








# Steps for Designing PCB Using KiCAD





## Schematic Editor

- ☐ 1. Create a  new blank project
- ☐ 2. Open the “Schematic Editor” from the right panel
- ☐ 3. Edit the page settings in the bottom
  - Title: LED Chaser Disc
  - Issue Date: 2024-03-29
  - Revision: 1
  - Company: Purdue University Northwest
  - Comment1: PNW IEEE
  - Comment2: Your Name

[Tip:: Check the box “Export to other sheets” to apply the same information for the PCB sheet]

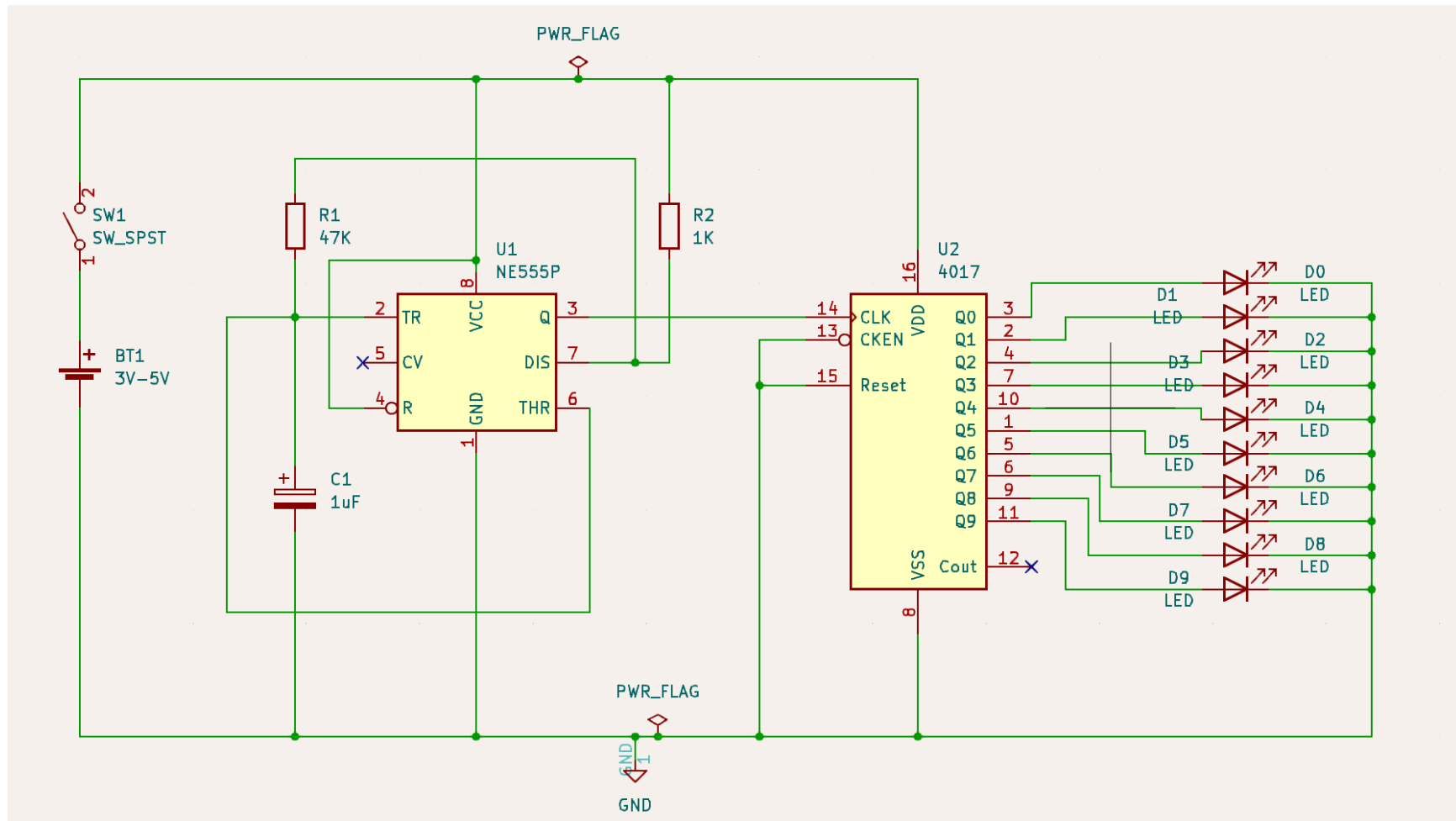
- ☐ 4. Place the IC 4017, and NE555P from the symbol library on the sheet.
- ☐ 5. Place “Battery\_Cell”, “SW\_SPST”, two resistors “R”, one polarized capacitor “C\_polarized”, a ground “GND”, 10 LED ‘s on the sheet. [Tip: Use the schematic provided at the end for reference on where to place the component]
- ☐ 6. Select the  wire from the right toolbar (or hit “w” on your keyboard) and make the connections as per the schematic diagram.
- ☐ 7. Add a power flag (“PWR\_FLG”) from the  power symbol library located on the right toolbar (or hit “p” on keyboard to open the library) for the following wires:
  - Connect one PWR\_FLG to the positive end of the battery cell.
  - And, the second PWR\_FLG to the negative end of the battery cell.
- ☐ 8. Use the  “symbol reference designator” from the top toolbar to annotate all the components on the schematic.
- ☐ 9. Select the  bulk edit symbols icon to assign the footprints for all the components on the schematic.
- ☐ 10. Now select the  library icon from the footprint column on each row to assign its respective footprint. Once completed, hit **Apply, Save Schematic & Continue** to apply the changes and then select **OK**.
  - Check Appendix C to add a custom footprint for a missing component.
- ☐ 11. Run the  Design Rule Check and make sure you have no errors before you proceed.
- ☐ 12. Now go to “file” and export the netlist for KiCad. You shall be later importing this netlist file into PCB editor.

