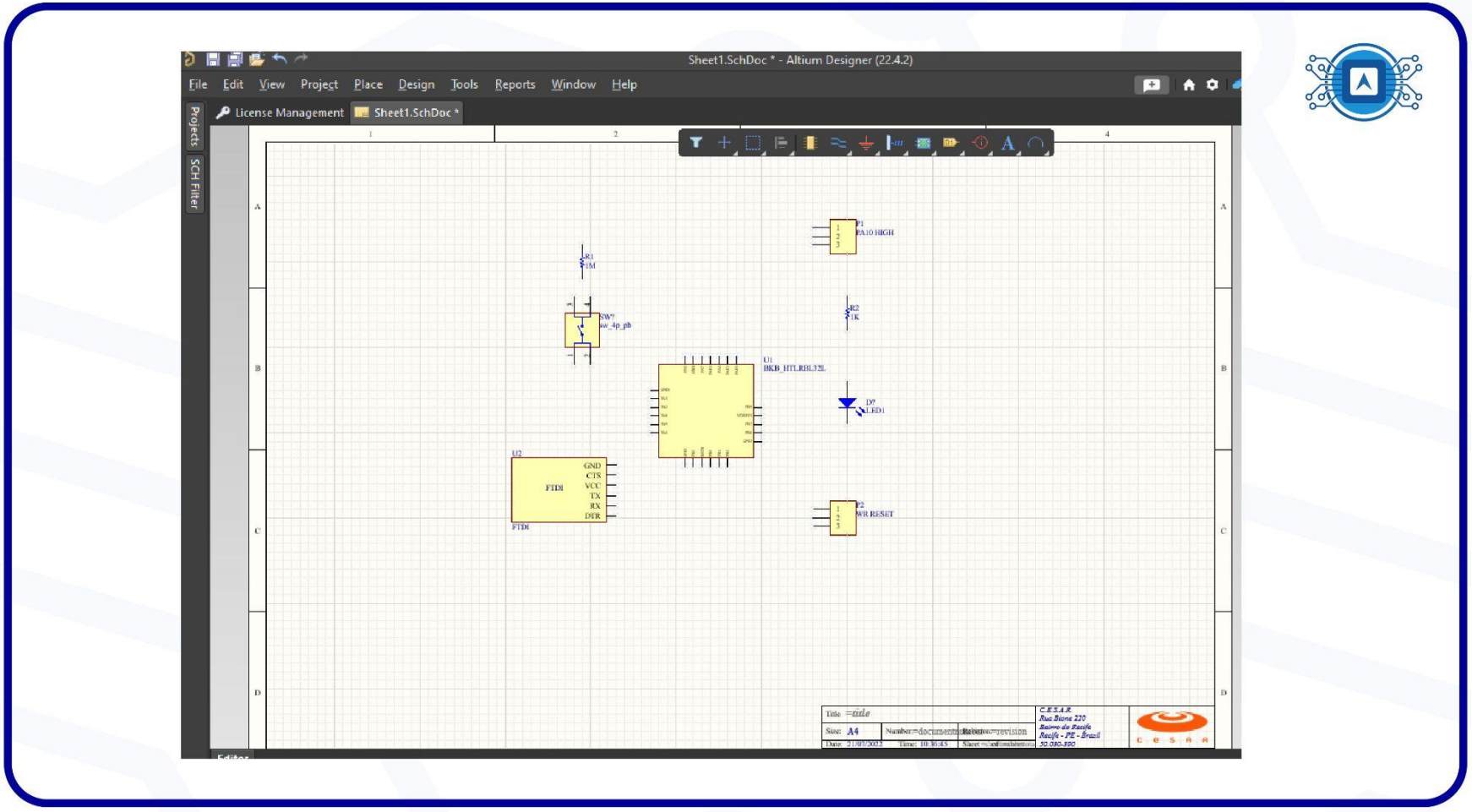

Image 04: Component arrangement. Source: *Screenshot by the Author.*

At this point the connections of the circuit components will be made. Start with the connection of R1 to the 3.3V terminal, of SW1 to the GND terminal, and also the connection of SW1 to PA10 of U1. Doing so for all other components.

