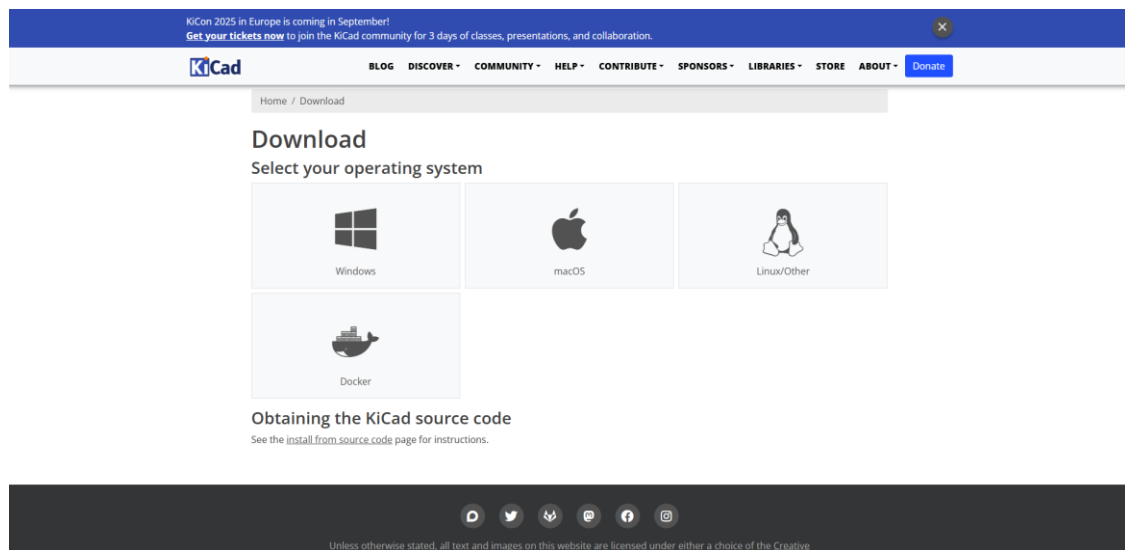# VISITING CARD_kicad

A practical, step-by-step user manual that walks you from a blank project to production-ready Gerbers using KiCad. Intended for beginners and intermediate users who want a clean, repeatable PCB design workflow.
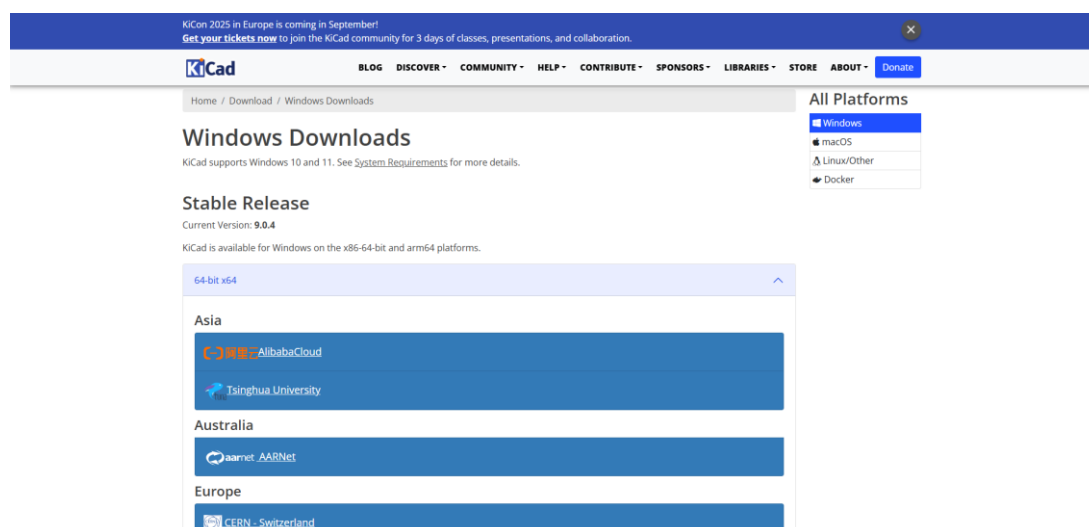
How to download the Kicad Software:

Step 1: Go to the official KiCad website-**Link**
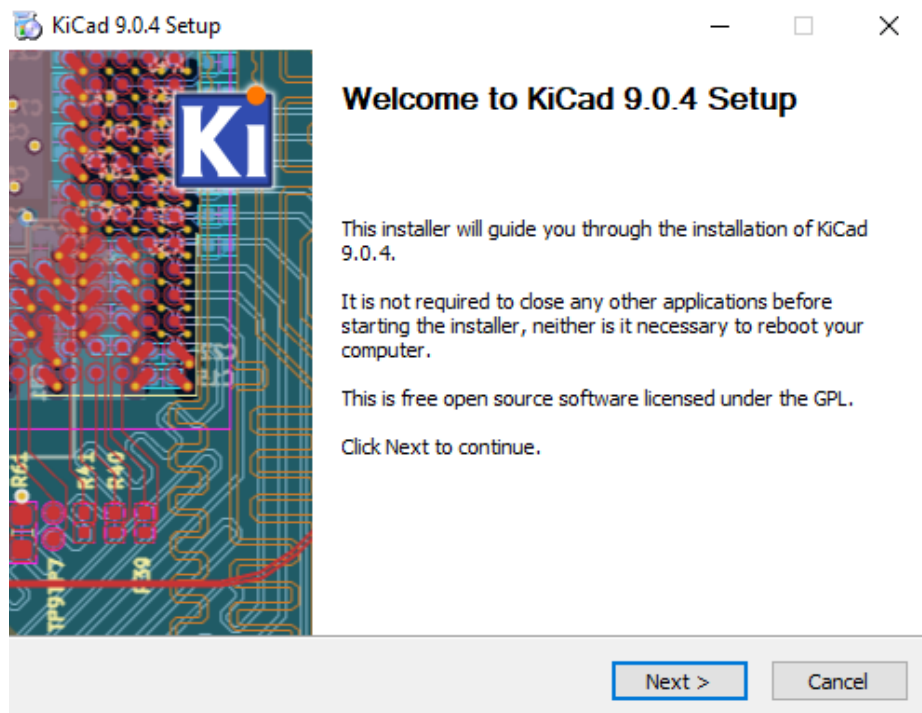

