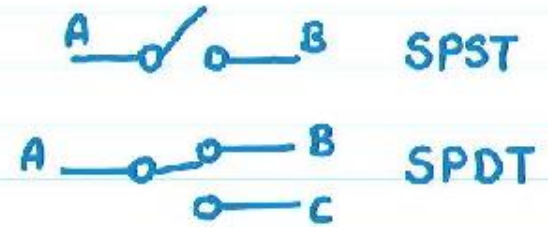


## Custom Symbols and Footprints :

The last circuit would be improved by adding a switch to turn-ON/turn-OFF the LED.

- ↳ Creating a new symbol
- ↳ Associated new Footprint.

We will design a SPST switch.



## Library and Library Tables :

Symbols and Footprints are organized into libraries.

KiCad keeps track of symbol and footprint libraries in Library Tables.

To access symbol library tables → Preferences → Manage symbol libraries.

To access footprint library tables → Preferences → Manage footprint libraries.

## Creating New Global or Project Libraries:

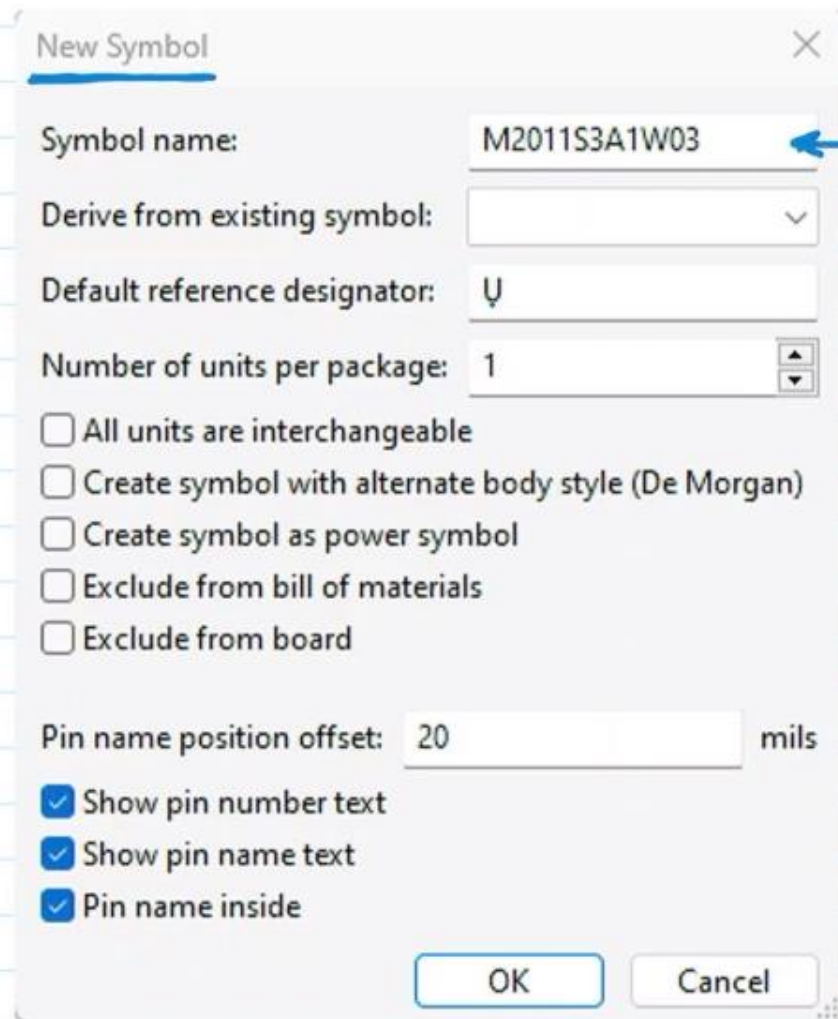
Click **File** → **New Library** and select **Project**

choose a name for the new library.

## Creating New Symbols:

**File** → **New Symbol**

Enter the part number in symbol name field.