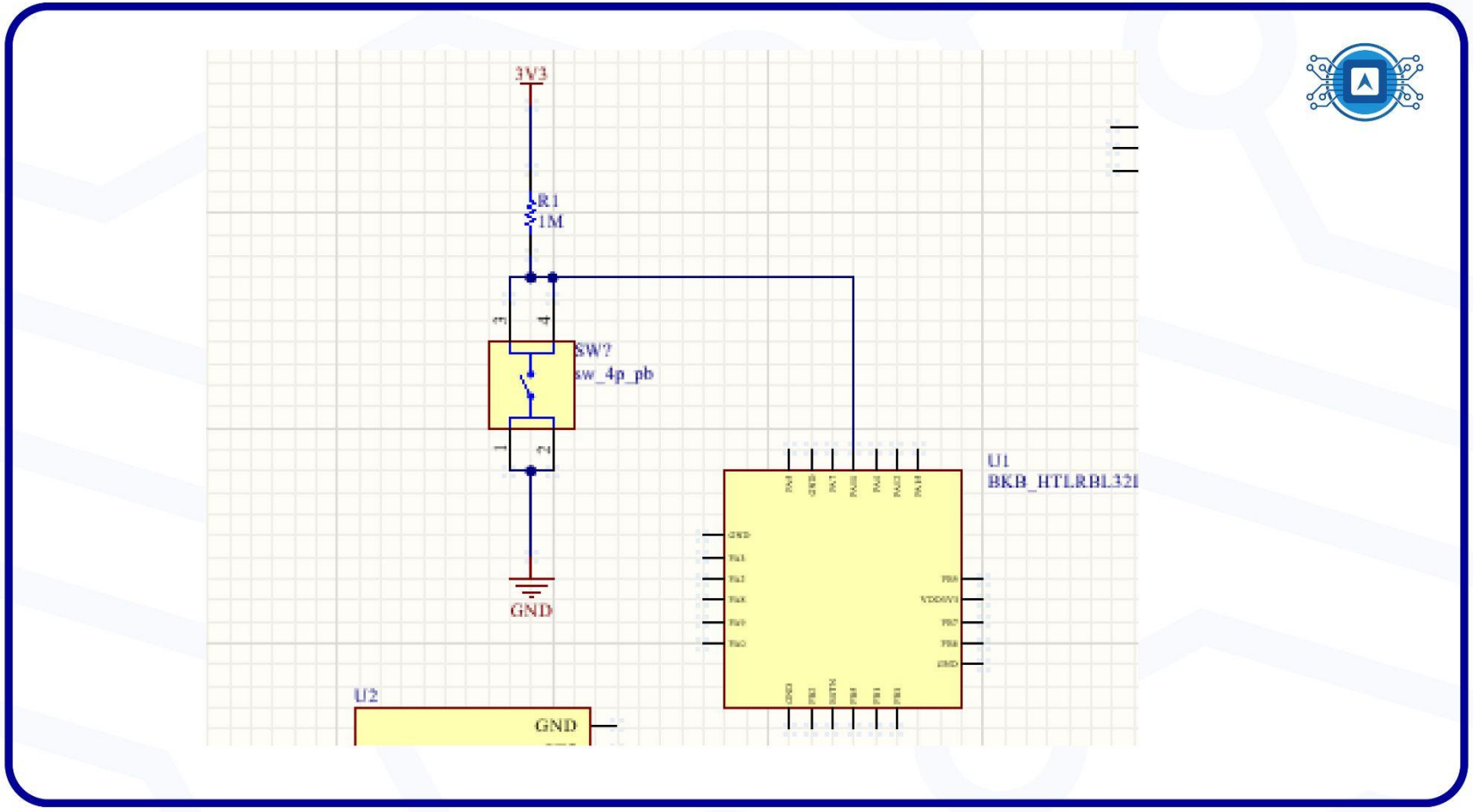

Image 05: Connecting components. *Screenshot by the Author.*

To finish, enter a text, example “Transmitter/Receiver” and validate the drawing for any remaining connections or errors, missing or duplicate component numbering, then save the drawing.