Step 2: Choose your Operating System

- 'Windows' → Download the installer (.exe)
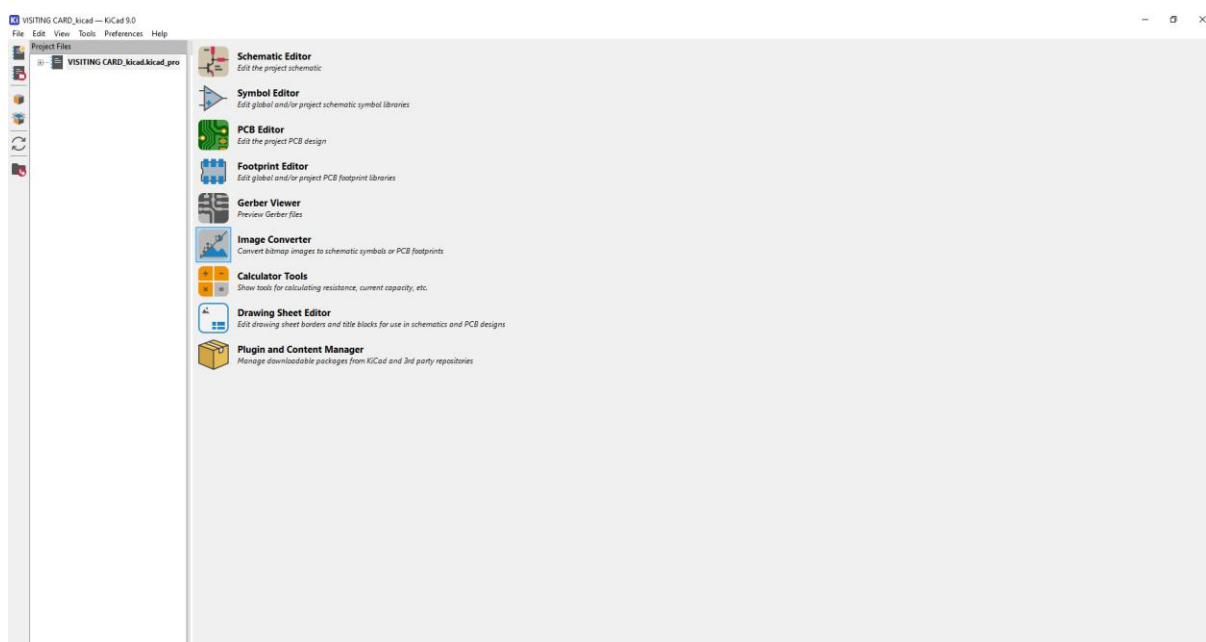- To select the Asia version.

Step 3: Install

- Run the downloaded installer and follow the setup wizard.