New Symbol

Symbol name: M2011S3A1W03

Derive from existing symbol:

Default reference designator: U

Number of units per package: 1

☐ All units are interchangeable

☐ Create symbol with alternate body style (De Morgan)

☐ Create symbol as power symbol

☐ Exclude from bill of materials

☐ Exclude from board

Pin name position offset: 20 mils

☒ Show pin number text

☒ Show pin name text

☒ Pin name inside

OK Cancel

## Symbol Pins :

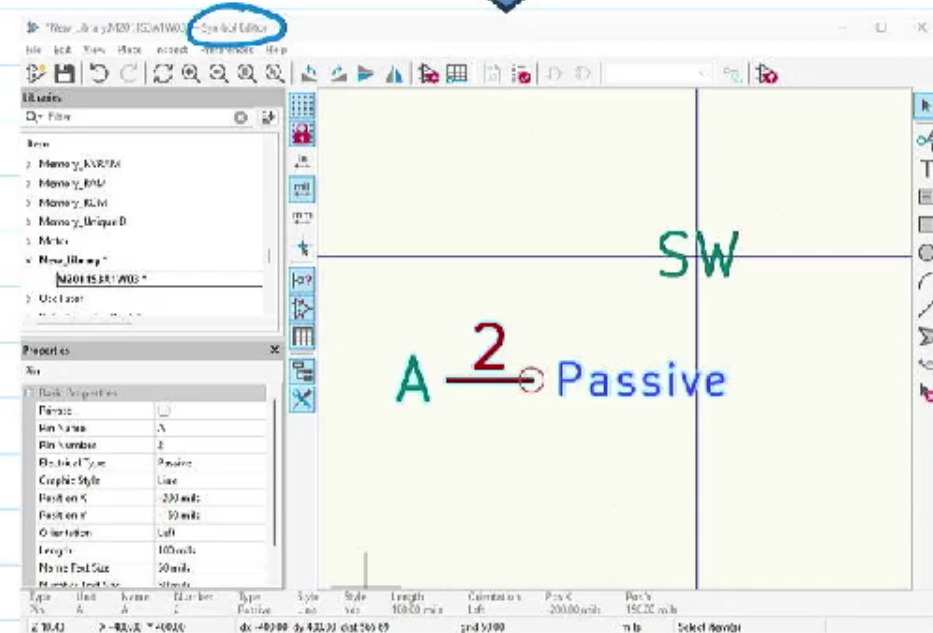
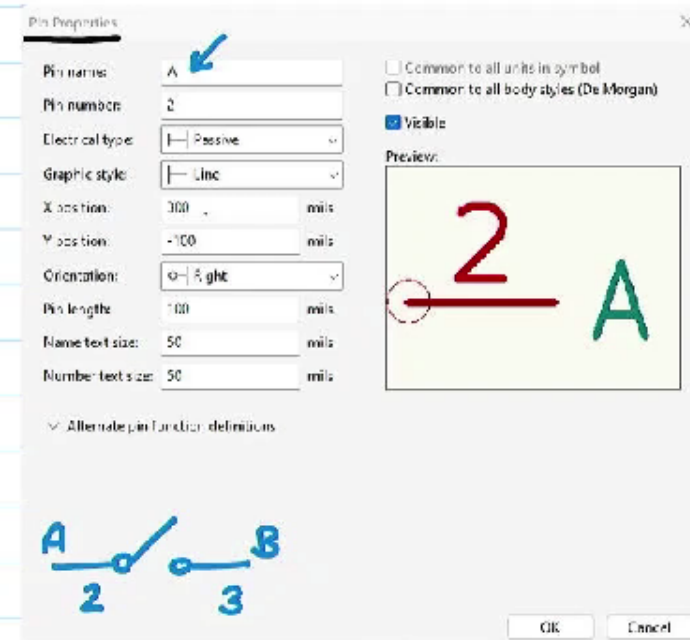
Start drawing the symbol by adding a pin.

Click the button "Add a pin".

pin name → A  
pin number → 2

Click OK.

- Rotate clockwise.



## Graphical Features :

\*New\_Library\M2011S3A1W03 - Symbol Editor

File Edit View Place Inspect Preferences Help

Libraries

Q Filter

Item

- > Memory\_NVRAM
- > Memory\_RAM
- > Memory\_ROM
- > Memory\_Unique D
- > Meter
- ✓ New\_Library \*
- M2011S3A1W03 \*
- > Oscillator

Properties

No objects selected

Name: M2011S3A1W03 Unit: A Body: Standard Type: Symbol Description: Keywords: Datasheet:

Z: 7.74 X: -350.00 Y: 500.00 dx: -350.00 dy: 500.00 dist: 610.33 grid: 50.00 units Select item(s)

SW

2

3

A

Passive

B



## Symbol properties :

File → Symbol properties.

add spst switch toggle to the keyword field to make it easier to find the symbol by searching.