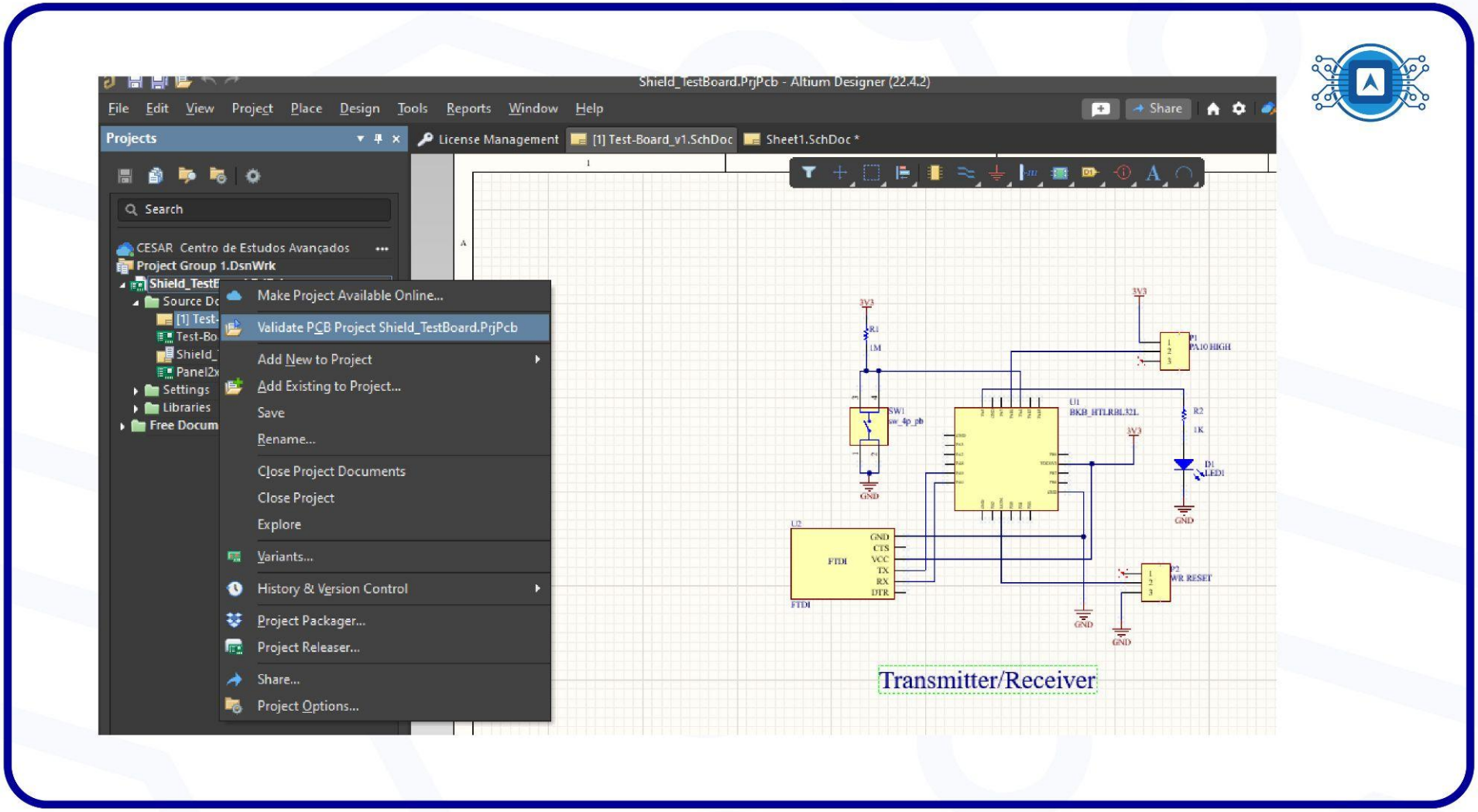

Image 06: Design Validation. Source: *Screenshot by the Author.*

RF antenna circuit design

In this section a schematic of a circuit using the HTLV BL32 (LoRa/BLE) chip will be drawn, with values found through the simulation present in the hypertext about **Test Board Definition**.

