In this guide, you will learn how to use KiCAD to design the layout for a simple BJT (Bipolar Junction Transistor) amplifier circuit.

Step-by-Step Guide

1. Install and Open KiCAD:

- Download and install KiCAD from https://www.kicad.org/.
- Open KiCAD and create a new project by clicking on "File" → "New Project".
- Name your project, choose a folder, and click Save.

2. Schematic Design:

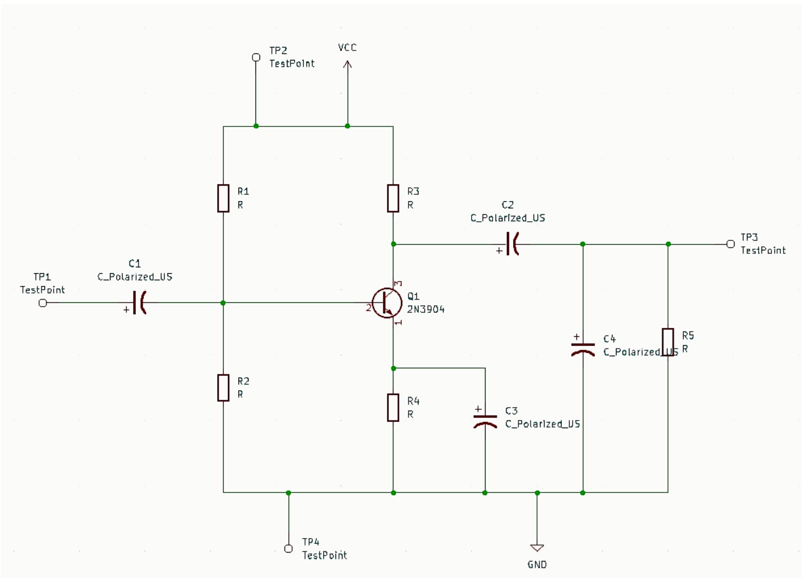


Figure 1: Example of small signal amplifier circuit

To draw the schematic design as shown in Figure 1:

Step 2.1: Create a New Schematic

- Click on "Schematic Layout Editor" from the project window.

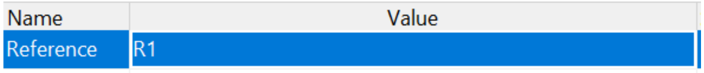
Step 2.2: Add Components

- Click "Place" → "Add Symbol" button or click icon  or press the `A` key to add components.

- Search for and add the following components:

  - NPN Transistor (BJT): For example, a 2N3904.
  - Resistors: Typically, R1 and R2 for voltage divider, R3 for collector resistor, R4 for emitter resistor, and R5 for load resistor.
  - Polarized Capacitors: For coupling (C1 at the input and C2 at the output), for bypass (C3), for external parallel capacitor (C4)
  - Power Supply (VCC) and Ground (GND): Add testpoint for the DC voltage supply and circuit ground.
  - Input, Output connectors: Add testpoint for the input signal and output.

- Double click on each component to rename.



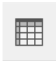
Step 2.3: Connect Components

- Click "Place" → "Wire" tool (shortcut: `W`) or click icon  to connect the components as per the small signal amplifier circuit.

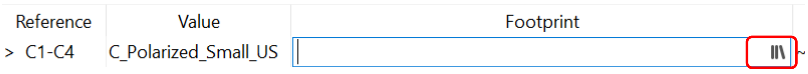
Step 2.4: Annotate Components

- After placing all components, click on "Tools" → "Annotate Schematic" to assign reference designators (R1, C1, Q1, etc.).

Step 2.5: Assign Footprints

- Click on "Tools" → "Edit Symbol Fields" or icon  to assign the physical footprints to each component.

