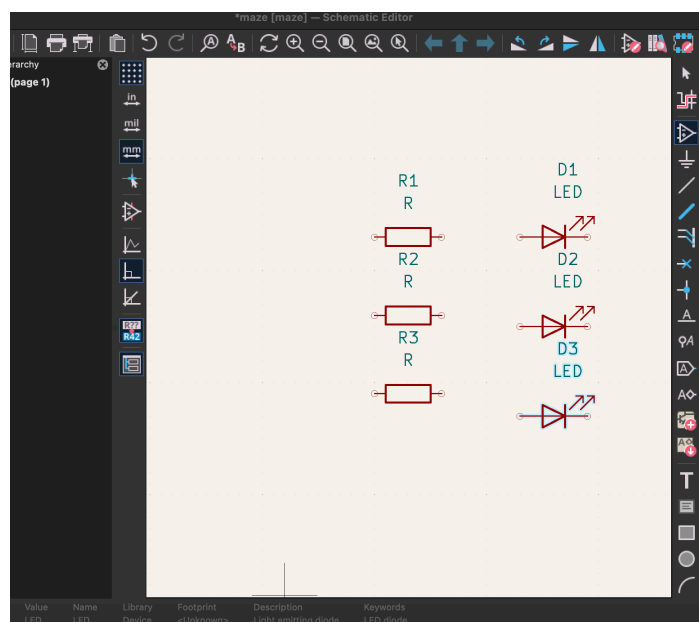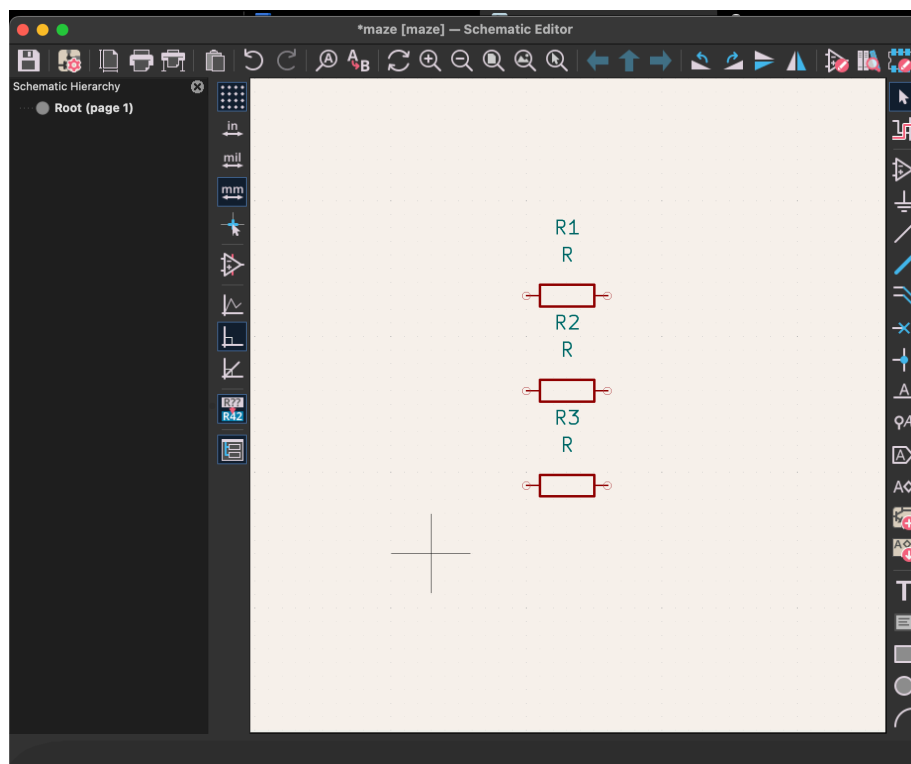