PCB

Board size: 100.0x50.0 mm (3.94x1.97 inches)

- This is the size of the rectangle that contains the board
- Thickness: 1.6 mm (63 mils)
- Material: FR4
- Finish: None
- Layers: 12
- Copper thickness: 35 µm

Solder mask: TOP / BOTTOM

- Color: Green

Silk screen: TOP / BOTTOM

- Color: White

Stackup:

Name	Type	Color	Thickness [µm]	Material	Er	Loss tan
F.SilkS	Top Silk Screen					
F.Paste	Top Solder Paste					
F.Mask	Top Solder Mask		10			
F.Cu	copper		35			
dielectric 1	prepreg		100	FR4	4.5	0.020
In1.Cu	copper		35			
dielectric 2	core		112	FR4	4.5	0.020
In2.Cu	copper		35			
dielectric 3	prepreg		100	FR4	4.5	0.020
In3.Cu	copper		35			
dielectric 4	core		112	FR4	4.5	0.020
In4.Cu	copper		35			
dielectric 5	prepreg		100	FR4	4.5	0.020
In5.Cu	copper		35			
dielectric 6	core		112	FR4	4.5	0.020
In6.Cu	copper		35			
dielectric 7	prepreg		100	FR4	4.5	0.020
In7.Cu	copper		35			
dielectric 8	core		112	FR4	4.5	0.020
In8.Cu	copper		35			
dielectric 9	prepreg		100	FR4	4.5	0.020
In9.Cu	copper		35			

Name	Type	Color	Thickness [µm]	Material	Er	Loss tan
dielectric 10	core		112	FR4	4.5	0.020
In10.Cu	copper		35			
dielectric 11	prepreg		100	FR4	4.5	0.020
B.Cu	copper		35			
B.Mask	Bottom Solder Mask		10			
B.Paste	Bottom Solder Paste					
B.SilkS	Bottom Silk Screen					

Important sizes

Clearance: 0.2 mm (8 mils)

Track width: 0.4 mm (16 mils)

- By design rules: 0.0 mm (0 mils)

Drill: 0.9 mm (35 mils)

- Vias: N/A mm (N/A mils) [Design: 0.4 mm (16 mils)]
- Pads: 0.9 mm (35 mils)
- The above values are real drill sizes, they add 0.1 mm (4 mils) to plated holes (PTH)

Via: N/A/N/A mm (N/A/N/A mils)

- By design rules: 0.5/0.3 mm (20/12 mils)
- Micro via: yes [0.2/0.1 mm (8/4 mils)]
- Buried/blind via: yes
- Total: 0 (thru: 0 buried/blind: 0 micro: 0)

Outer Annular Ring: 0.22 mm (9 mils)

- By design rules: 0.22 mm (9 mils)

Eurocircuits class: 4A - Using min drill 0.9 mm for an OAR of 0.22 mm

General stats

Components count: (SMD/THT)

- Top: 0/12 (THT)
- Bottom: 0/0 (NONE)

Defined tracks:

- 0.4 mm (16 mils)

Used tracks:

- 0.4 mm (16 mils) (61) defined: yes

Defined vias:

Used vias:

Holes (excluding vias):

- 0.8 mm (31 mils) (4)
- 0.9 mm (35 mils) (2)
- 1.0 mm (39 mils) (14)
- 1.02 mm (40 mils) (30)
- 1.1 mm (43 mils) (7)
- 1.19 mm (47 mils) (2)
- 1.2 mm (47 mils) (4)
- 1.8 mm (71 mils) (4)

Oval holes:

Drill tools (including vias and computing adjusts and rounding):

- 0.9 mm (35 mils) (4)
- 1.0 mm (39 mils) (2)
- 1.1 mm (43 mils) (44)
- 1.2 mm (47 mils) (7)
- 1.3 mm (51 mils) (6)
- 1.8 mm (71 mils) (4)

Solder paste stats:

Using a paste with 87.75 % alloy, that has an specific gravity for the alloy of 7.4 g/cm^3 and 1.0 g/cm^3 for the flux. This paste has an specific gravity of 4.15 g/cm^3.