## PCB Editor


- ☐ 13. Go to File > Import > Netlist. In the field for “Netlist file”, provide the path to the netlist saved from step 12. Leave the default settings. Select **Load and Test Netlist** and then hit **Update PCB**.
  - ☐ 14. Place this Rat’s nest on the sheet temporarily as it will be organized and traced in the later steps.
  - ☐ 15. Select “Edge cuts” from the “Appearance > Layers” section on the right toolbar and make sure the arrow is pointing to “Edge cuts”.
  - ☐ 16. Now, draw a approximate circle by selecting the  icon on the right toolbar. Open the circle properties by double clicking on the center point and change the following values.
    - X: 110 mm
    - Y: 100 mm
    - Radius: 42 mm
  - ☐ 17. Drag the components onto the circle, use APPENDIX D as a reference to arrange these components.
- Tip:
- Place the IC’s (Integrated circuits) first. Then place components with a large area (ex: battery holder) followed by resistors, capacitors, and other components.
  - Make sure the components are closer to each other and there is enough gap for routing a trace.
- ☐ 18. Use the “Freerouting” auto-router software to trace the routes of the PCB. (See APPENDIX E on how to download and use “Freerouting” tool).
    1. “File > Export > Specctra DSN” to export the specctra session which will be used by the Freerouting tool.
    2. Once all the components are routed using the software, import it from “File > Import > Specctra Session” and proceed with the following steps.
  - ☐ 19. Now create a ground zone using  “Filled zone” on the right toolbar around the outside circle. Select “F.Cu” from the Appearance, start creating the filled zone around the circle, and choose GND when the “Copper zone properties” window pops up. Repeat the same steps and create a filled zone for “B.Cu”
  - ☐ 20. You can view your PCB in 3D by going to “View > 3D viewer”
  - ☐ 21. Go to “File > Fabrication Outputs > Gerbers”
    1. Enter the path for “output directory” (Ex: fab)
    2. Hit **Generate Drill Files...** to generate gerber’s for drill files
    3. Then select **Plot** to generate the gerber files for selected layers. Then close the window.

Hurray! You have successfully finished designing a printed circuit board.