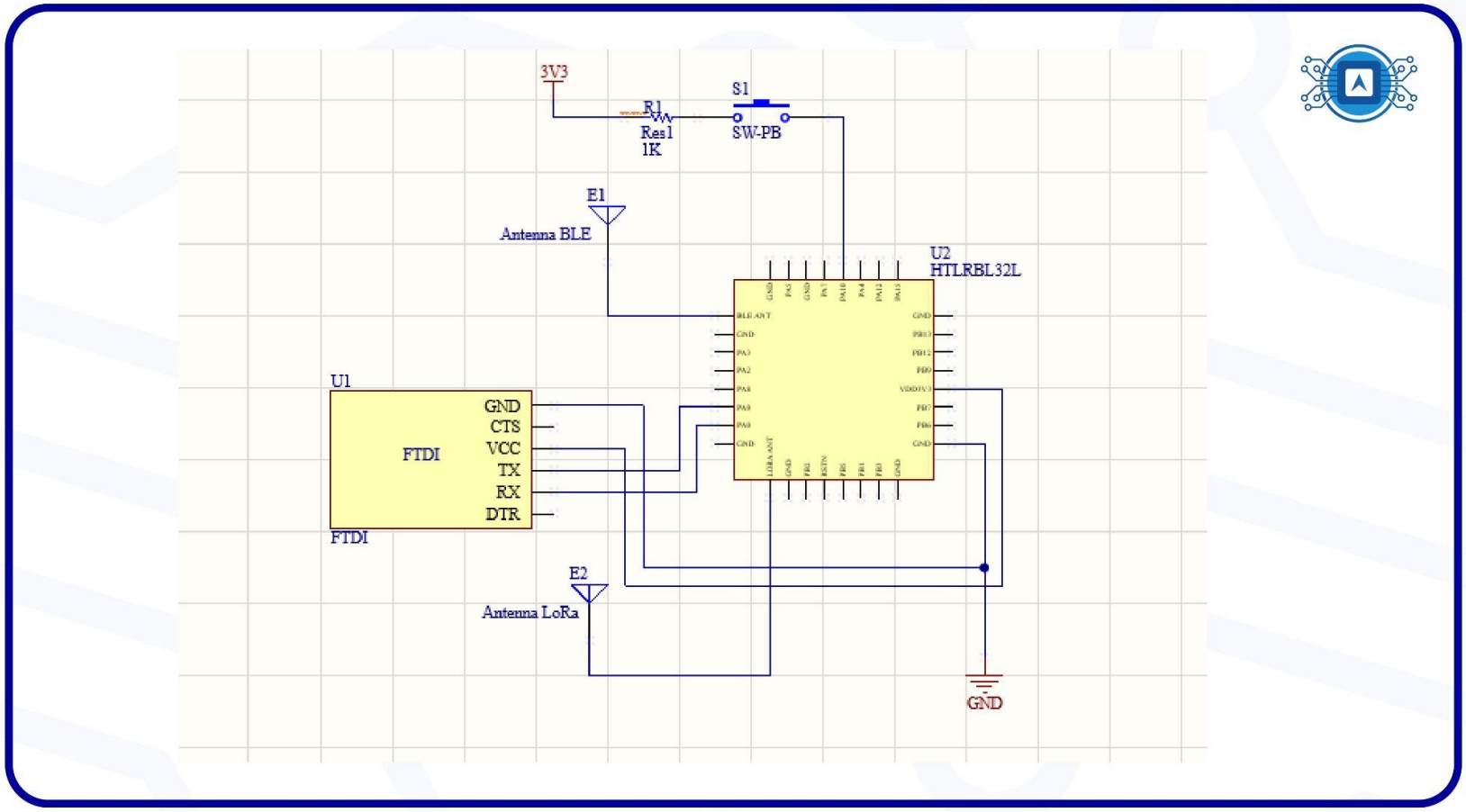

Image 07: Basic circuit of a transmitter (Test-board). Source: *The Author*

According to the electrical schematic shown in image 08, in the LoRa antenna line you will include the components found in the simulation. It is known that recently ANATEL published **act 14448** regulating LoRa technology in Brazil, defining the frequency plan for Latin America the same as the Australian-standard 923 MHz (915 MHz to 928 MHz). Use the values found in the matching line simulation for the 900MHz frequency (Exercise 2)

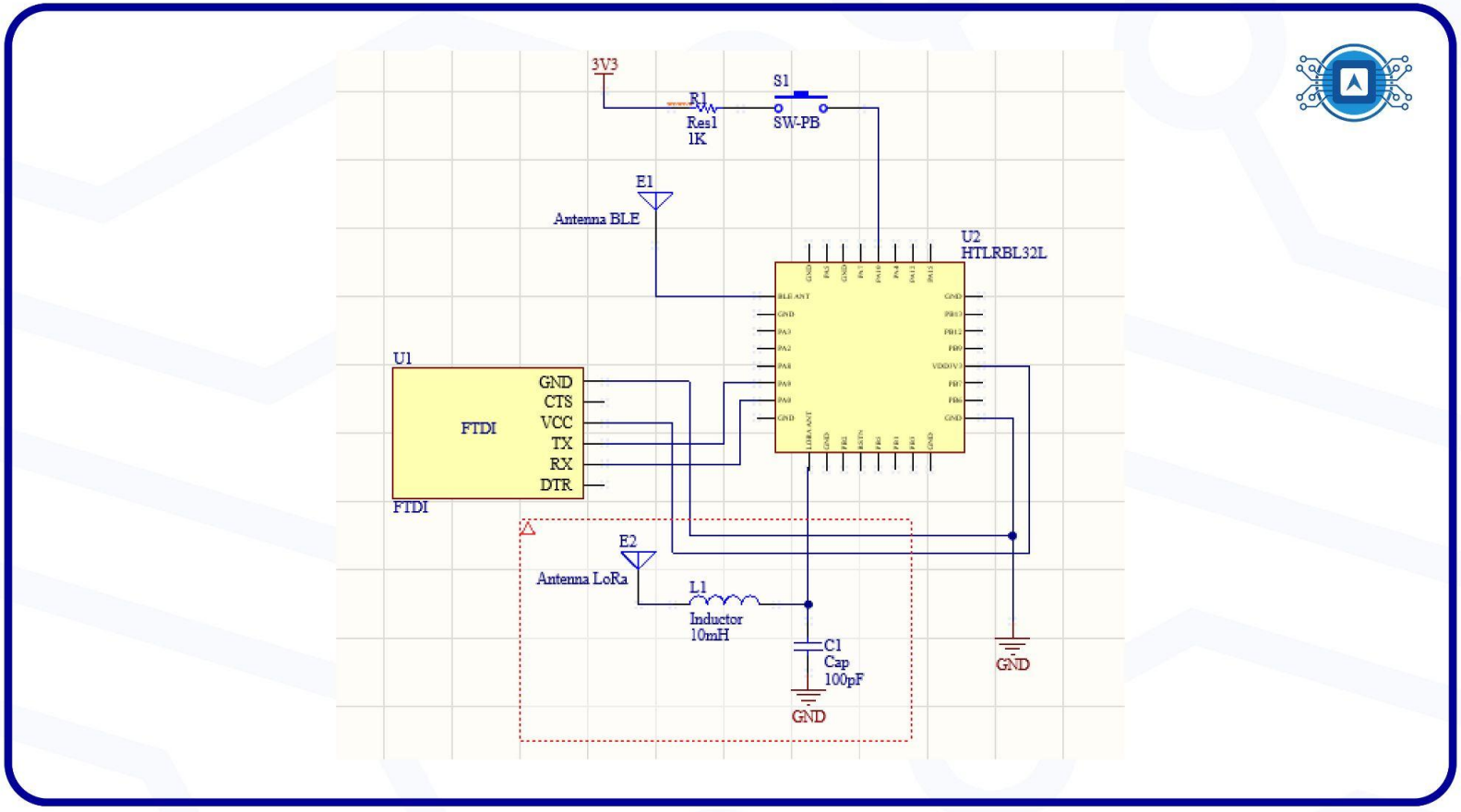

Image 08: Test-board circuit with impedance matching. Source: *The Author.*

Test Board Layout Design

Now, from the schematic diagram design of the test board, done earlier, design a PCB in Altium Designer or other electronic CAD software.

- To make the PCB design, in Altium Designer, go to FILE - NEW - PCB.
- Save it with the name ShieldTestBoard.