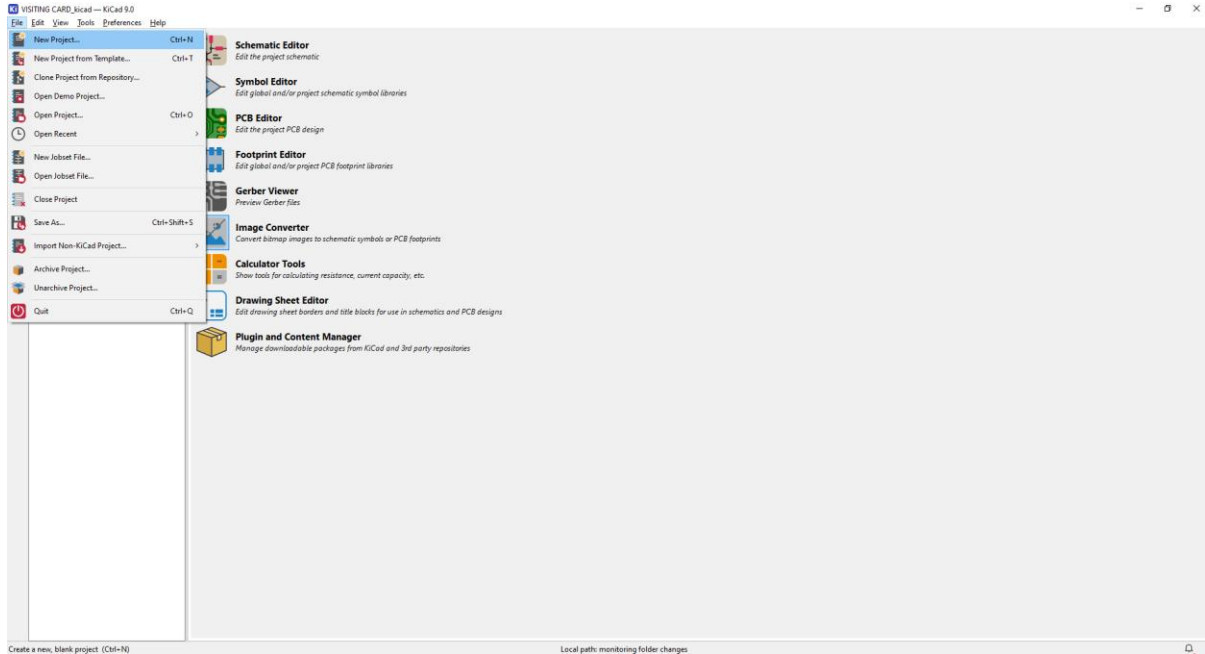

Step 4: Verify installation

- Open KiCad from your applications menu.
- The KiCad Project Manager window should appear, showing buttons for Schematic Editor and PCB Editor.
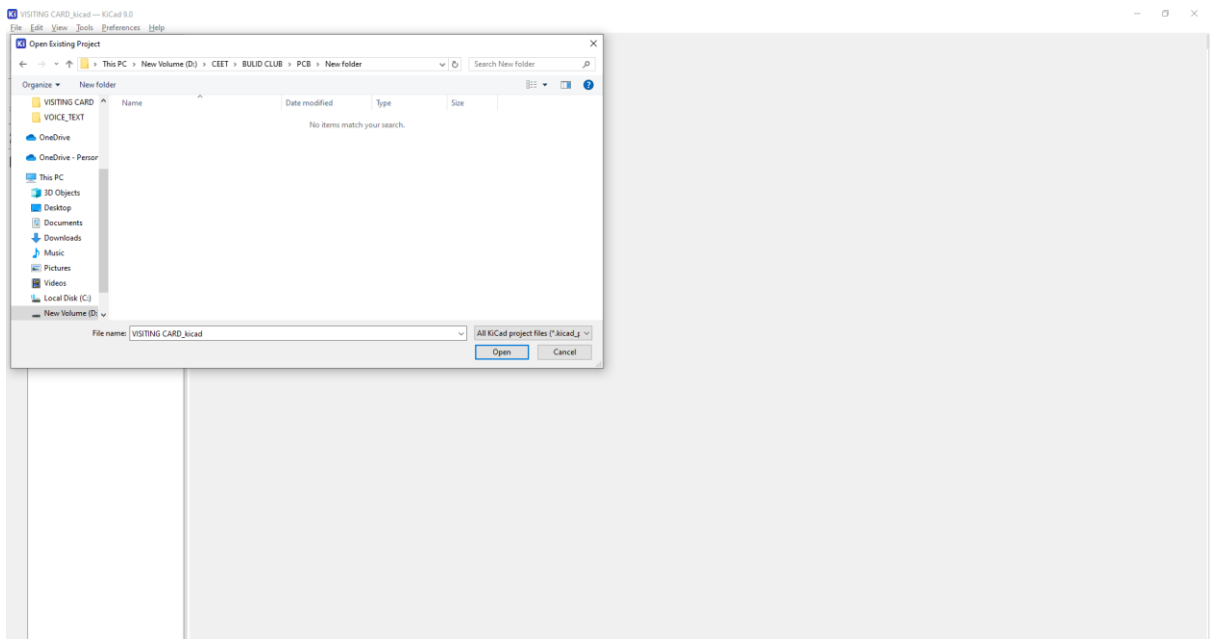
Start a project:

Step1:

- File → New Project → give a clear name
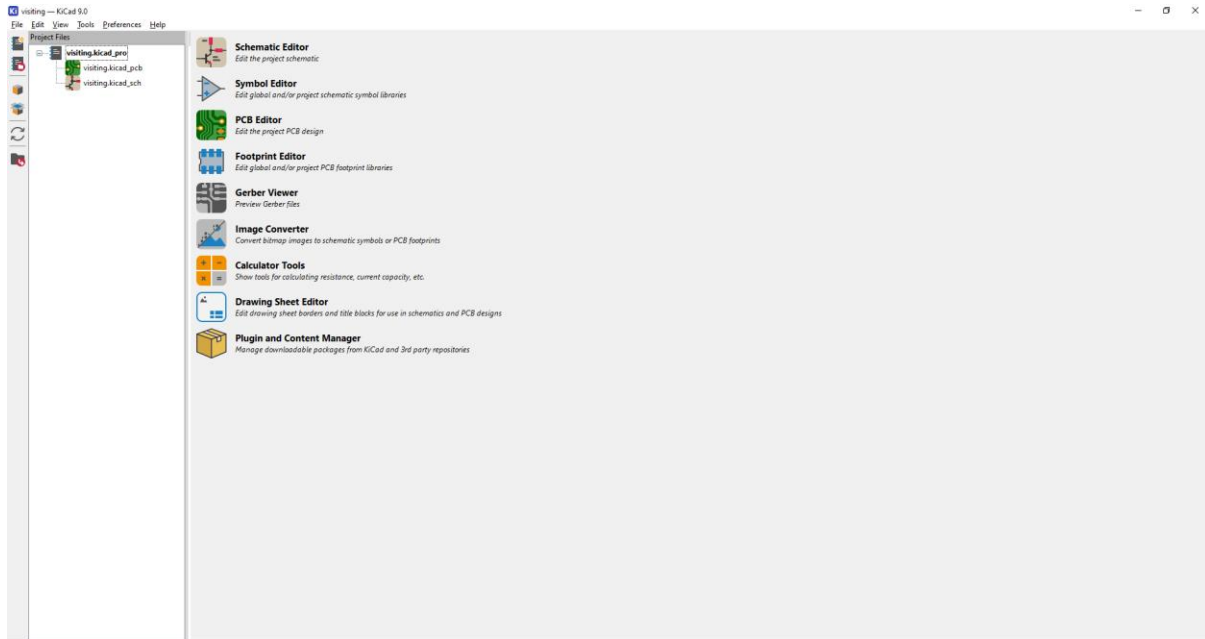

Step2:

- Enter the project title
- Save in a folder structure.

Step 3:
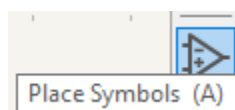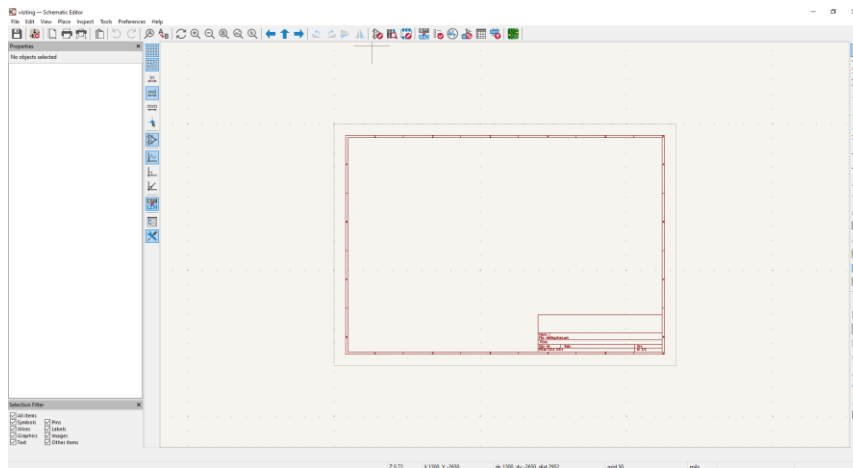
- Open the schematic editor file

Step 4:

- Select all components list

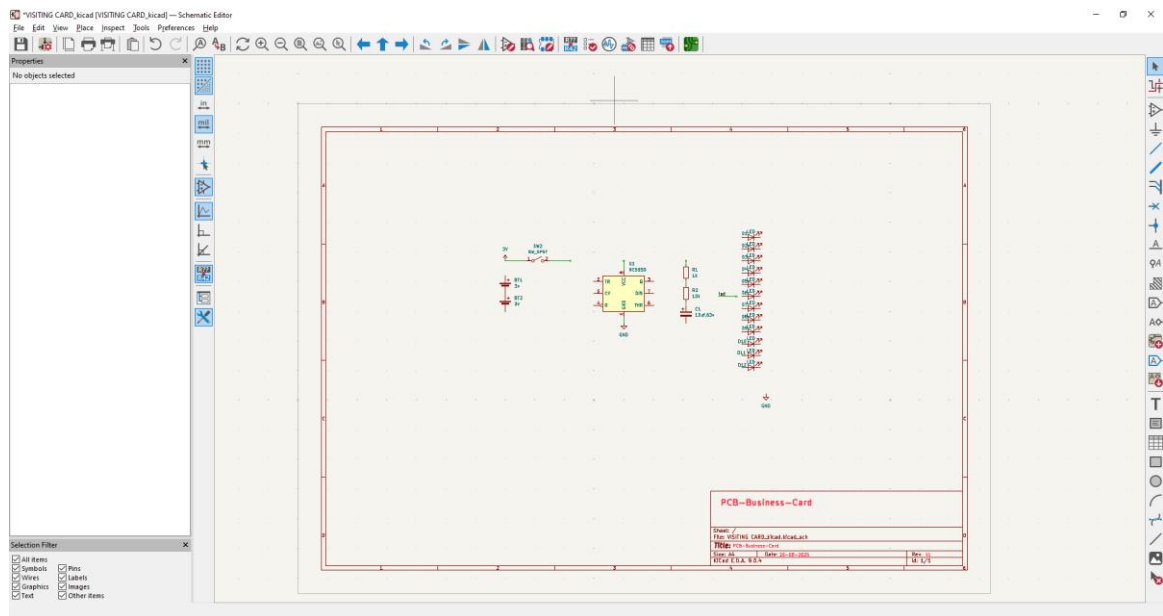| S. No	Components	Quantity
1.	NE555D	1
2.	SW SPST	1
3.	10uF 63V Capacitor DIP	1
4.	1k Ohm 1/4W 0603	2
5.	LED 0805	12
6.	Cell Holder	2

- Click the Place Symbols ion,

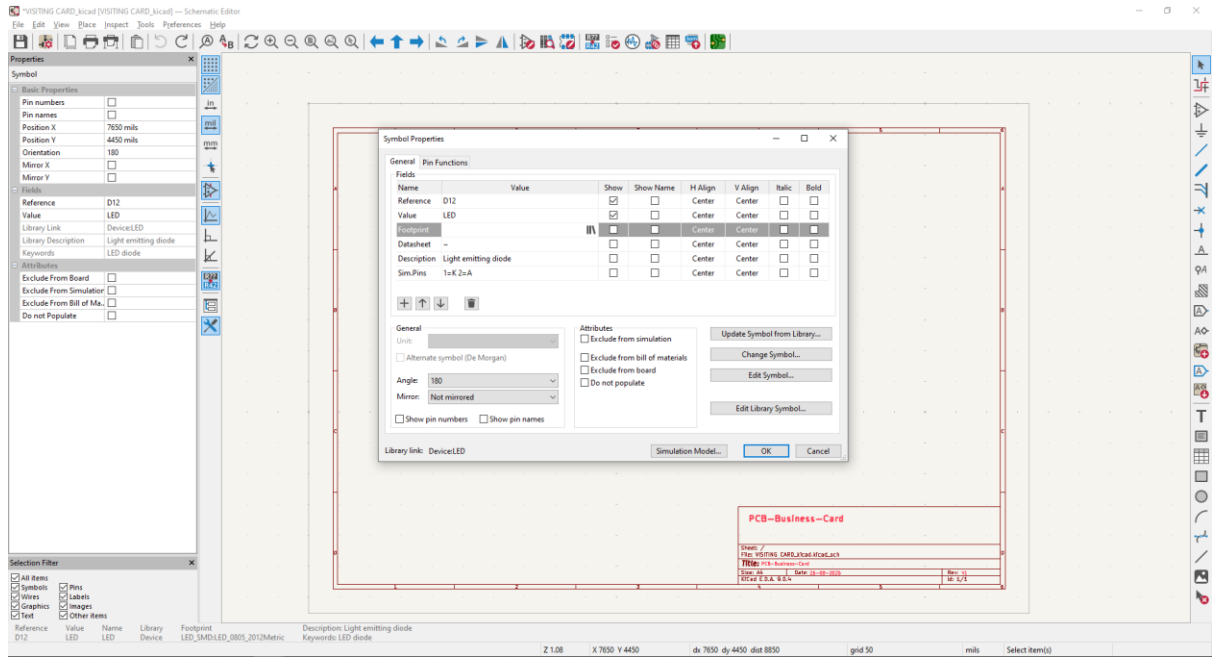- On your right-side top corner / press 'A' key word

Step 5:

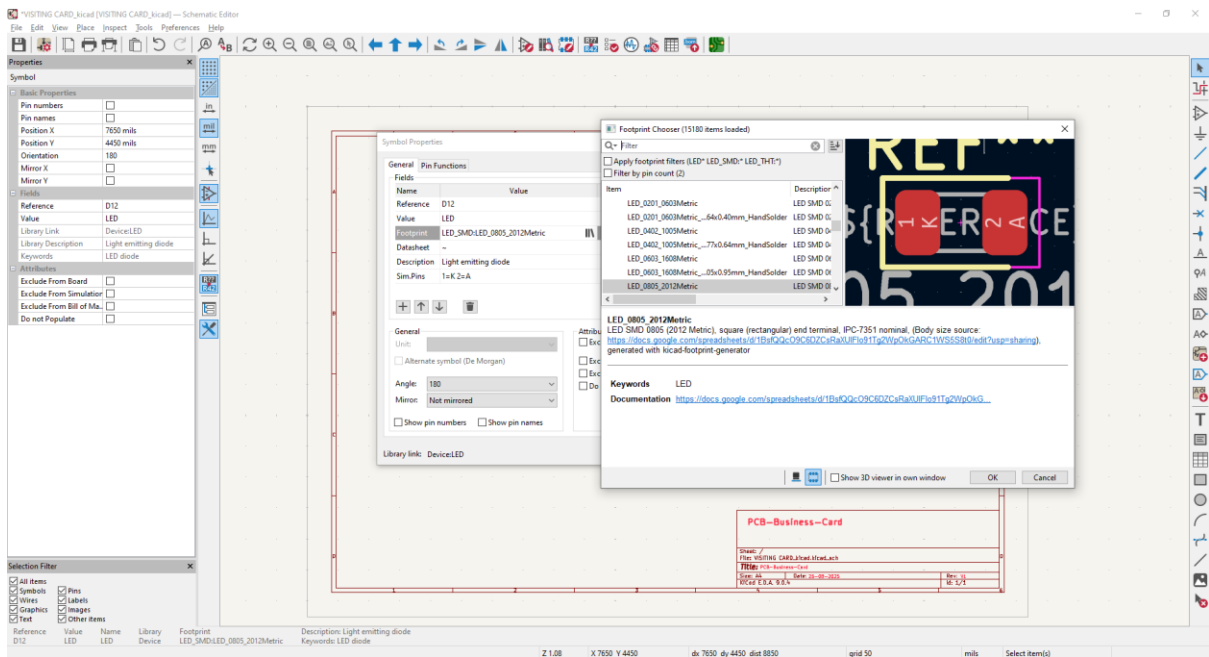- To place all components on the sheet

- And import the footprint for all Components
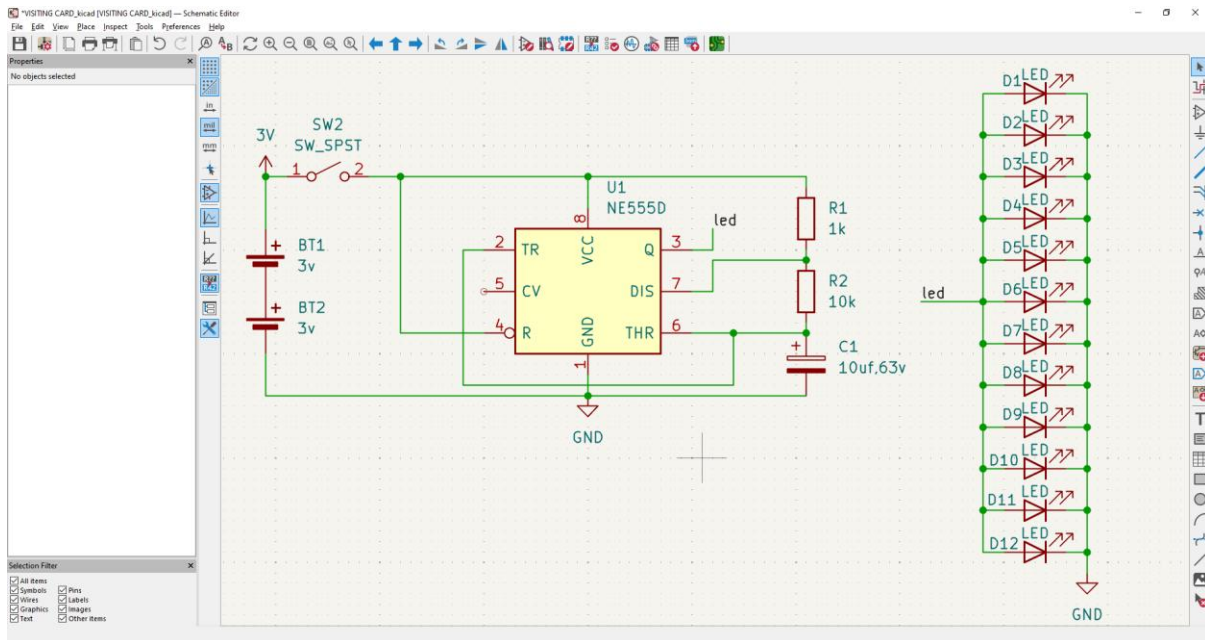- Select the component press 'E' add the footprint

Step 6:

- Select the part number (like a package-0603)