The stencil thickness is 0.12 mm.

Side	Pads with paste	Area [mm^2]	Paste [g]
Total	0	0.00	0.00

Note: this is just an approximation to the theoretical value. Margins of the solder mask and waste aren't computed.

Schematic

Schematic in SVG format

PCB Layers

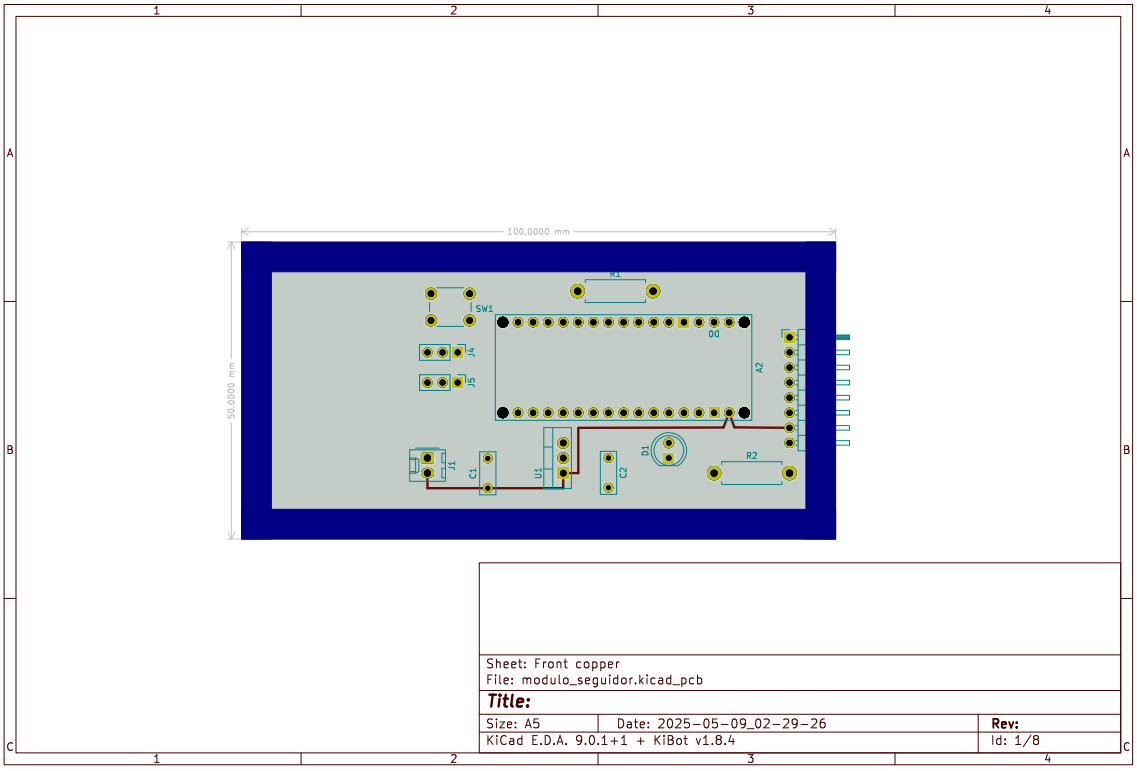

Figure 1: PCB Front copper

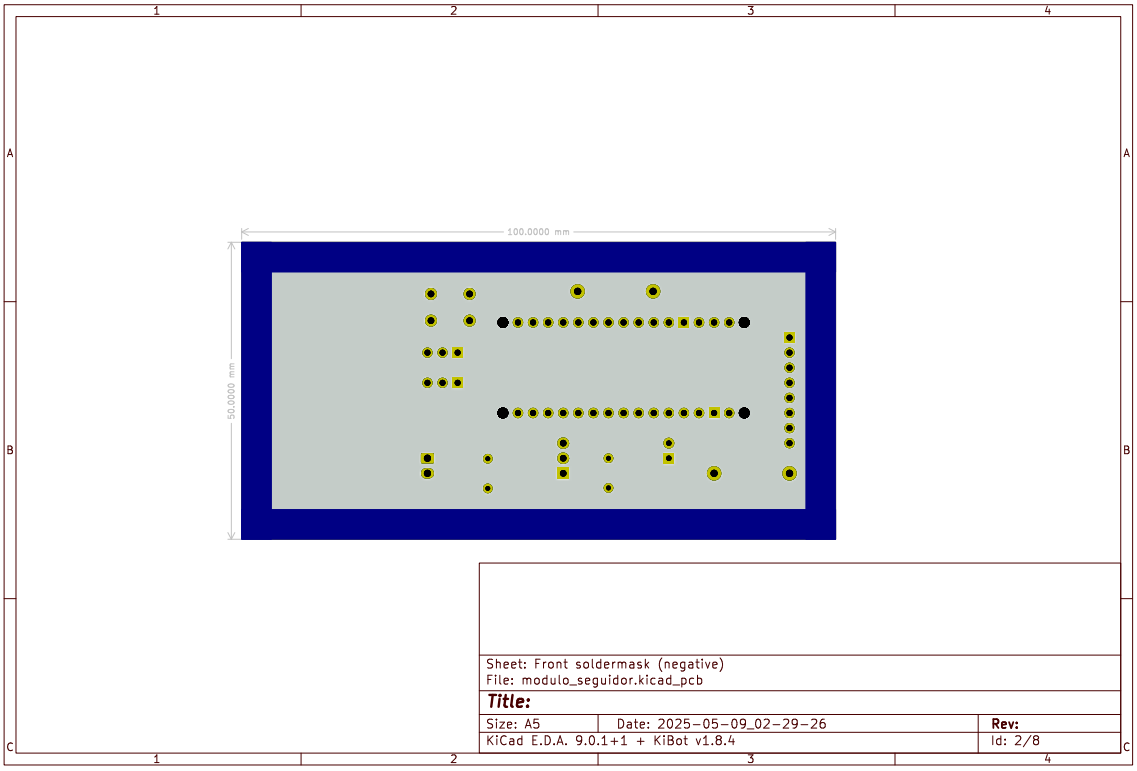

Figure 2: PCB Front soldermask (negative)