Library Symbol Properties

General Footprint Filters

Fields

Name	Value	Show	Show Name	H Align	V Align	Italic	Bold
Reference	SW	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>
Value		<input checked="" type="checkbox"/>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>
Footprint		<input type="checkbox"/>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>
Datasheet		<input type="checkbox"/>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>
Description		<input type="checkbox"/>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>

+ ↑ ↓

Symbol name: M2011S3A1W03

Keywords: SPST Switch Toggle

Derive from symbol:

General

Number of units: 1

☒ All units are interchangeable

☐ Has alternate body style (De Morgan)

☐ Define as power symbol

Pin Text Options

☐ Show pin number

☒ Show pin name

☒ Place pin names inside

Position offset: 0 mils

Attributes

☐ Exclude from simulation

☐ Exclude from bill of materials

☐ Exclude from board

Edit Simulation Model... OK Cancel

# Creating New Footprint :

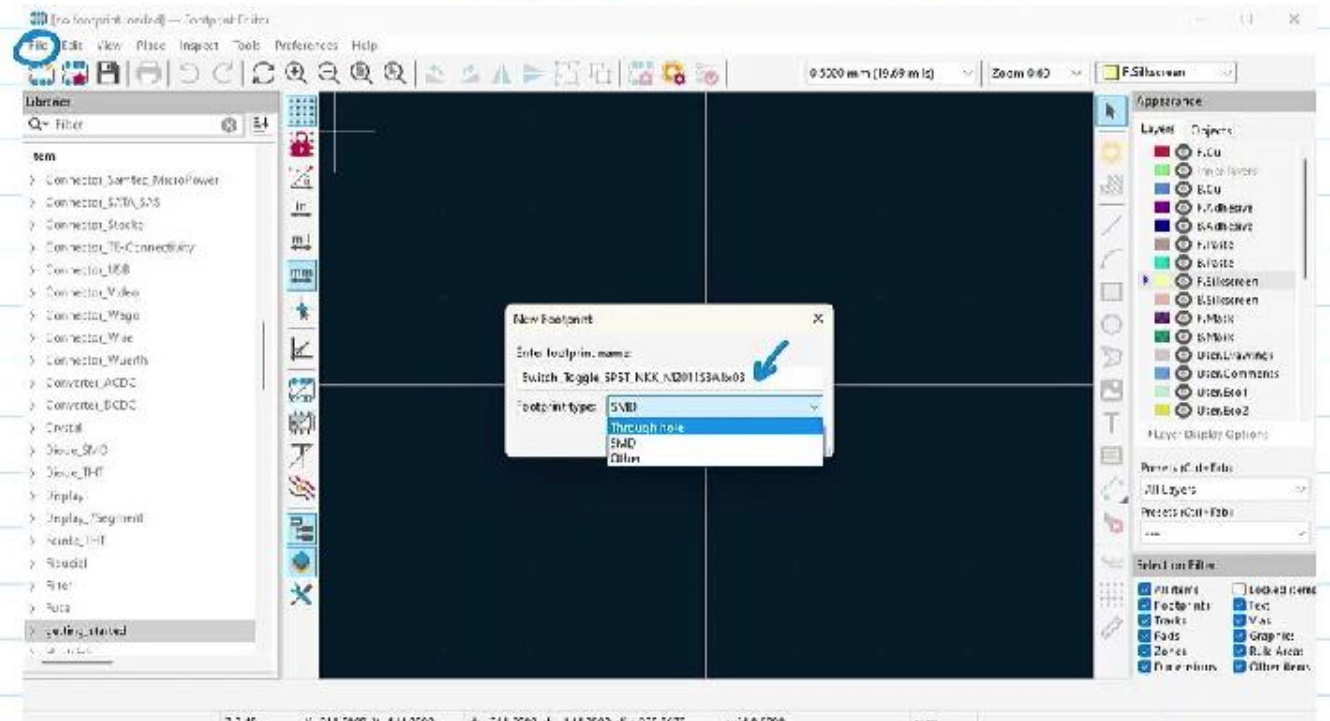
Open the Footprint editor and create a footprint library. **File → New Library.**

The new footprint library will be added to project library table.

**File → New Footprint.**

Select SMD or Through-Hole type.

