File → New Project
New Project
New Schematic/Board

Browse to your desired location, and give your project a name.

KiCad 6.0 — getting-started-in-kicad-6.0

File View Tools Preferences Help

Project Files

→ Schematic file ←
→ Board file

Schematic Editor ✓
Edit the project schematic

Symbol Editor
Edit global and/or project schematic symbol libraries

PCB Editor ✓
Edit the project PCB design

Footprint Editor
Edit global and/or project PCB footprint libraries

Gerber Viewer
Preview Gerber files

Image Converter
Convert bitmap images to schematic symbols or PCB footprints

Calculator Tools
Show tools for calculating resistance, current capacity, etc.

Drawing Sheet Editor
Edit drawing sheet borders and title blocks for use in schematics and PCB designs

Plugin and Content Manager
Manage downloadable packages from KiCad and 3rd party repositories

Symbol Library Table Setup :

The first time the schematic editor is opened, a dialog box will appear asking how to configure the global symbol library table.

Configure Global Symbol Library Table

KiCad has been run for the first time using the new symbol library table for accessing libraries. In order for KiCad to access symbol libraries, you must configure your global symbol library table. Please select from one of the options below. If you are not sure which option to select, please use the default selection.

- ● Copy default global symbol library table (recommended)
- ○ Copy custom global symbol library table
- ○ Create an empty global symbol library table

Select global symbol library table file:

(None)

OK

These are standard library path to particular operating system.

- ✓ Windows: C:\Program Files\KiCad\6.0\share\kicad\template\
- ✓ Linux: /usr/share/kicad/template/
- ✓ macOS: /Applications/KiCad/KiCad.app/Contents/SharedSupport/template/