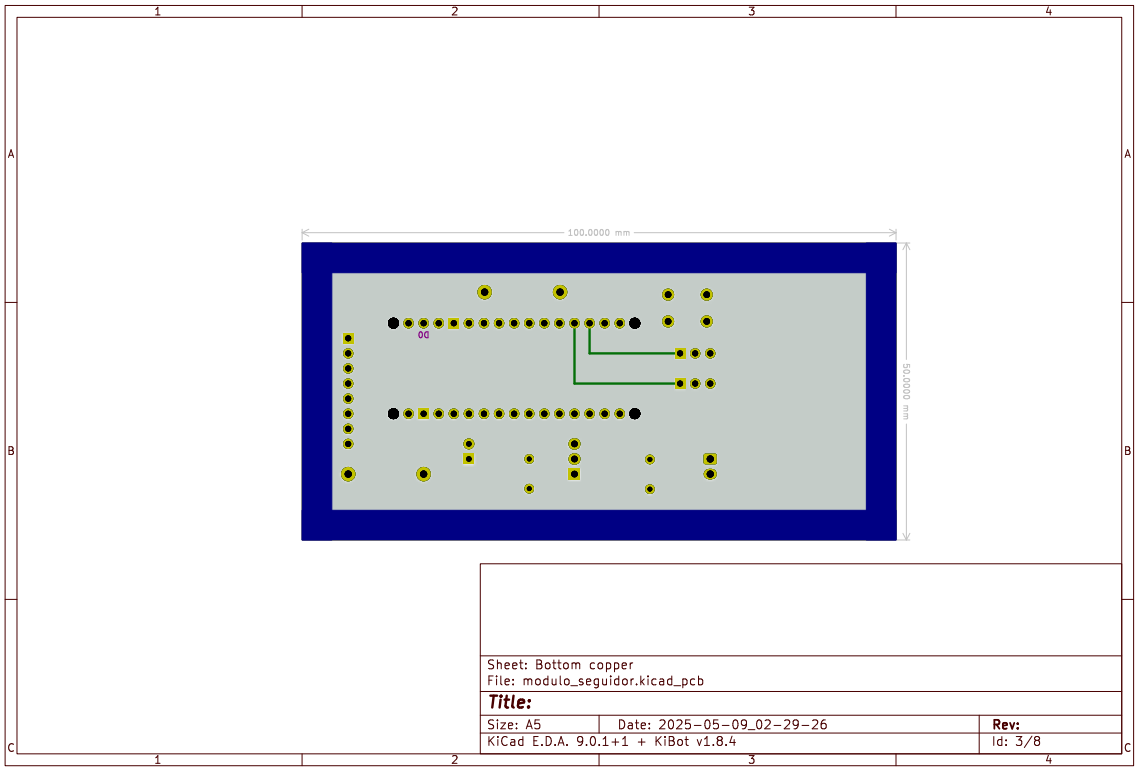

Figure 3: PCB Bottom copper

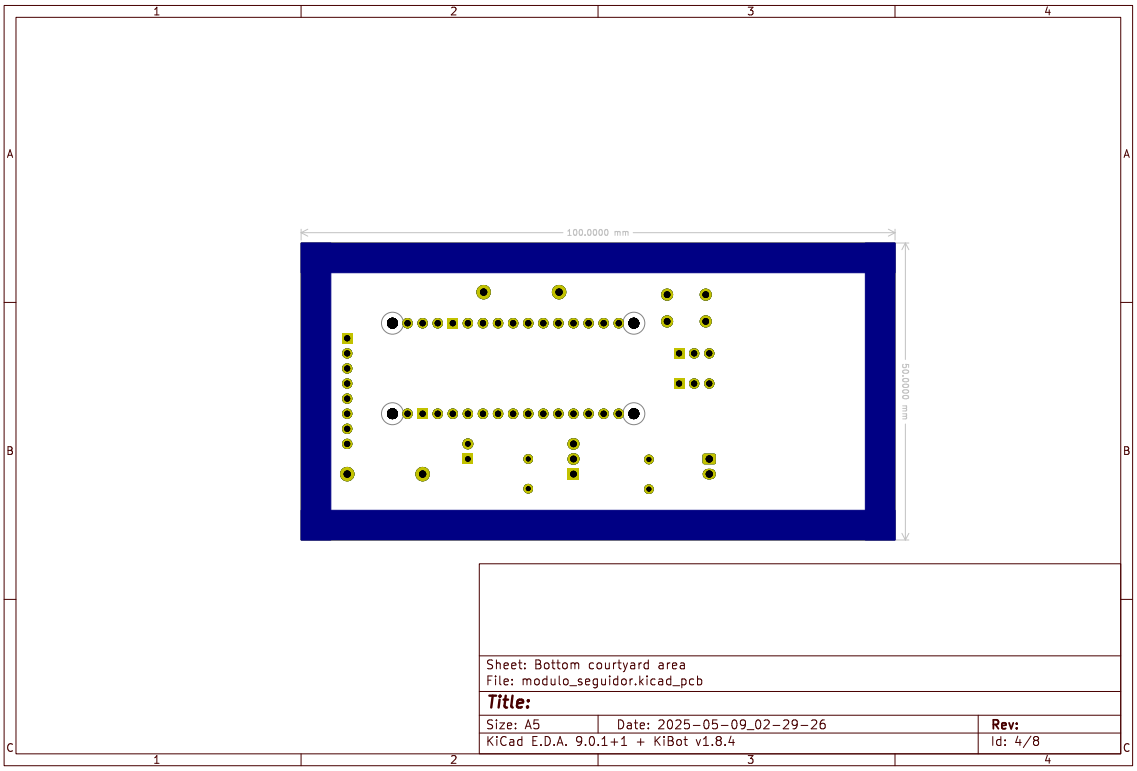