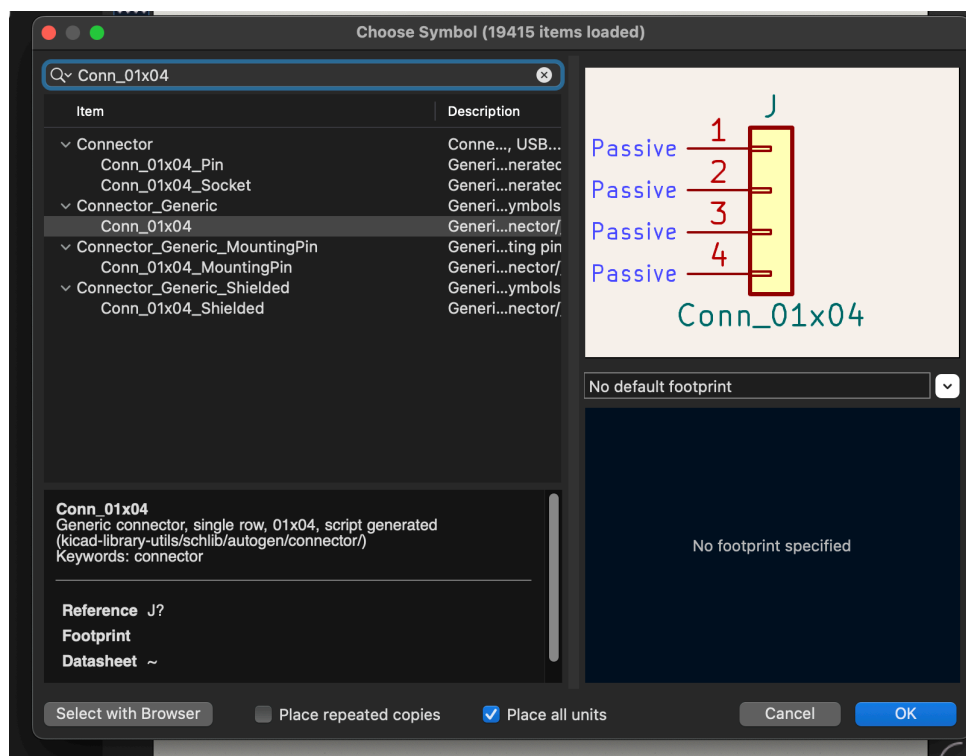# LED PCB using KiCAD