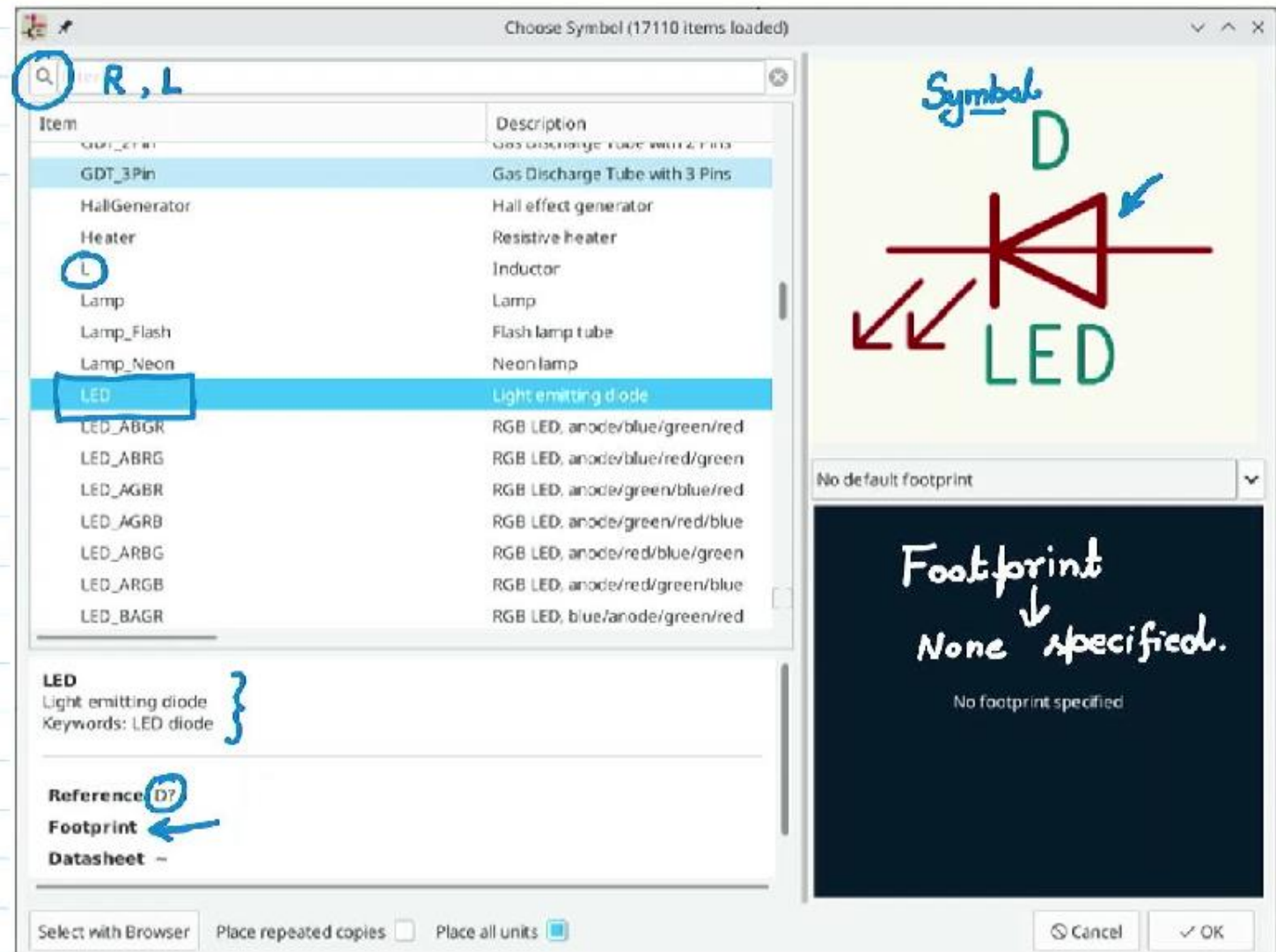NPTEL

Schematic Sheet setup:

Goto File → Page settings.

Title block is for the information purpose.

Page Settings dialog:

Paper

Size:
A 8.5x11in

Orientation:
Landscape

Custom paper size:
Height: 279.4 mm
Width: 431.8 mm

☐ Export to other sheets

Preview

Drawing Sheet

File:

Title Block

Number of sheets: 1   Sheet number: 1

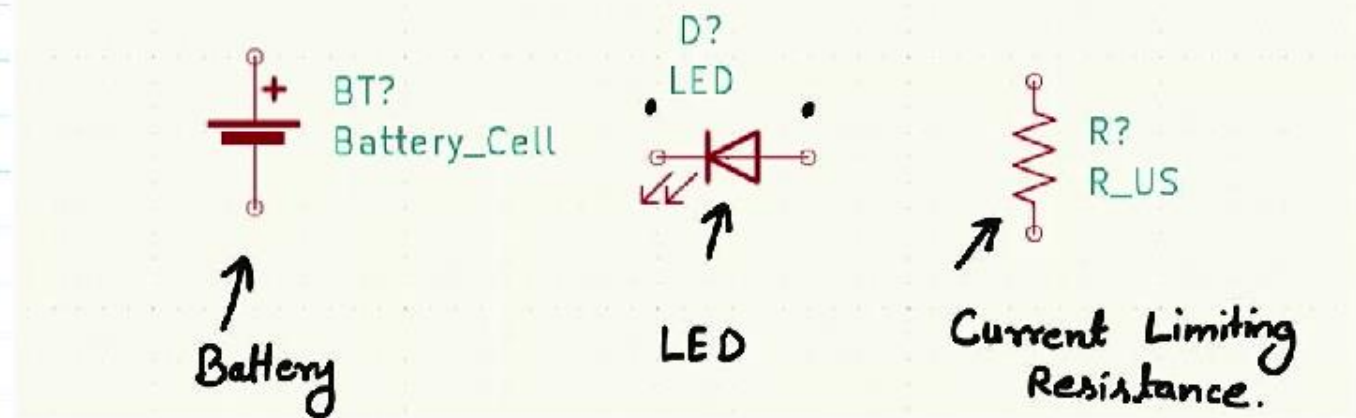|			
Issue Date:	2021-12-11	<<< 12/11/2021 ▼	☐ Export to other sheets
Revision:	0		☐ Export to other sheets
Title:	Getting Started in KiCad 6.0		☐ Export to other sheets
Company:			☐ Export to other sheets
Comment1:			☐ Export to other sheets
Comment2:			☐ Export to other sheets
Comment3:			☐ Export to other sheets
Comment4:			☐ Export to other sheets
Comment5:			☐ Export to other sheets
Comment6:			☐ Export to other sheets
Comment7:			☐ Export to other sheets
Comment8:			☐ Export to other sheets
Comment9:			☐ Export to other sheets

Information

⊘ Cancel   ✓ OK

NPTEL

Adding the symbols to the schematic:

We start making the circuit by adding desired symbols to the schematic.

Click on "Add a symbol" button or pressing "A".