Figure 4: PCB Bottom courtyard area

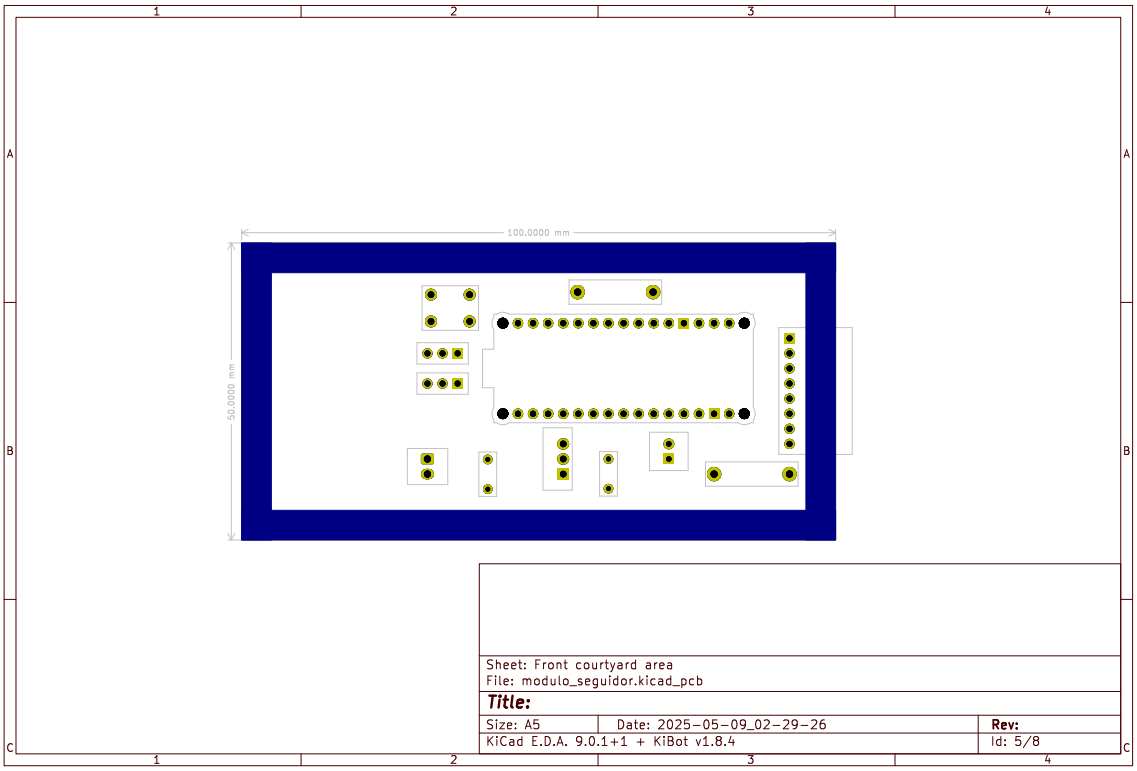

Figure 5: PCB Front courtyard area

