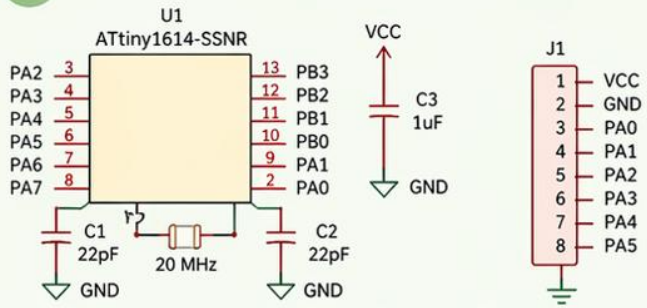
PCB DESIGN PROCESS (Up to Step 6)

1 PROJECT DEFINITION

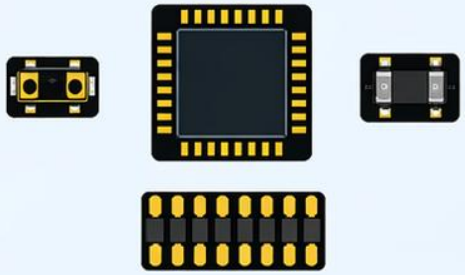
- Define the objective of my board.
- Choose the main microcontroller (ATtiny1614).
- Identify the inputs, outputs and required components.

2 SCHEMATIC DESIGN IN KICAD



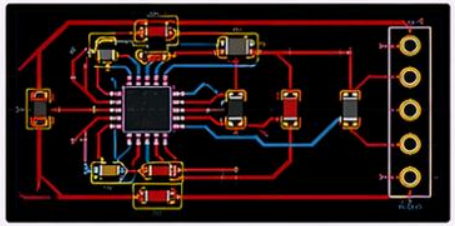
- Design the electrical schematic including: microcontroller, resonator, regulator, capacitors and connectors.

3 FOOTPRINT ASSIGNMENT



- Assign footprints to all components according to the schematic.

4 PCB LAYOUT DESIGN

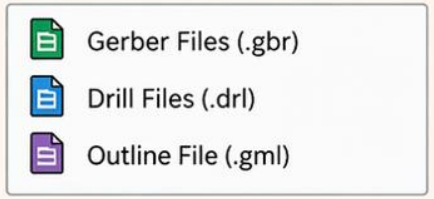


- Route the PCB in KiCad, configuring track widths and applying design rules.

5 DESIGN RULE CHECK (DRC)

- Verify that there are no electrical errors or design rule violations.

6 GENERATE FABRICATION FILES



- Generate Gerber, drill and outline files for PCB fabrication.