Choose Symbol (17110 items loaded)

Search: R , L

Item	Description
GDT_2Pin	Gas Discharge Tube with 2 Pins
GDT_3Pin	Gas Discharge Tube with 3 Pins
HallGenerator	Hall effect generator
Heater	Resistive heater
L	Inductor
Lamp	Lamp
Lamp_Flash	Flash lamp tube
Lamp_Neon	Neon lamp
LED	Light emitting diode
LED_ABGR	RGB LED, anode/blue/green/red
LED_ABRG	RGB LED, anode/blue/red/green
LED_AGBR	RGB LED, anode/green/blue/red
LED_AGRB	RGB LED, anode/green/red/blue
LED_ARBG	RGB LED, anode/red/blue/green
LED_ARGB	RGB LED, anode/red/green/blue
LED_BAGR	RGB LED, blue/anode/green/red

LED
Light emitting diode
Keywords: LED diode

Reference D?
Footprint
Datasheet –

No default footprint

No footprint specified

Select with Browser   Place repeated copies ☐   Place all units ☑

⊘ Cancel   ✓ OK

Symbol
D
LED

Footprint
↓
None specified.

NPTEL

Selecting and Moving objects:

Schematic Window:

Battery — BT? Battery_Cell
LED — D? LED
Current Limiting Resistance — R? R_US

Selected objects are moved
by pressing "M".
and rotated by pressing "R".

BT? Battery_Cell
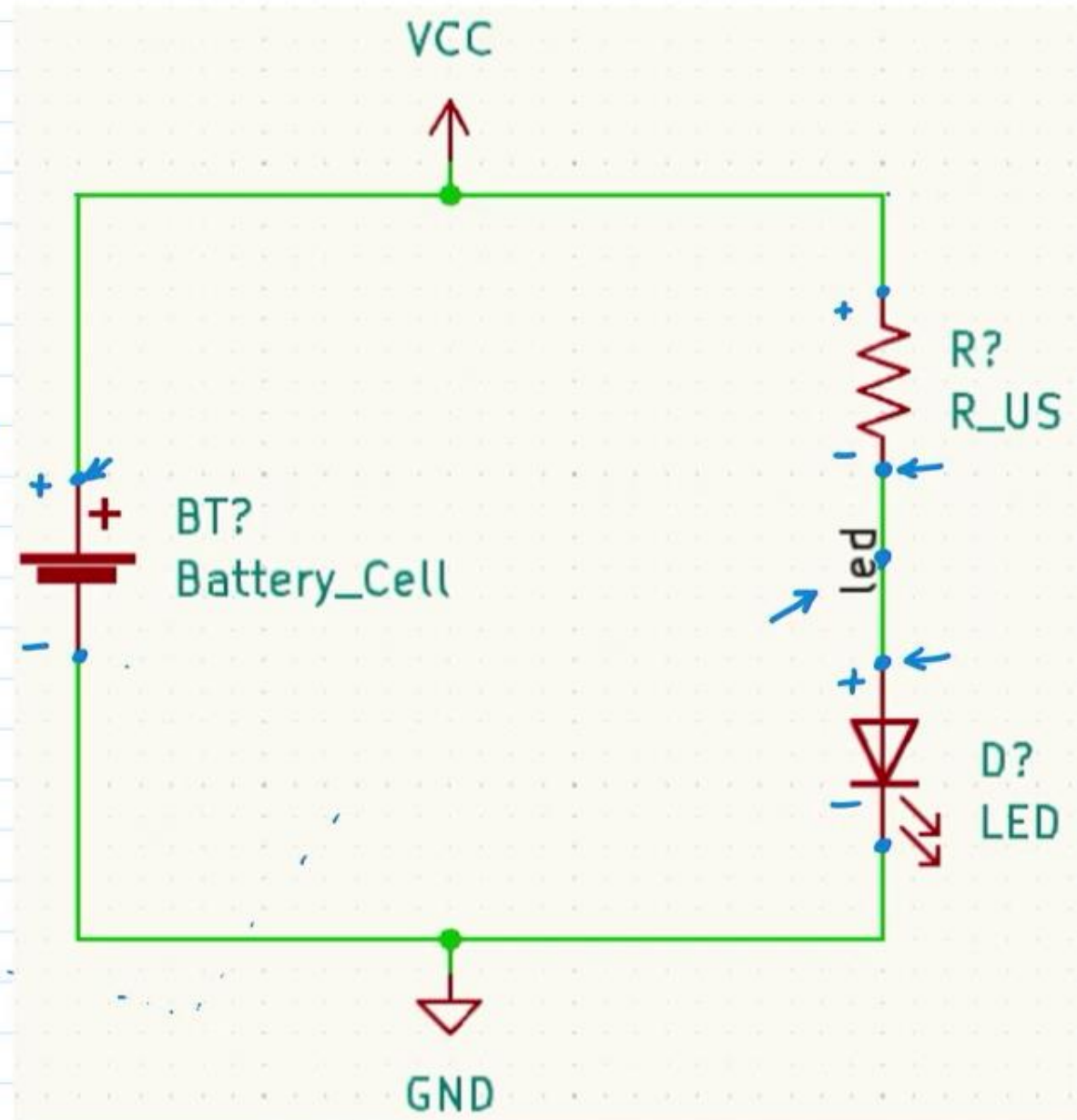R? R_US
D? LED

NPTEL

Wiring the Schematic :

→ click on "Add a wire" button on the right hand toolbar.