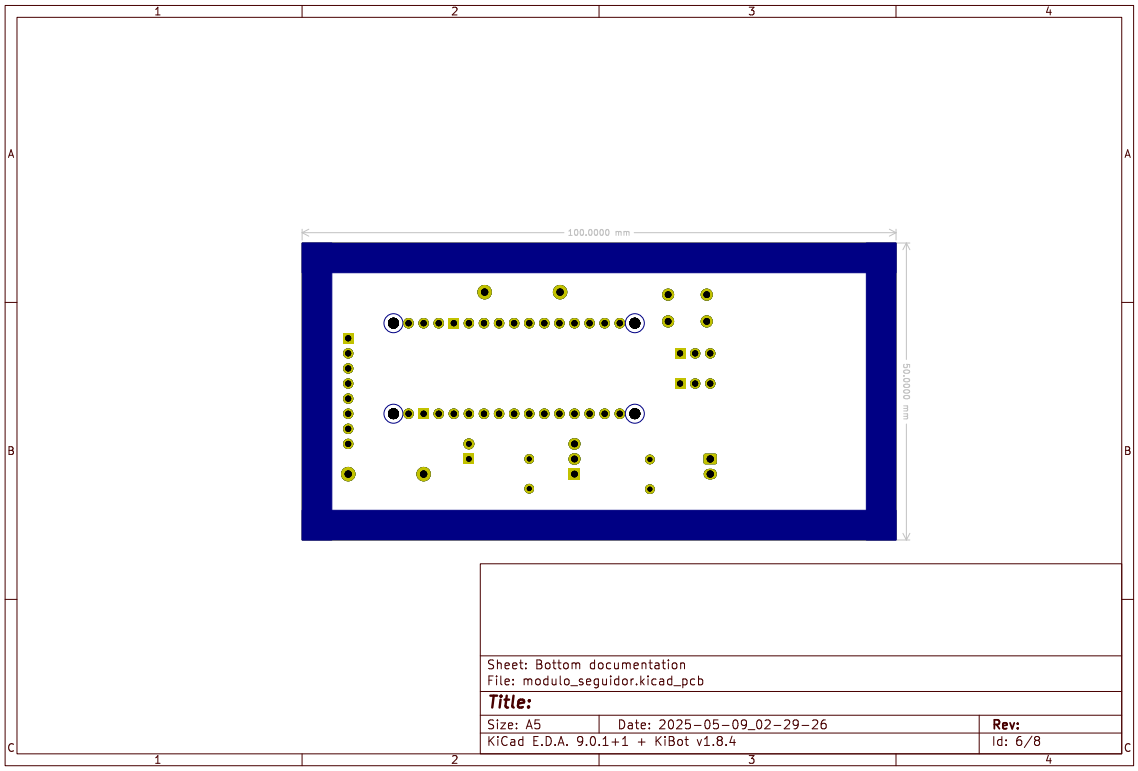

Figure 6: PCB Bottom documentation

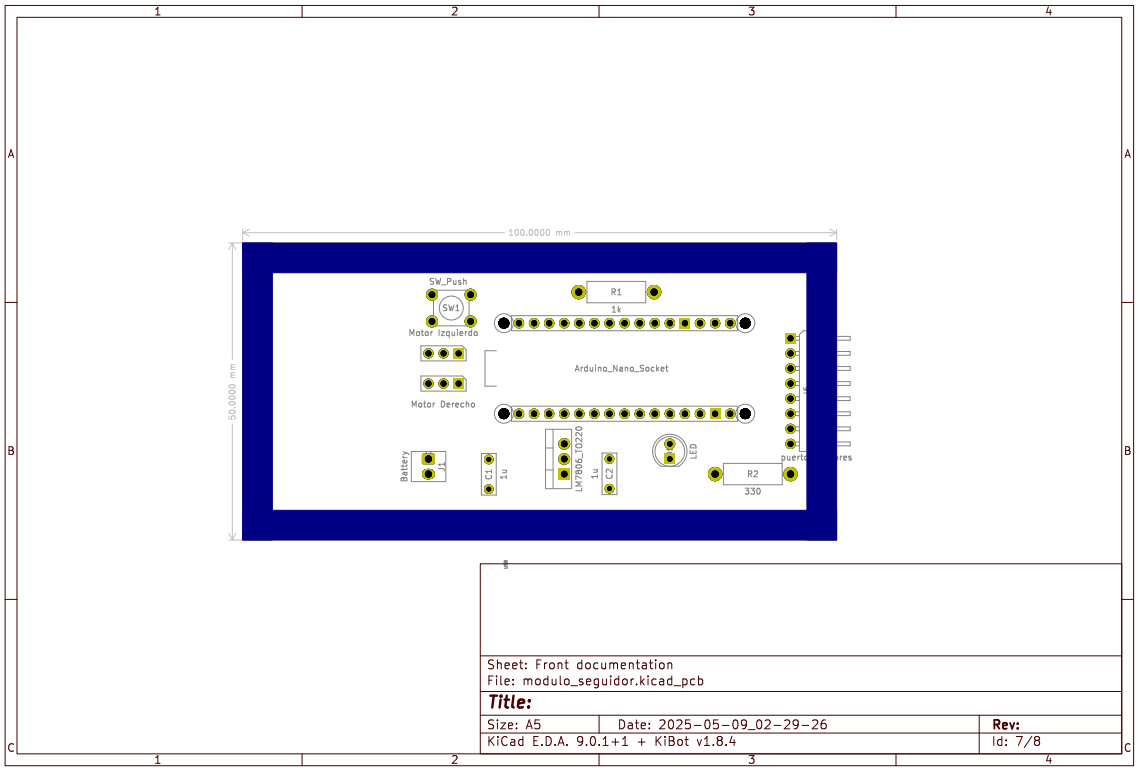