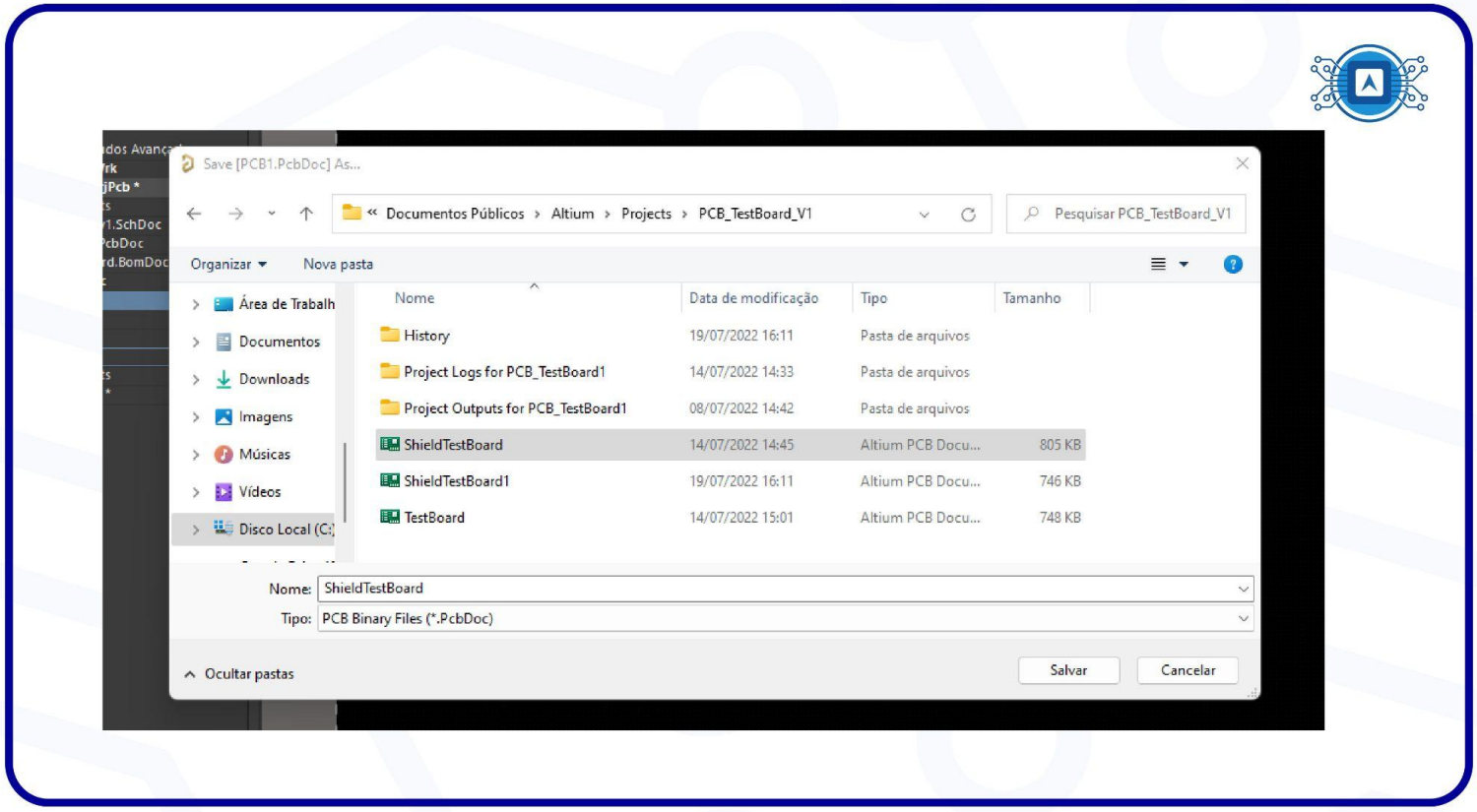

Image 09: Saving a PCB file. Source: *Screenshot by the Author.*

- Import the components from the electrical schematic, under *DESIGN* > *Import Changes From* (name of the schematic design). They should already have the pad design of the components that are called *footprint*;
- A confirmation screen with all the components and links to Validate and Execute will be shown. If everything is correct click OK;
- The components after being imported are laid out on the side of the board to be inserted into the dark area, which is designated to be the PCB area, as shown in the illustration below.