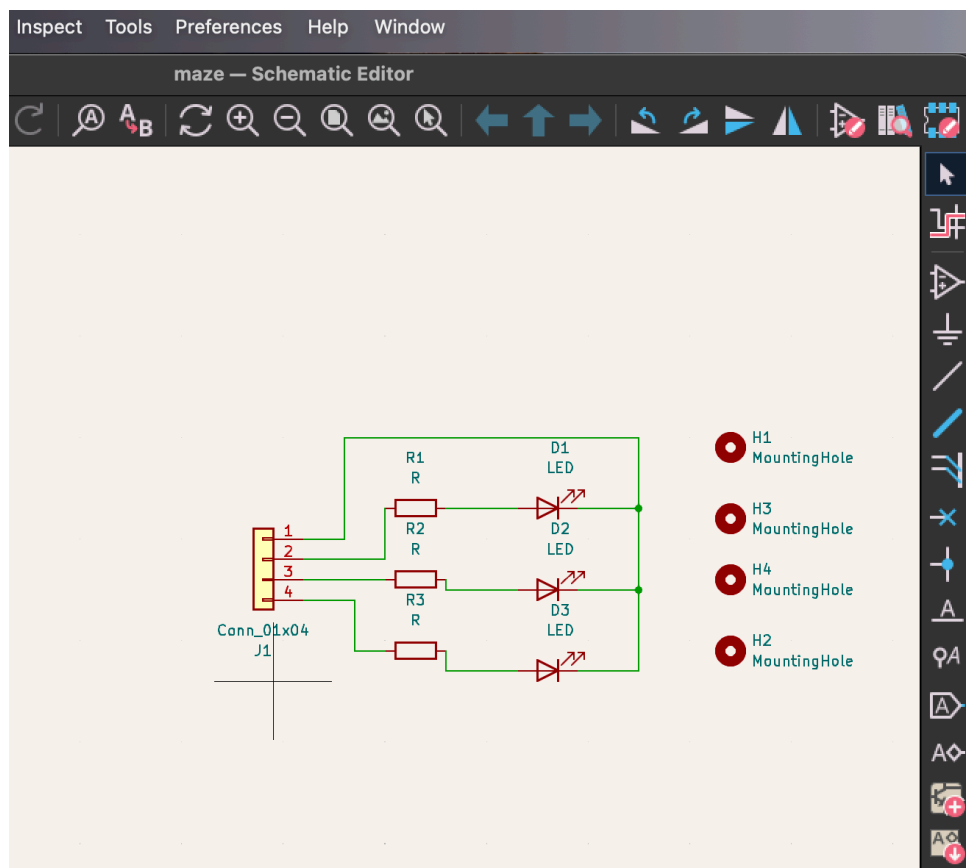**Adding symbols/Components:**

Led-LED
Resistor-R
Connector -Conn_01x04
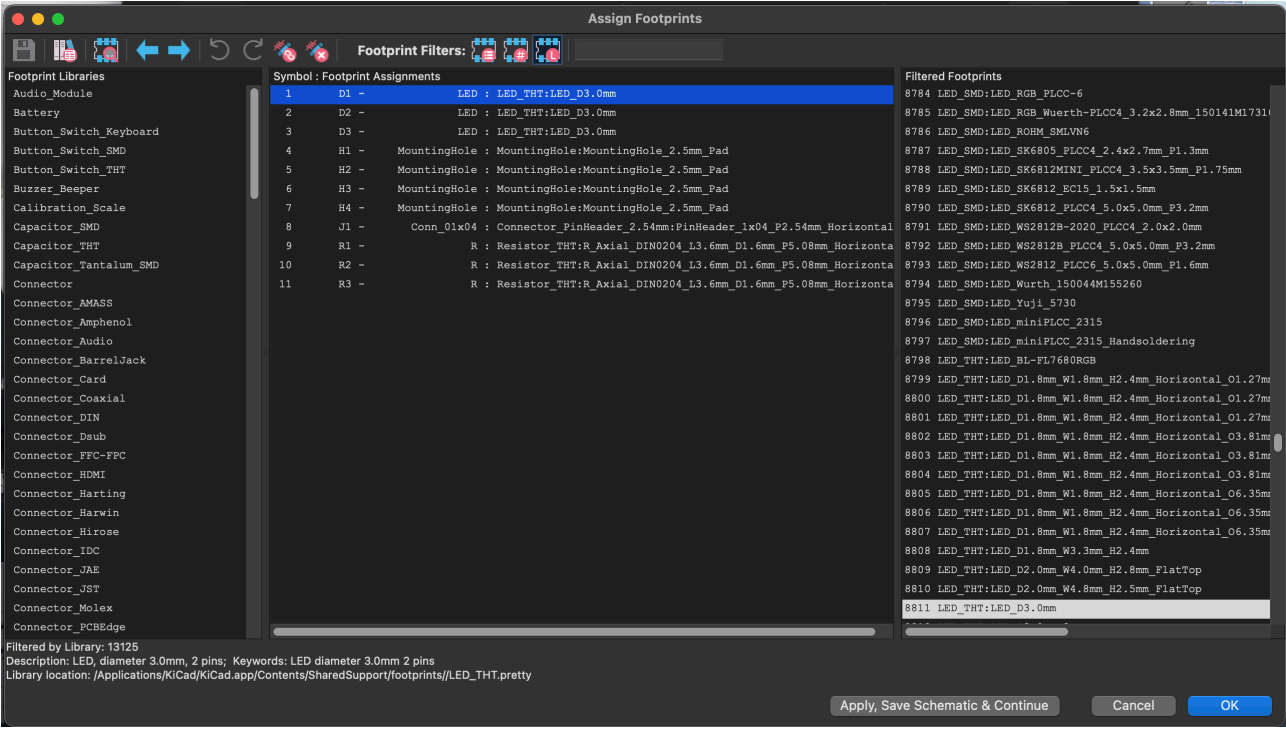Mounting holes- MountingHole

Connect all the components using the wires-

Adding footprints to the components:

Tools—>Assign Footprints or
Select the component—>Properties—>Footprints

3D view/ PCB view:

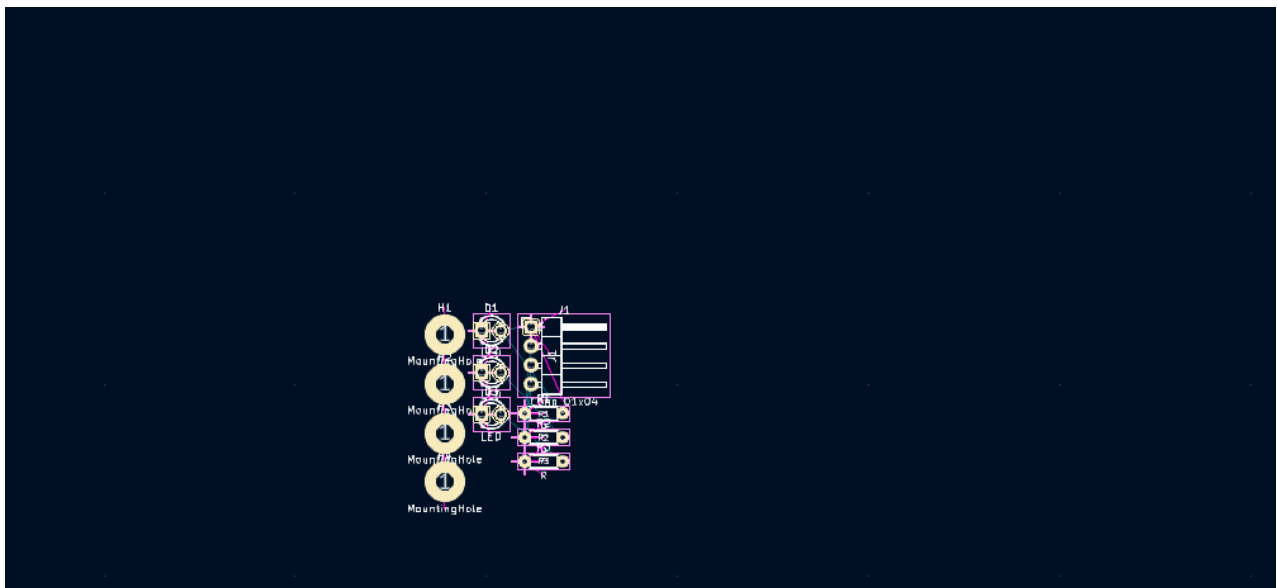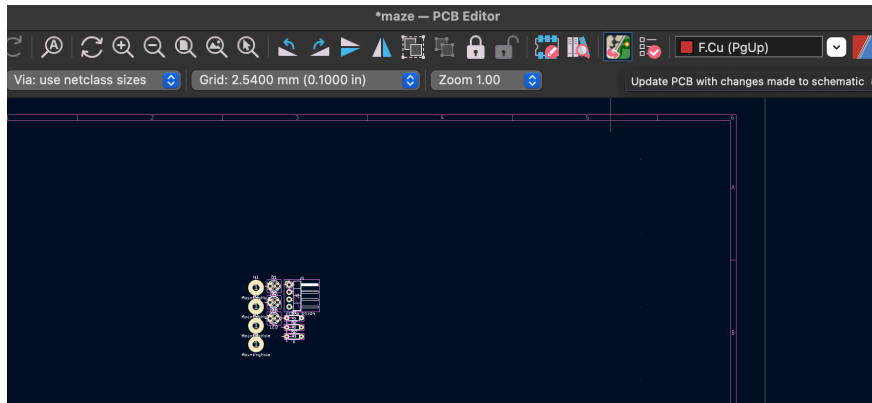| Resistors	Resistor_THT:R_Axial_DIN0204_L3.6mm_D1.6mm_P5.08mm_Horizontal
Led	LED_THT:LED_D3.0mm
Connector	Connector_PinHeader_2.54mm:PinHeader_1x04_P2.54mm_Horizontal
4 Mounting holes	MountingHole:MountingHole_2.5mm_Pad

Update PCB from schematics:

Arrange the components in according to the circuit by flipping & rotating the components. (Click to select)
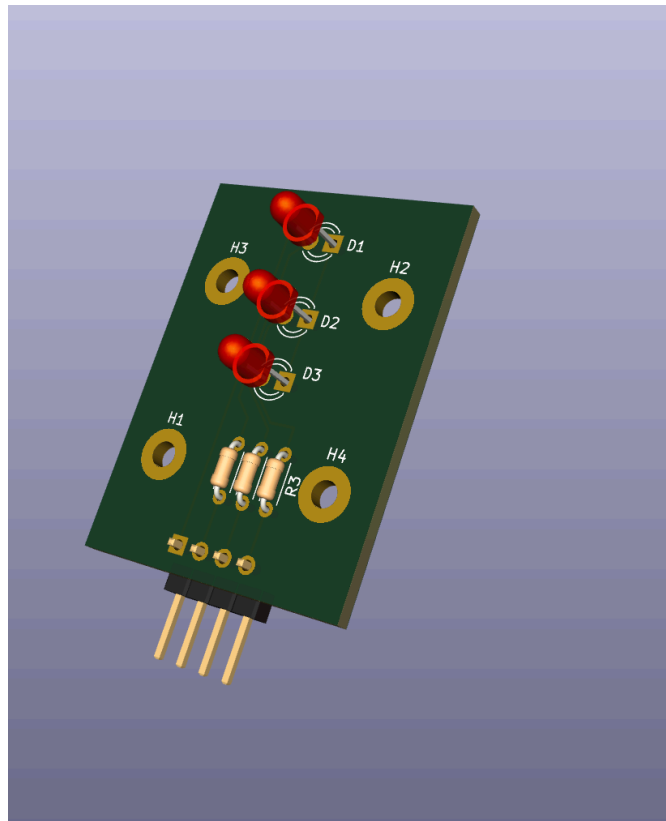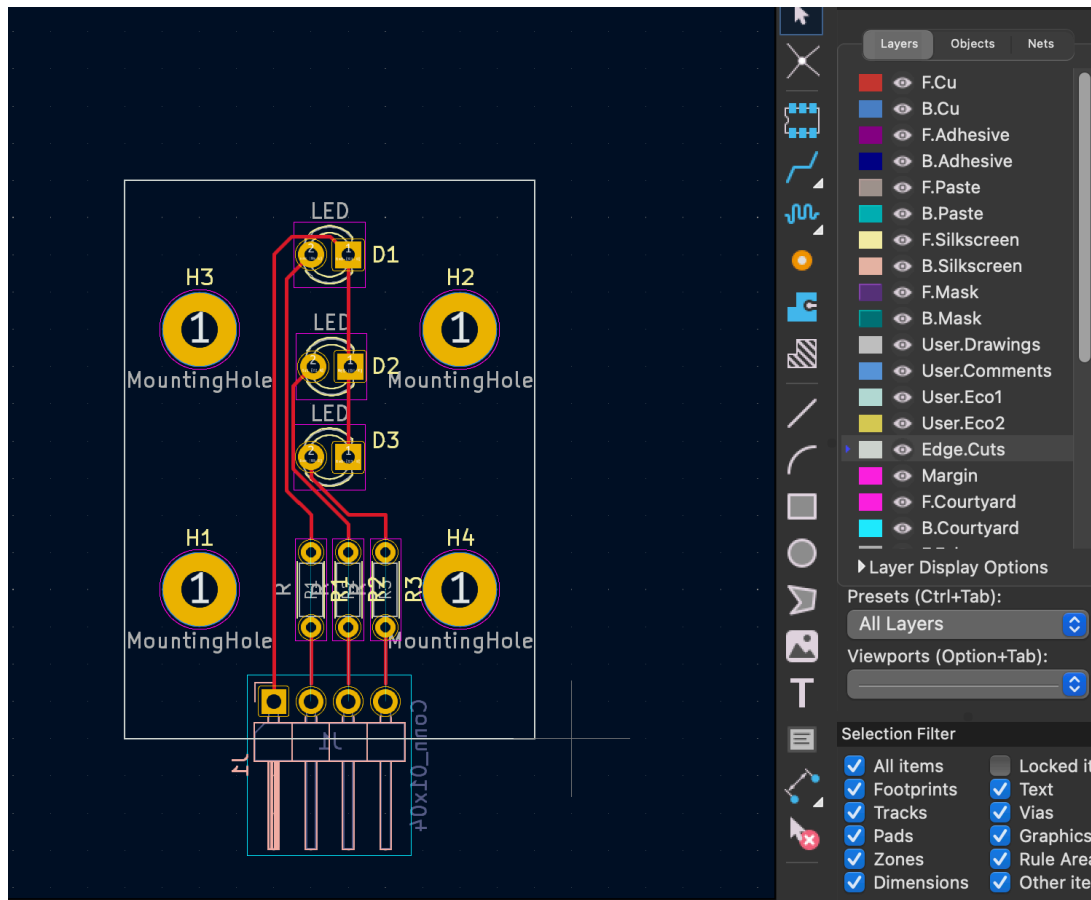
Select "edge cuts" in the layers section & draw a rectangle around the components.

For a 3D view: Go to View—> 3D viewer

Select 'Edge.Cuts' in 'Layers' and draw a rectangle using line ![line icon] around the components (acts as a support to the PCB components.)

Add Wires using the symbol: ![wire icon]

Final PCB view which can be viewed using the 3D viewer

Converting to a Gerber file:

File—>Fabrication outputs—>create a Gerber file—>generate Drill files—> Close

Select all the drill & Gerber files to make a zip file to share

References-

1. Guide on installing KiCAD and understanding it tools can be found here:
   https://docs.kicad.org/7.0/en/getting_started_in_kicad/getting_started_in_kicad.html

2. LED PCB:
   https://www.youtube.com/watch?v=EPH23zhPg50