Figure 7: PCB Front documentation

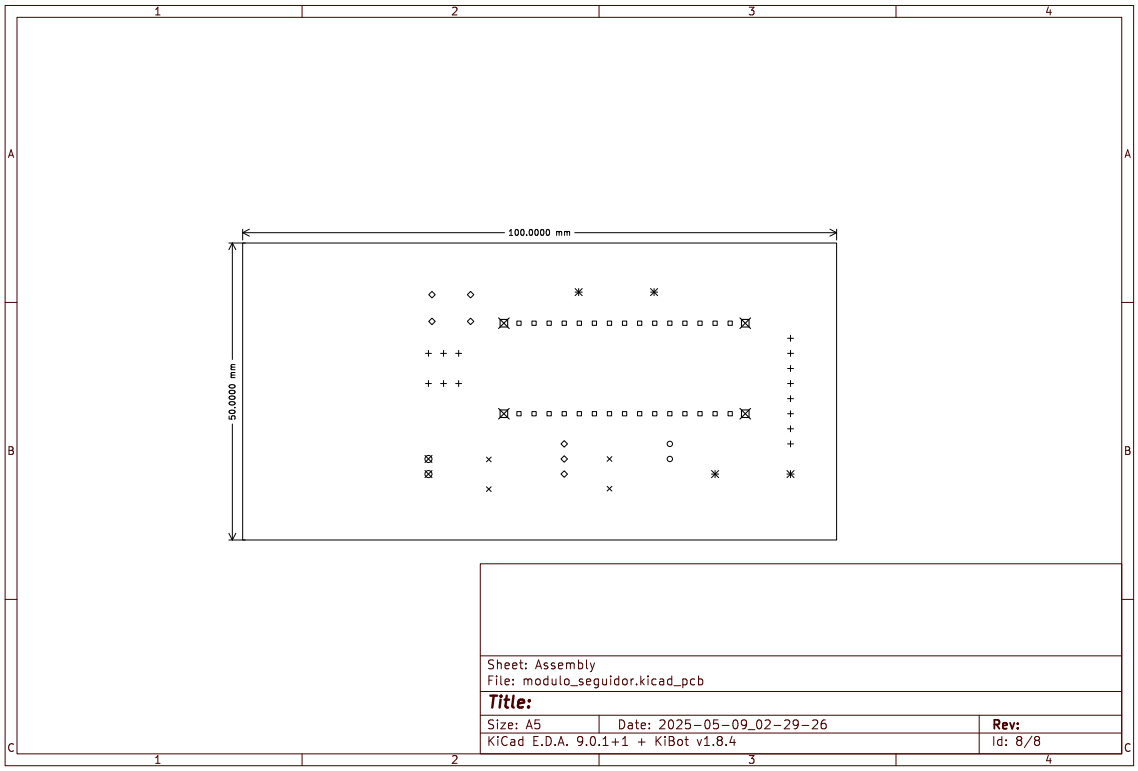

Figure 8: PCB Assembly