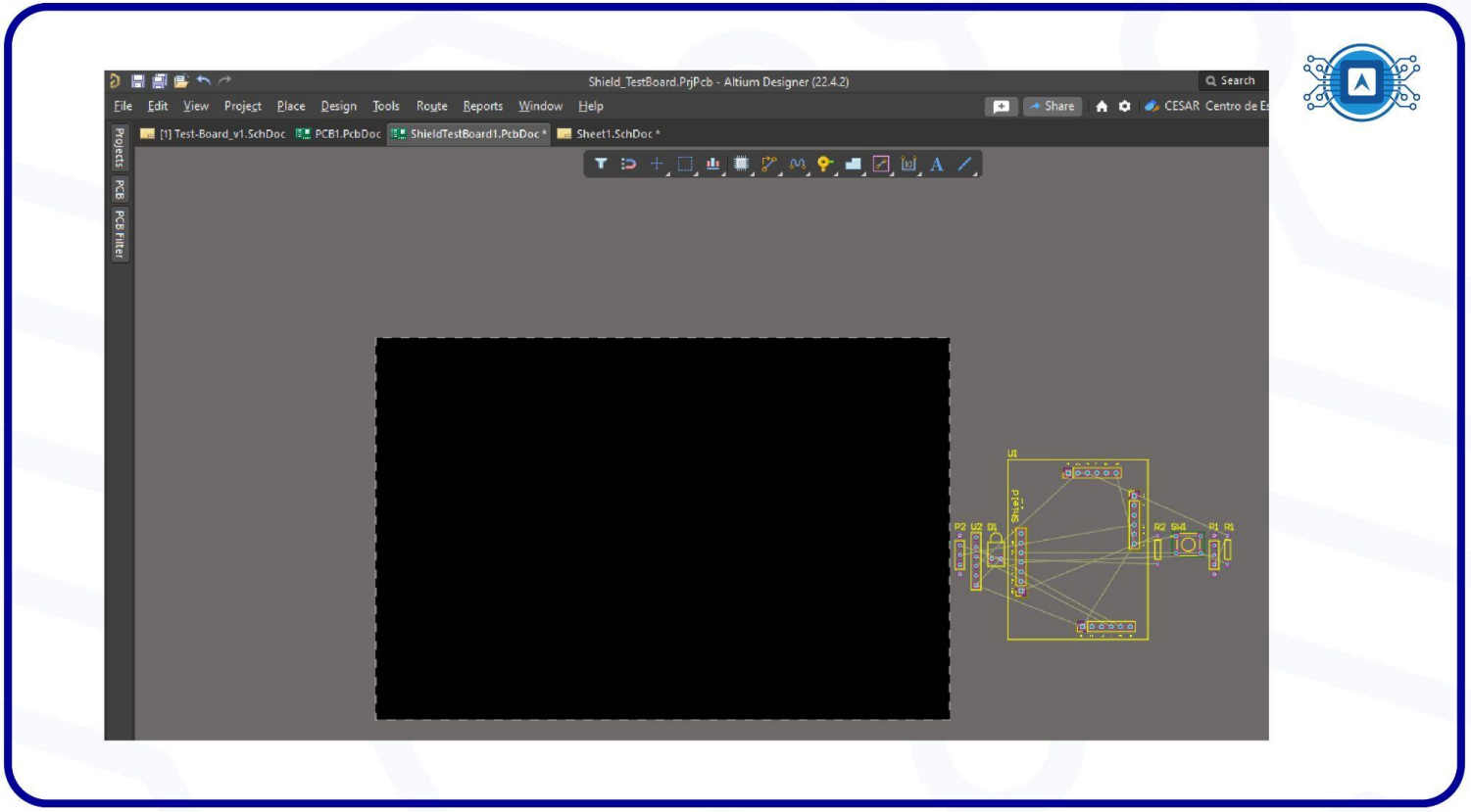

Image 10: Importing components into the PCB Layout. Source: *Screenshot by the Author.*

- Now place all the components of the board starting always with the main component, in this case the Breakout board design;
- Then insert all the other components;
- To mark out the Test Board area, use the markings on the Breakout Board design. This design will serve as a guide for the size of the board. After that, create a KEEP-OUT LAYER line;
- Using the KEEP-OUT tool we will mark the size of the plate with this line;
- Run the contour line of the board as shown in the image below:

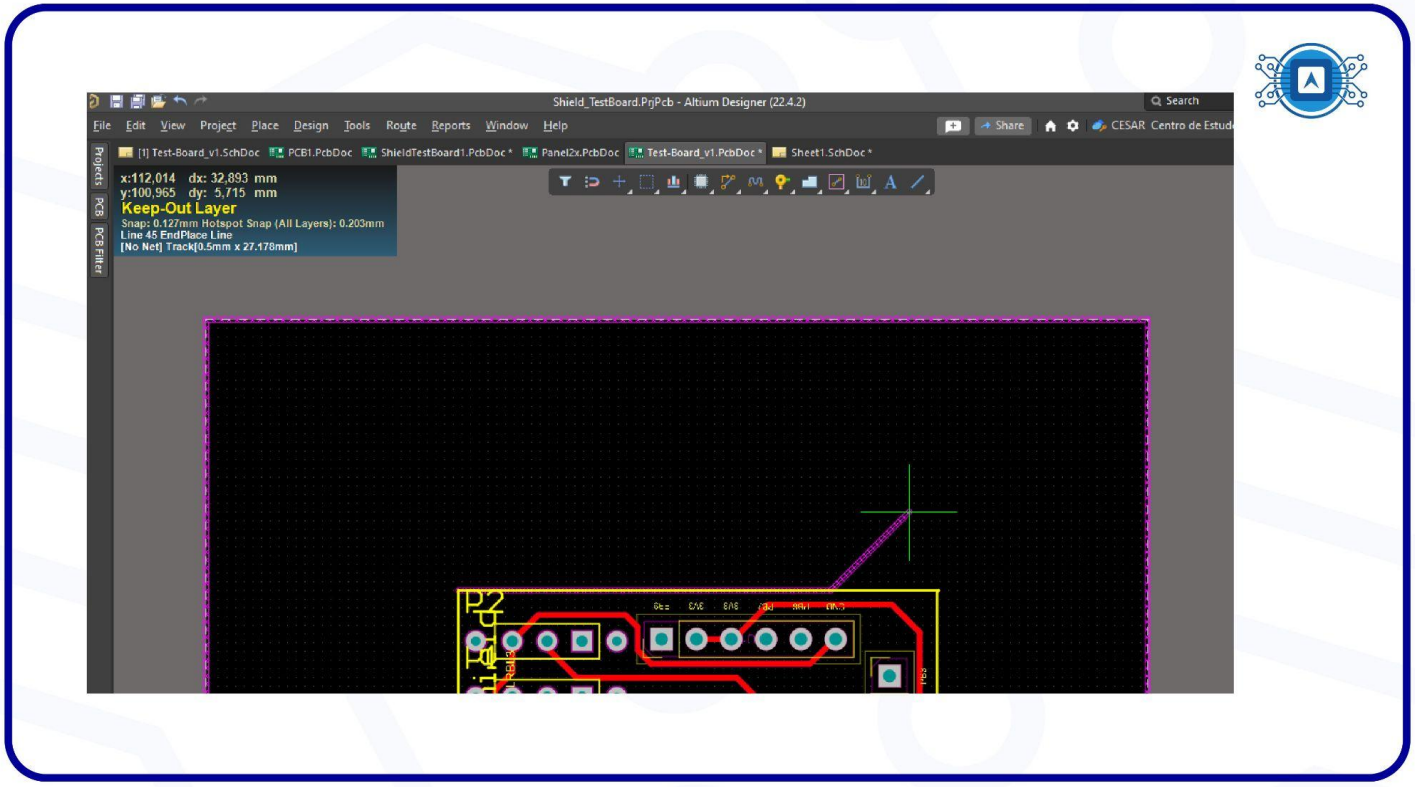

Image 11: Keep-out line design. Source: *Screenshot by the Author.*

- At this point, start the trail design, setting the width of 20 mills for this design;
- Then make the connections from the tracks to the components;
- After finishing all the tracks on the board, place a ground plane;
- This procedure of inserting a plane will help prototyping the plate, as it will decrease the amount of resources such as drills, in case it is done on a milling machine;
- To create this ground plane, insert a polygon as demonstrated in the prototyping best practices video;
- After creating the polygon, select it and change the plane property to GND;
- Next, replace this polygon on the board and then cover all the grounding tracks on the board, as shown in the image below;

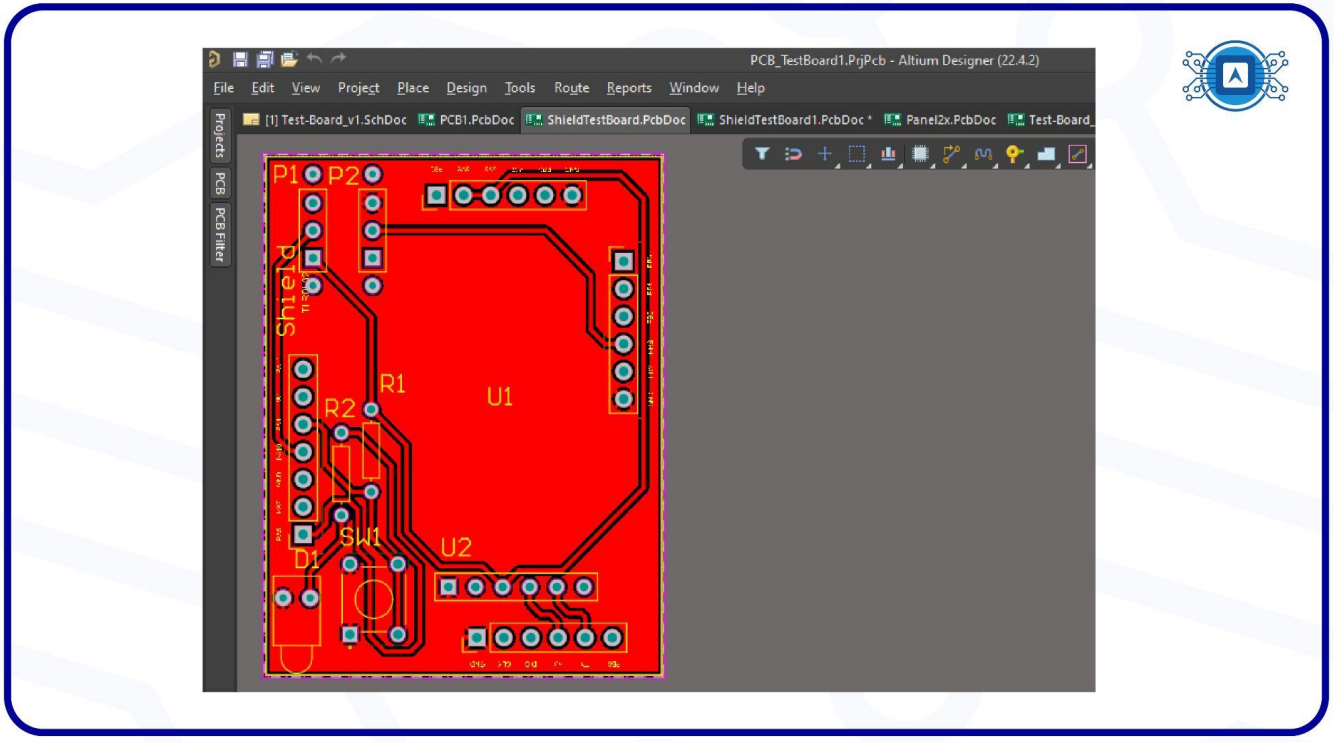

Image 12: Ground plane on PCB. Source: *Screenshot by the Author.*

- Approaching the end, work on the thermal relief. In this case, go to RULES and select the type of thermal relief you want. For example: change to 4 conductors with 45 degrees and apply it to the board;
- Then return to the board, select the polygon and reset again;
- Now preview the changes, and then finish the board;
- You can also check the plate in 3D, as in the image below. To do this, save the design and press button 3;