→ Add a VCC symbol and GND symbol. and connect them to circuit with wires

→ Click on "NET Label" button and name "led" to the net.

→ Labels with same name are connected together.

VCC

BT?
Battery_Cell

R?
R_US

led

D?
LED

GND

Annotation :

each symbol needs a unique reference.

Click "Annotate" and choose default setting.

Symbol properties :

Fill in the values for each component.

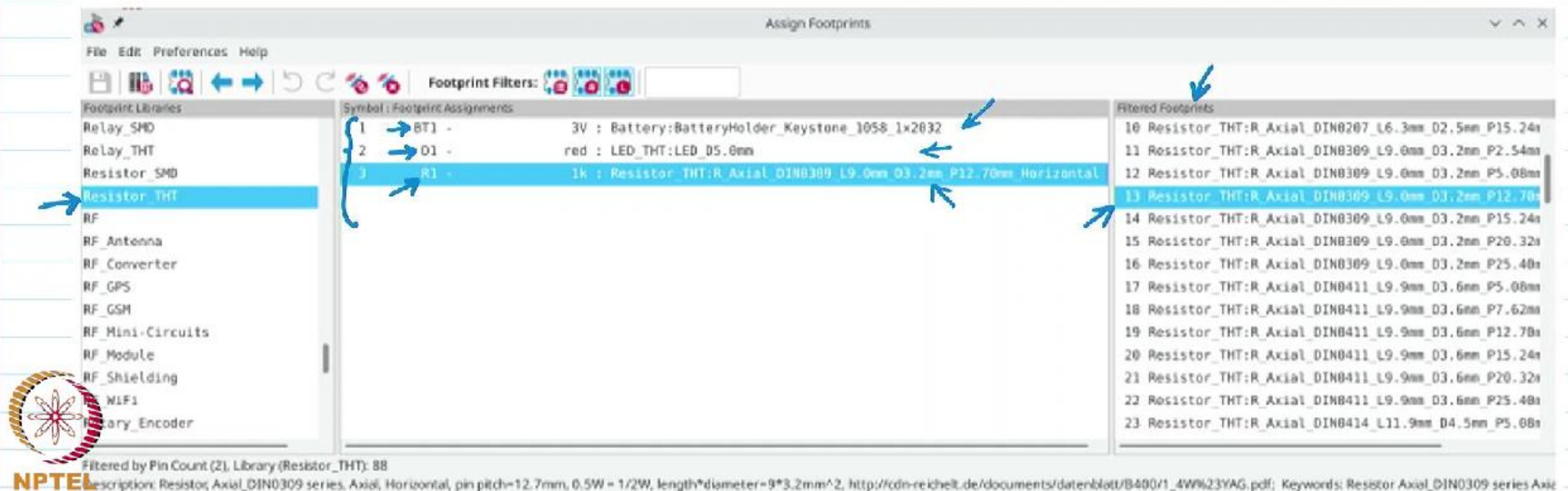Lithium coin cell battery $\longrightarrow$ 3 V