## Appendix A

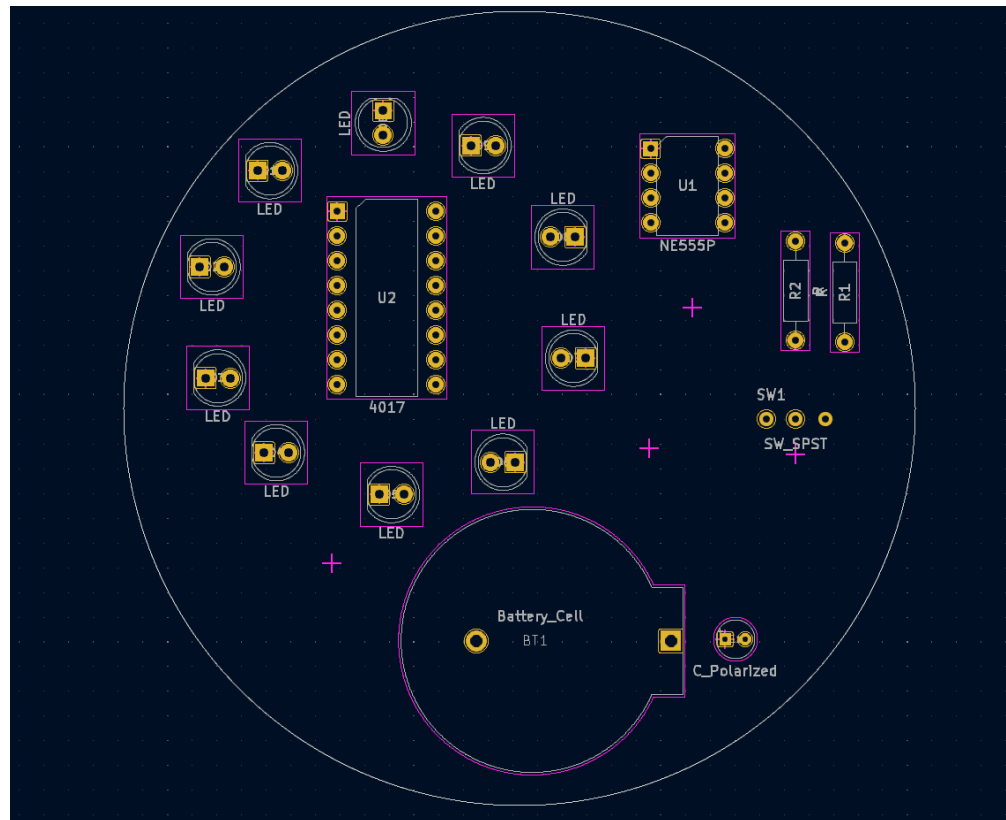


## Appendix B

Symbol Fields Table					
<input type="text" value="Filter"/>			<input checked="" type="checkbox"/> Group symbols		
Reference	Value	Footprint		Datasheet	Q...
BT1	3V-5V	Battery:BatteryHolder_ComfortableElectronic_CH273-2450_1x2450	~		1
C1	1uF	Capacitor_THT:CP_Radial_D4.0mm_P2.00mm	~		1
> D0-D9	LED	LED_THT:LED_D5.0mm	~		10
R1	47K	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal	~		1
R2	1K	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal	~		1
SW1	SW_SPST	custom_library_ieee:spst_ieee	~		1
U1	NE555P	Package_DIP:DIP-8_W7.62mm		<a href="http://www.ti.com/lit/ds/symlink/ne555p.pdf">http://www.ti.com/lit/ds/symlink/ne555p.pdf</a>	1
U2	4017	Package_DIP:DIP-16_W10.16mm		<a href="http://www.intersil.com/content/dam/intersil/products/datasheets/4017.pdf">http://www.intersil.com/content/dam/intersil/products/datasheets/4017.pdf</a>	1

Export as CSV...
Apply, Save Schematic & Continue
Cancel
OK

## APPENDIX D



## APPENDIX E

### Freerouting Setup

1. Go to “Releases” page (<https://github.com/freerouting/freerouting/releases>), scroll down to “Assets”, then download the windows installer “freerouting-1.9.0-windows-x64.msi”.
2. Once downloaded, locate the installer and go over the installation process. Leave all the options default.
3. Locate the application “Freerouting” from “Start” and open it.
4. Select the design file produced from Step-18 and open it.
5. Hit “Start auto-router” to let the software perform automated routing. Once finished select “File > Export Specctra Session File”. This will save the current configuration as a specctra session file.
6. Go back to Step-18, item 2 and follow the remaining steps.

Note: the input to this application ends with “.dsn” corresponding to “Specctra DSN” and the output from this is “Specctra session” ending with “.ses”