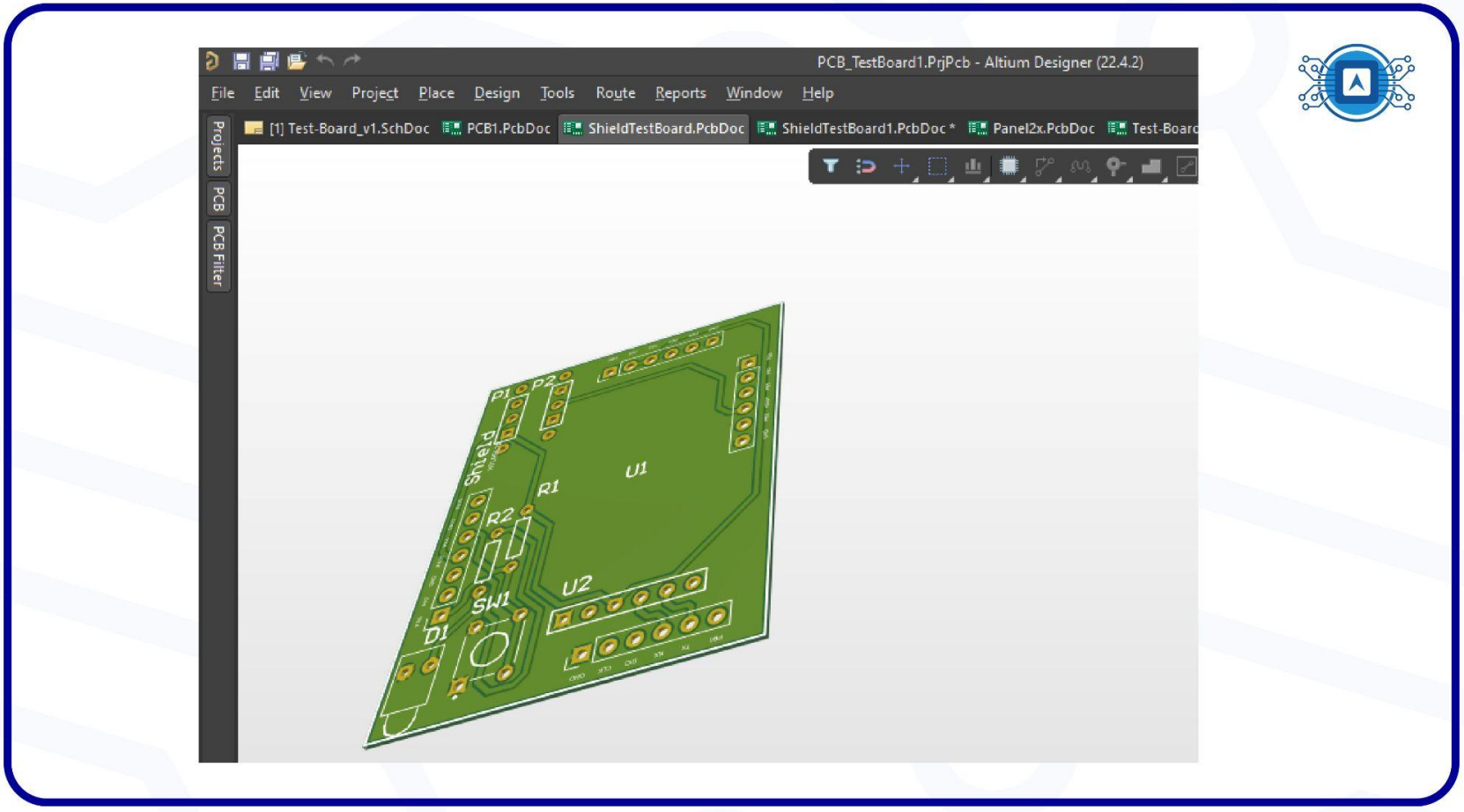

Image 13: 3D view of the PCB. Source: *Screenshot by the Author.*

Printed circuit board fabrication/assembly

There are several methods that can be used to manufacture plates, some simpler, others more complex. You can learn more about them by accessing the websites below:

Thermal process:
https://www.youtube.com/watch?v=1bJoTFhJ5CY

Process using a milling machine
https://www.youtube.com/watch?v=zRdJrE80Vjk

References

AMIT, Bahl. **High-Speed PCB Design Guide.** Sunnyvale, CA: Copyright Sierra Circuits Inc., 2020. Available at: < https://www.protoexpress.com/pcb-design-guides/high-speed-pcb/ >. Accessed on june 29th 2022.