- Browse footprint library. Example:



- Select footprints for resistors, capacitors, and the BJT as shown in Table 1

Table 1

Components		Type
Capacitors	Capacitor_THT	CP_Radial_D5.0mm_P2.50mm
Vi, Vo, Vcc and Gnd	TestPoint	TestPoint_Loop_D2.50mm_Drill1.0mm
NPN Transistor	2N3904	Package_TO_SOT_THT:TO-92L_Inline_Wide
Resistors	Resistor_THT	R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal

Step 2.6: Electrical Rule Check (ERC)

- Run an ERC by clicking "Inspect" → "Electrical Rules Checker" to verify there are no errors or warnings. Ignore if errors appeared are: Input Power pin not driven by any Output Power pins

3. PCB Layout Design:

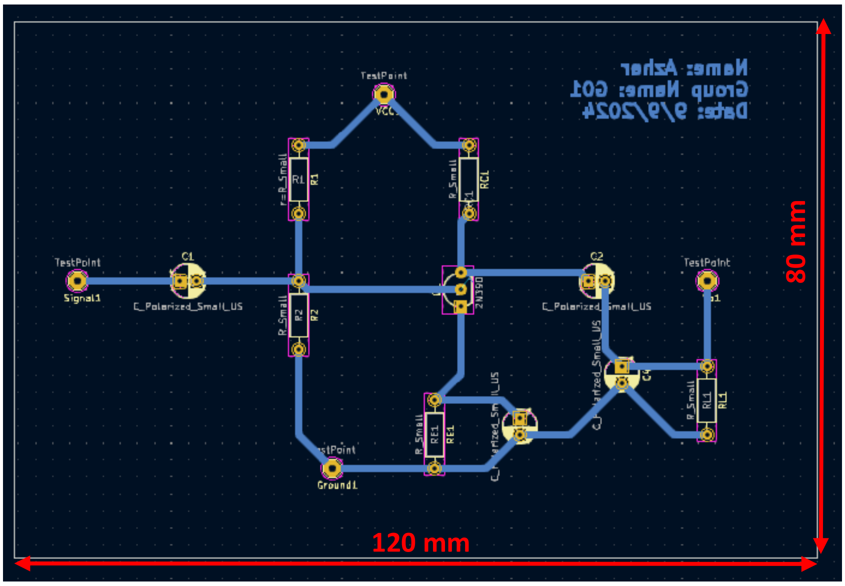
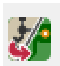


Figure 2: Example of PCB layout design for small signal amplifier circuit

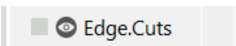
To design the PCB layout of the small signal amplifier circuit:

Step 3.1: Open PCB Layout Editor 

- Return to the main KiCAD window and click "PCB Editor".

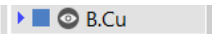
- Click on "Tools" → "Update PCB from Schematic" or icon 
- The components from the schematic will appear in the layout editor.

Step 3.2: Create PCB layout size

- Select Edge.Cuts layer 

and click icon  to create PCB layout size (120 mm x 80 mm)

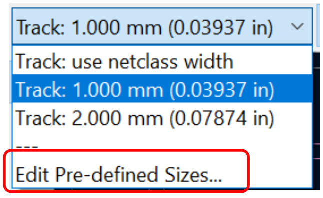
Step 3.3: Arrange Components

- Choose bottom Copper layer 

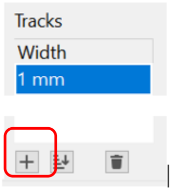
- Drag the components into the desired positions within the PCB size.
- Press R to rotate the components if needed
- Place components close to where they will be connected.
- Keep power connections (VCC and GND) in logical positions.
- Minimize the distance between related components (e.g., resistors, capacitors, and the BJT).

Step 3.4: Change route tracks size

- Click on Edit Pre-defined Sizes to create any route tracks size



- Click “+” button to add 1 mm track size



Step 3.5: Route the PCB

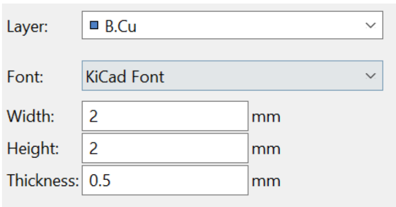
- Use the "Route Tracks" tool (shortcut: `X`) or icon  to manually connect the pads of the components using 1 mm track size.
- Ensure all connections are made as per the schematic.
- Keep traces short and direct to avoid unnecessary inductance and interference.