Resistance $\longrightarrow$ 1k Ω

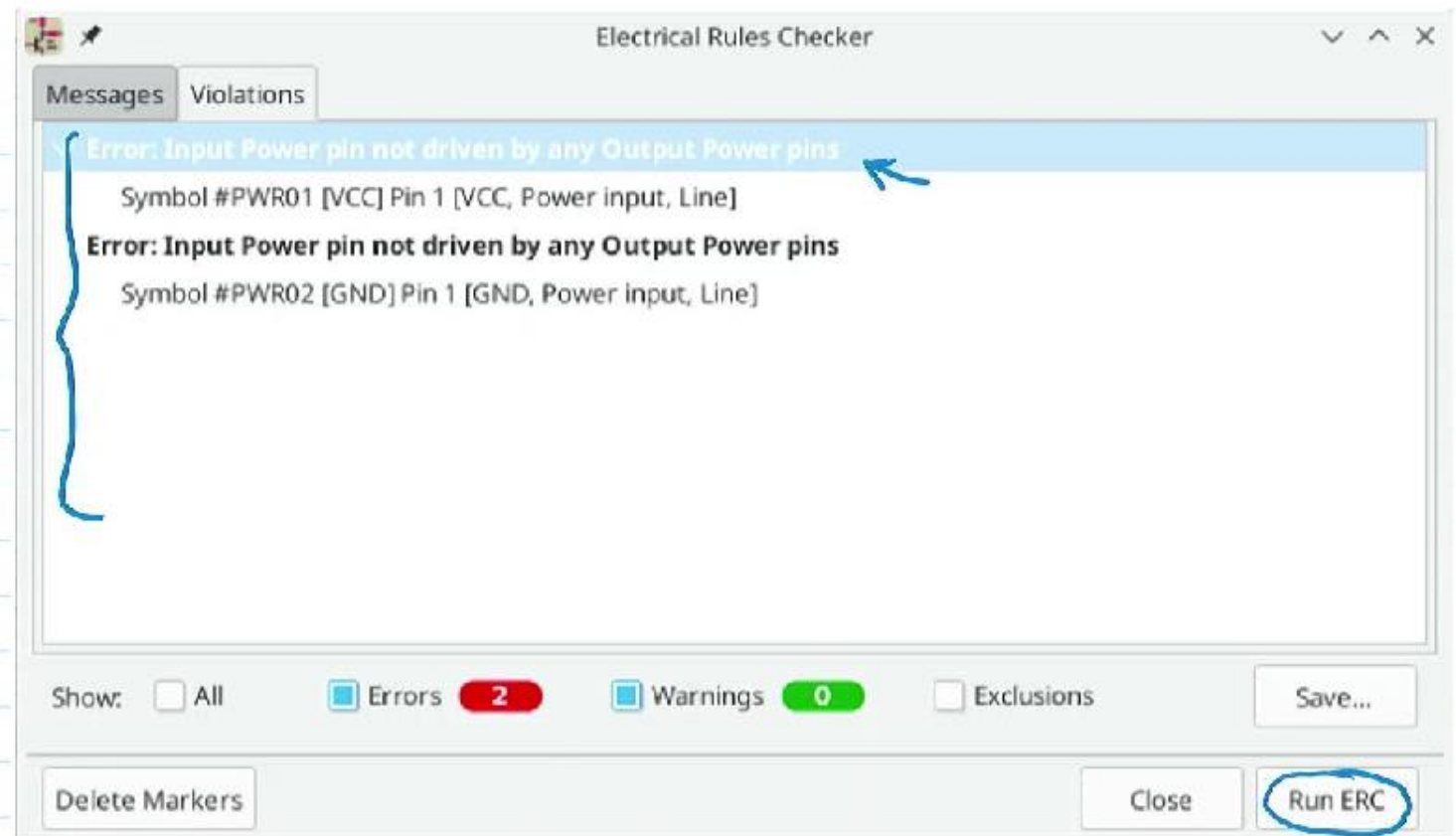**Foot print Assignment :** Many symbols are pre-assigned footprints.

All symbols need to be attached to particular fo footprints.

4r6
3r2

Electrical Rule Check : (ERC)

It is good to run ERC before starting Layout.

Electrical Rules Checker

Messages | **Violations**

Error: Input Power pin not driven by any Output Power pins
Symbol #PWR01 [VCC] Pin 1 [VCC, Power input, Line]
Error: Input Power pin not driven by any Output Power pins
Symbol #PWR02 [GND] Pin 1 [GND, Power input, Line]

Show: ☐ All   ☑ Errors **2**   ☑ Warnings **0**   ☐ Exclusions   Save...

Delete Markers   Close   **Run ERC**

NPTEL

Bill of Materials (BoM):

Tools → Generate BOM.

Bill of Material

BOM generator scripts:

bom_csv_grouped_by_value

bom_csv_grouped_by_value_with_fp

Generator nickname: bom_csv_grouped_by_value_with_fp

Output: CSV (comma-separated)
Grouped By: Value, Footprint
Sorted By: Ref
Fields: Ref, Qnty, Value, Cmp name, Footprint, Description, Vendor

Command line:
python "pathToFile/bom_csv_grouped_by_value_with_fp.py" "%I" "%O.csv"

Command line running the generator:

python "/usr/share/kicad-nightly/plugins/bom_csv_grouped_by_value_with_fp.py" "%I" "%O.csv"

Reset to Defaults   Help   Close   Generate