- Connect symbols with wires
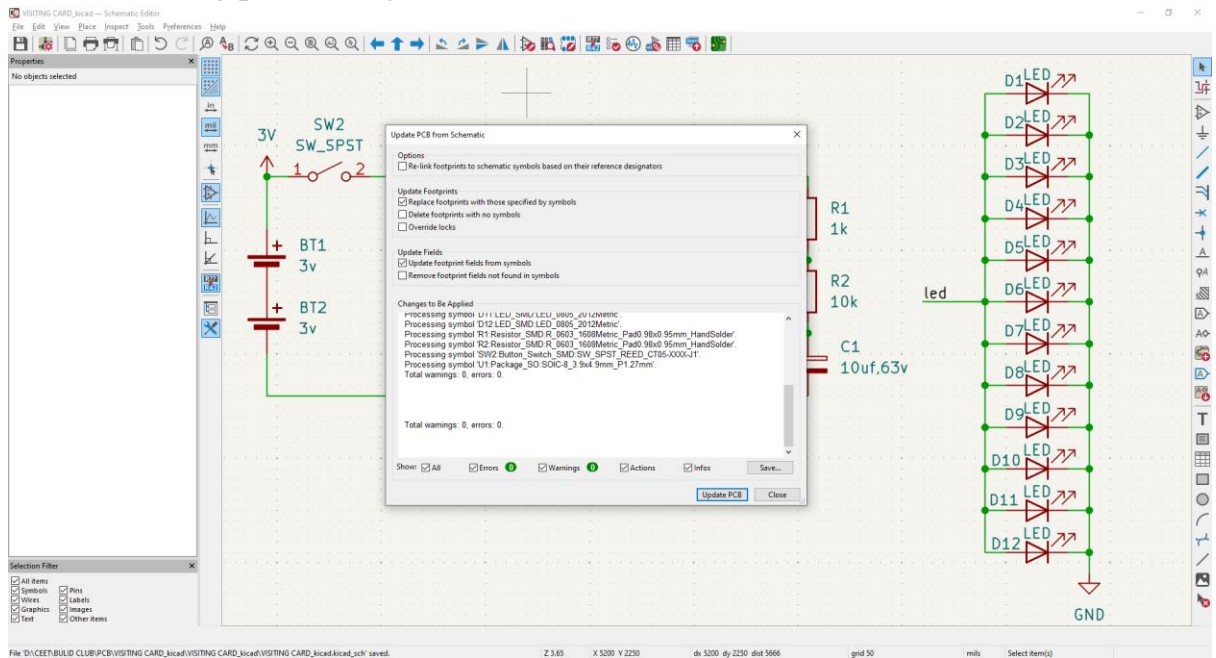
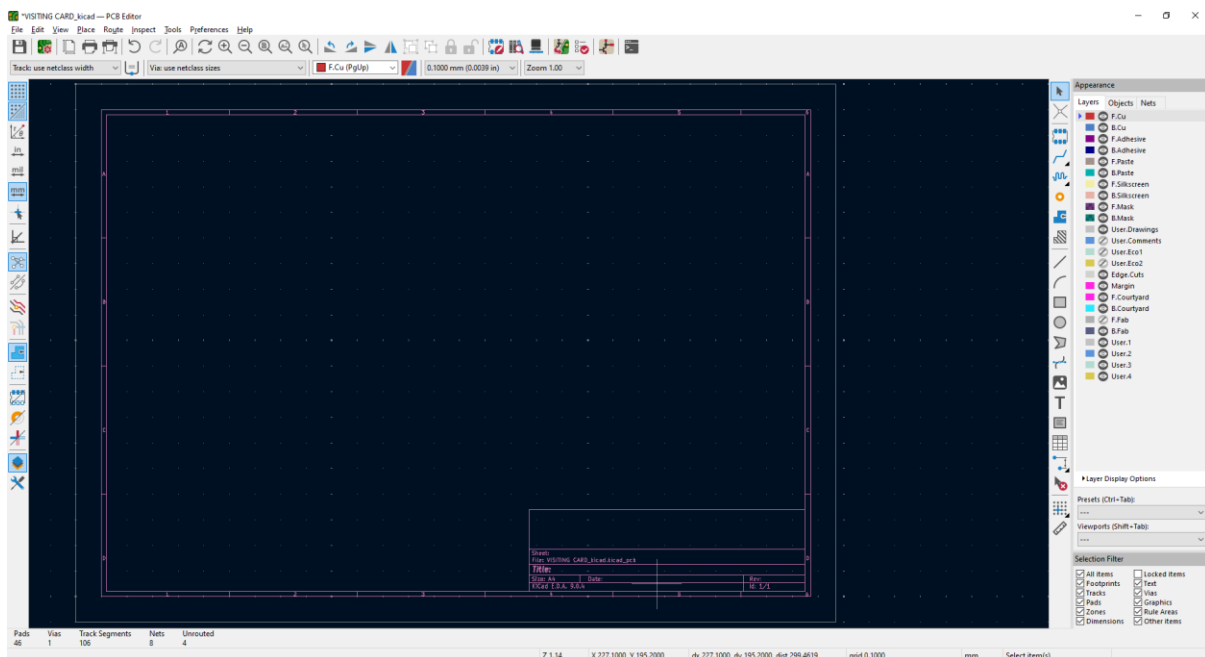- Press W key word

- Save the Schematic file.

Step 7:

- Go to 'Tool' → 'Update PCB from schematic' / F8
- Check each schematic symbol must have a matching PCB footprint
- If any footprint has missing it can't update PCB, it's shown error warning

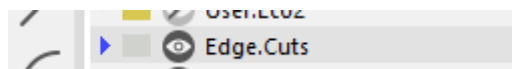- Go to clear the error (like a No unconnected, annotate symbols, Fix wiring errors, missing power flags)

Step 8:

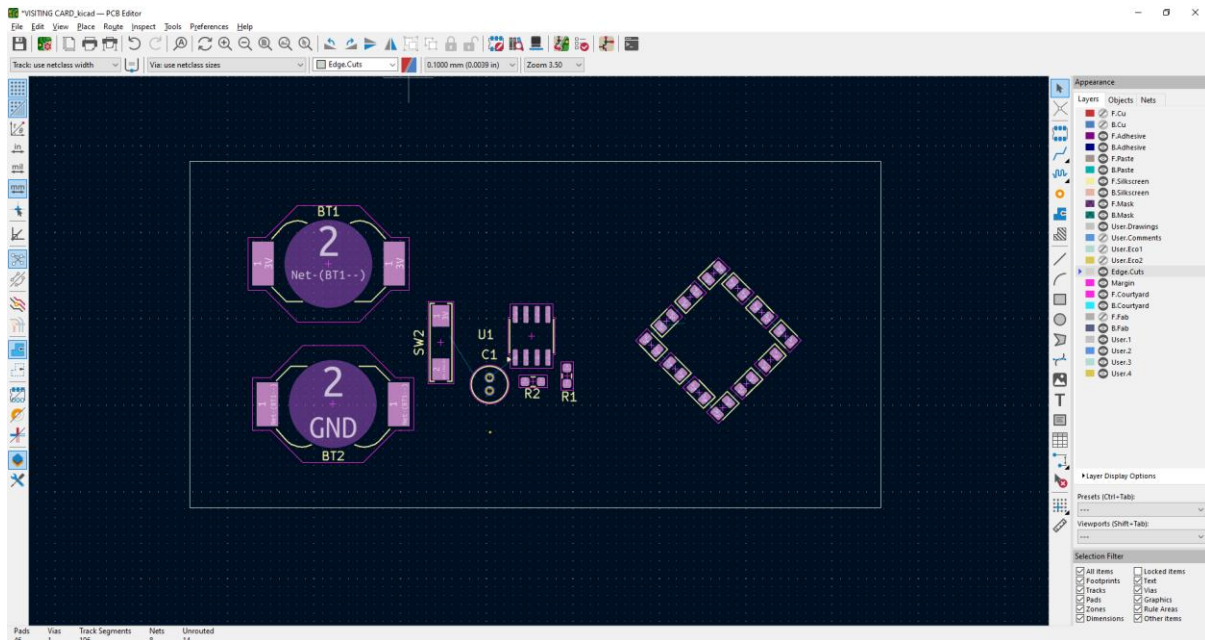- Set the board outline (Edge Cuts layer)