Step 3.6: Run Design Rules Check (DRC)

- Run the "Design Rules Checker" by clicking icon  to ensure there are no errors in the PCB layout.
- If any issues arise, resolve them before proceeding.

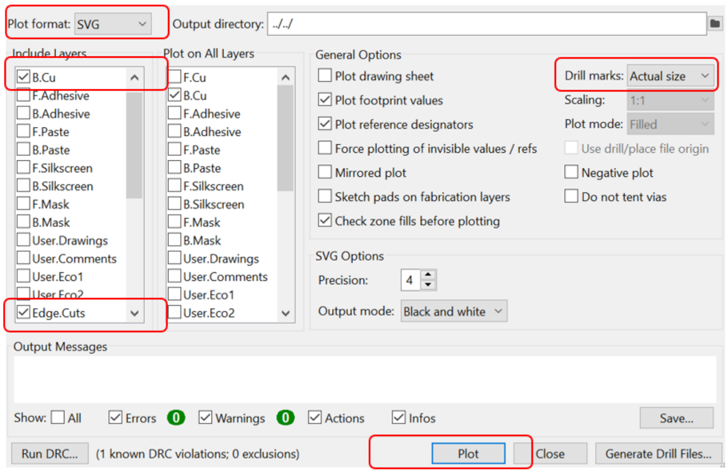
Step 3.7: Name the layout design

- Click icon  to add Name (only 1 representative name), Group Number, Lab Date) with specification as listed.



Step 3.8: Generate Printed Files

- Export SVG format file. Click “File” → “Fabrications Output” → “Gerbers”.
- Select B.Cu and Edge.Cuts layers and make sure Drill marks is Actual size. Then click Plot button



- Print 4 PCB layout design in single A4 size as shown in Figure 3. Hint: edit SVG file using open source application inkscape (https://inkscape.org/) or adobe illustrator to paste 4 PCB layout in single A4 size.

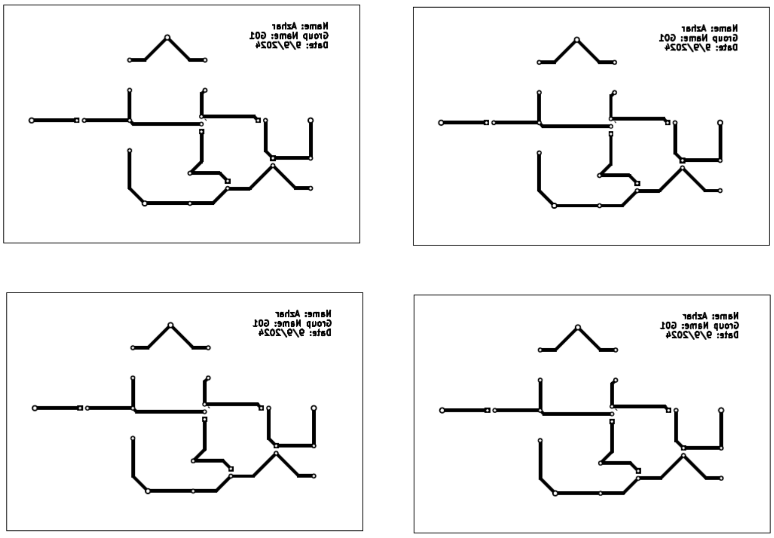


Figure 3: 4 PCB layout design in single A4 size

4. Final Review and Save:

- Review the design, both schematic and layout, to ensure everything is correct.
- Save your project by clicking "File" → "Save".

5. Print the PCB layouts on A4 paper and show it to your supervisor for verification purpose.