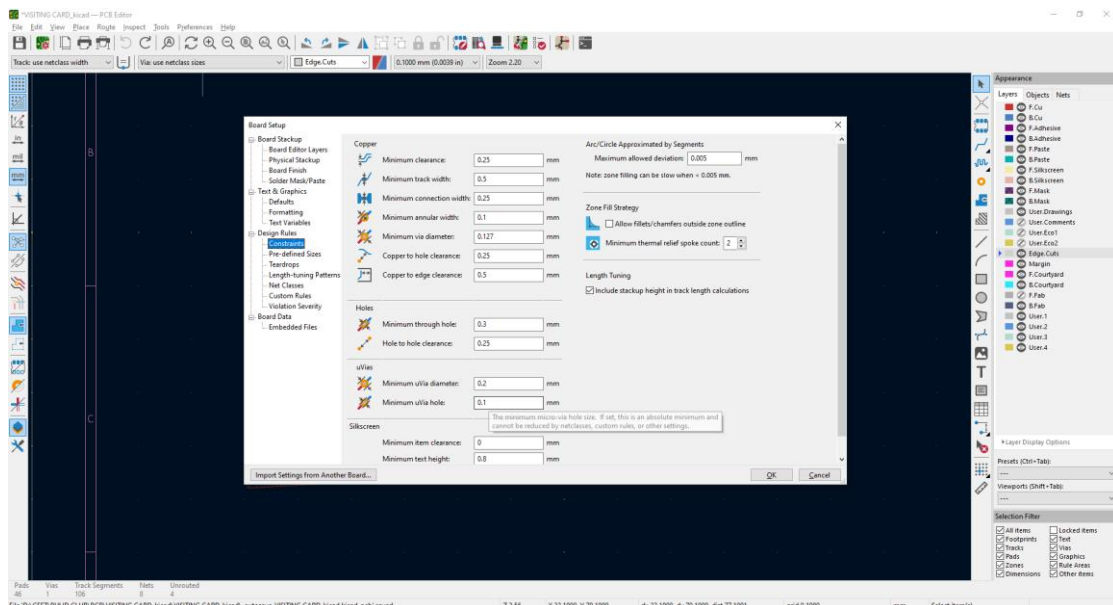- Click the Place Symbols ion, On your right-side corner

- Select the rectangle tool using to set the board outline
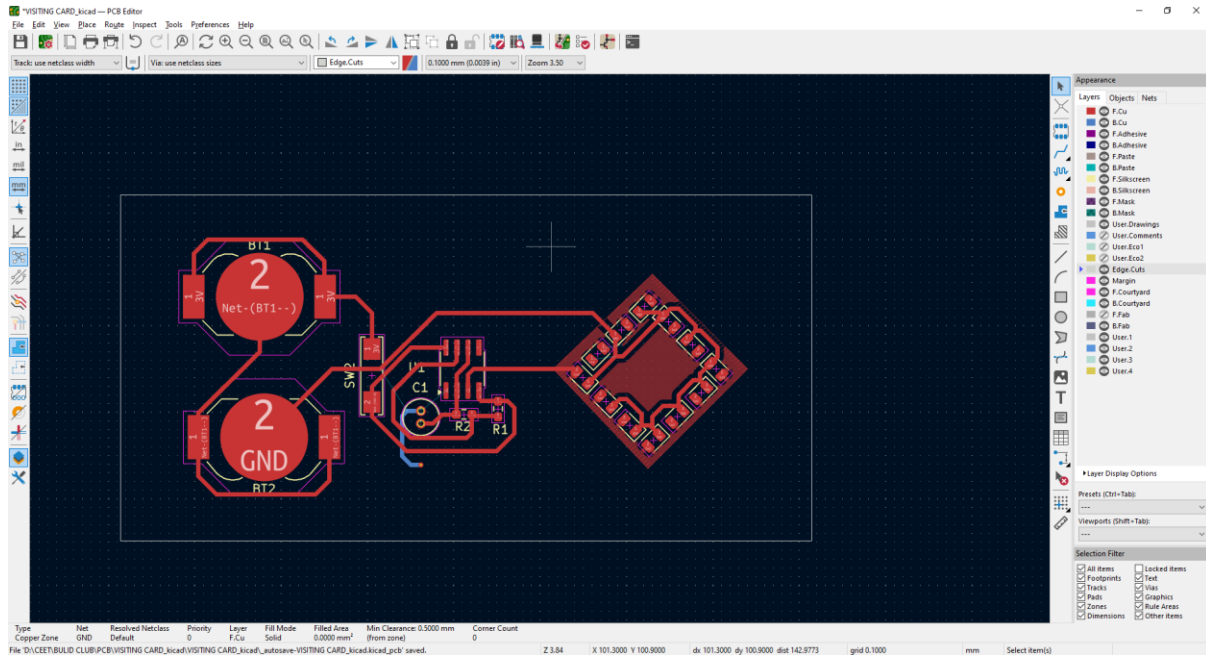
- To place all components on the sheet

Step 9:

- Next go to 'Board Setup'→ 'Constraints'
- Set the details of all option
- And refer the link (**Disclaimer** – 'Constraints' depends on PCB manufacturer)

Step 10:

- Go to select the layer after routing in the traces
- Traces thicker increase or decrease width via net class selection based

Step 11:

- Go to 'Inspect' → 'Design Rules checker'
- Check each Constraints value must have a matching PCB design
- If any Constraints value has missing it can't run DRC, it's shown error warning Go to clear the error (clearances, unconnected pins, minimum annular rings)
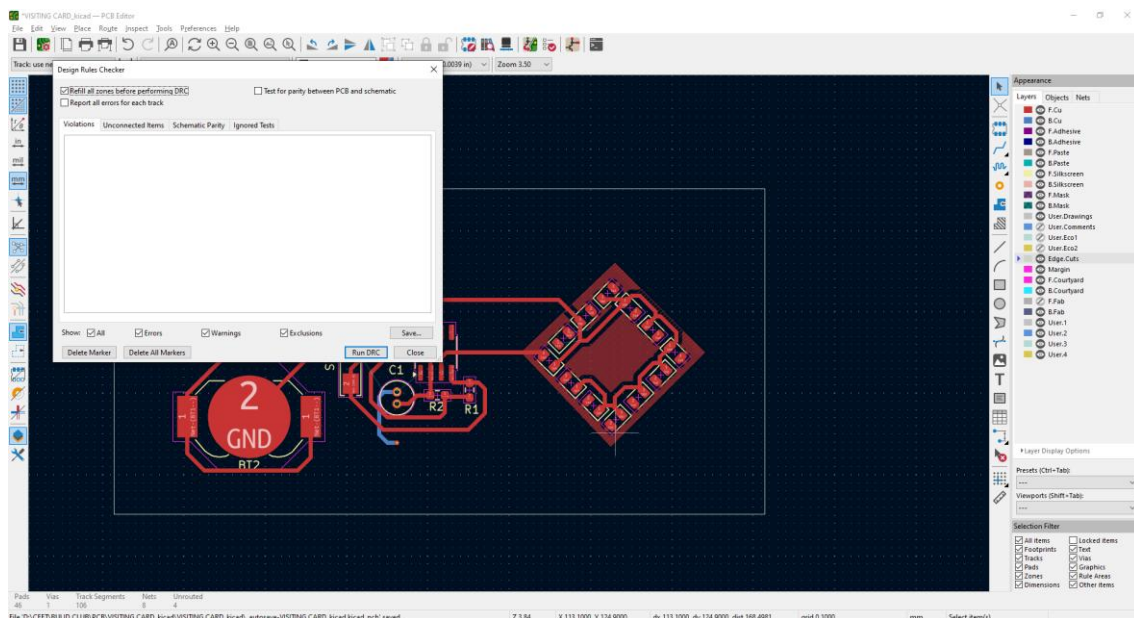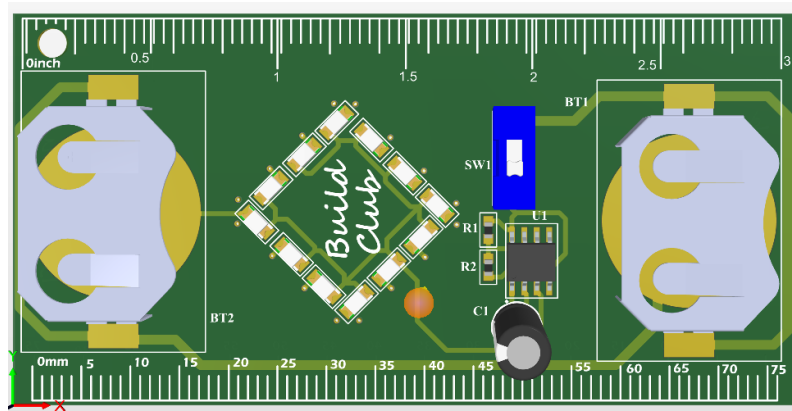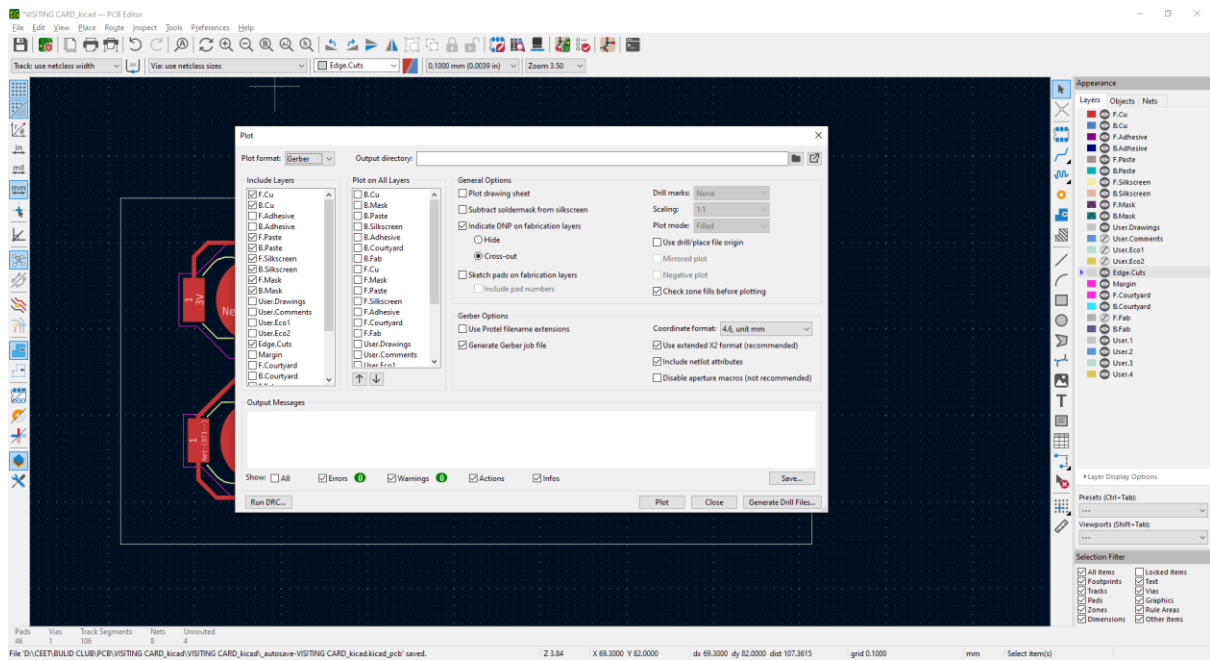

Step 12:

- If clear all error go to Fabrication outputs → Gerber, click it
- Select all layers

— F. Cu (Top copper)

— B. Cu (Bottom copper)

— F. Silk (Top silkscreen)

— B. Silk (Bottom silkscreen)

— F. Mask (Top solder mask)

— B. Mask (Bottom mask)

— Edge. Cuts (Board outline)

— Drill file (Excellon)

- Click the Generate Drill Files option

Reference [link](#)