Footprint editor window

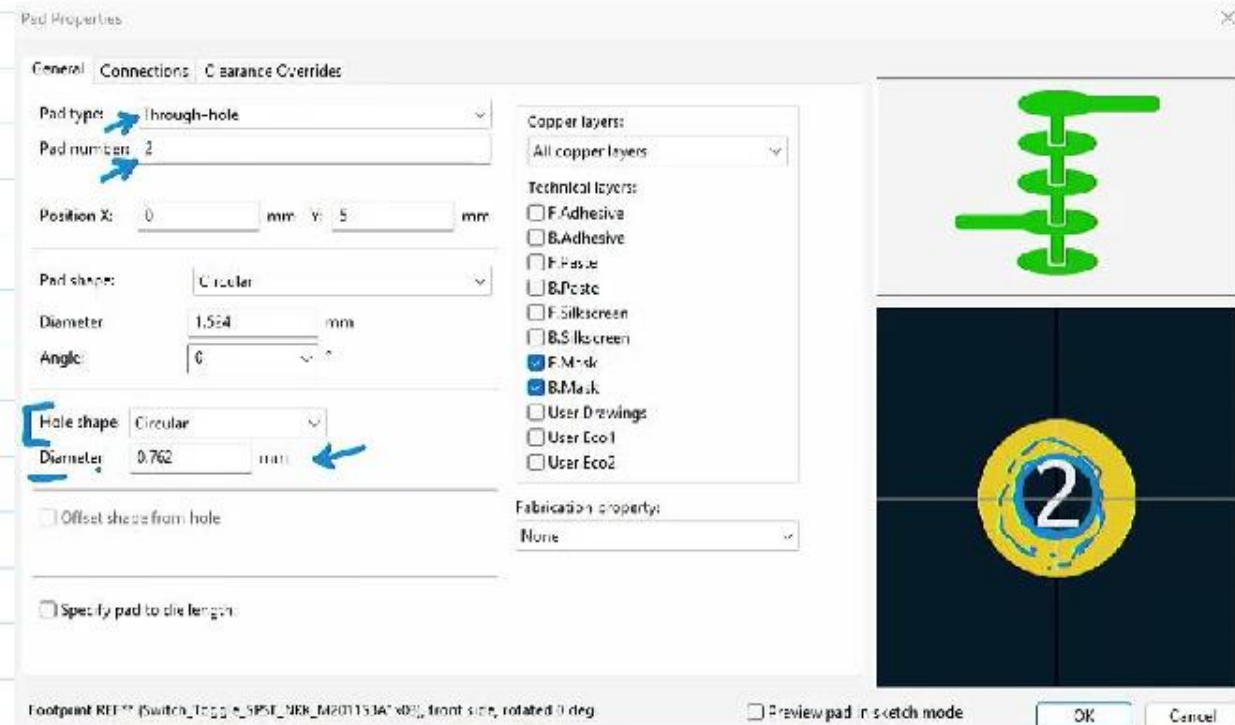
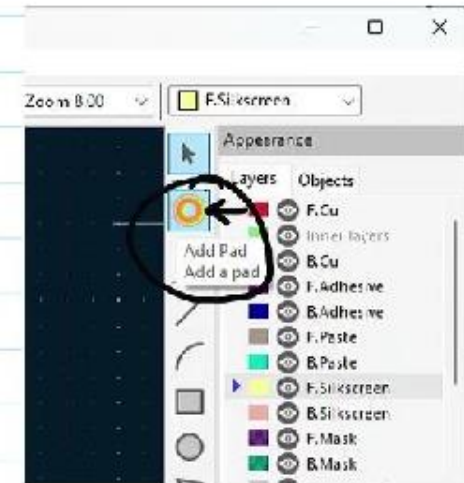


## Footprint Pad :

Use the "Add a pad" tool in right toolbar.

press escape and double click to edit pad's properties.

change the pad number to 2. ←

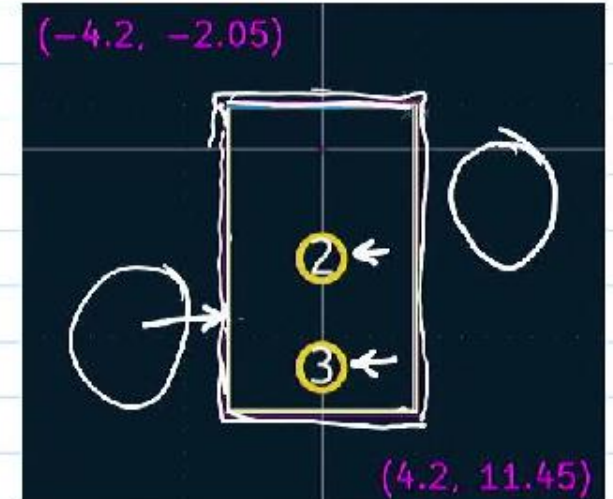


## Footprints Graphics :

F. Fab → Fabrication Layer

F. Silkacscreen → Front Silkacscreen Layer

F. Courtyard → Front Courtyard

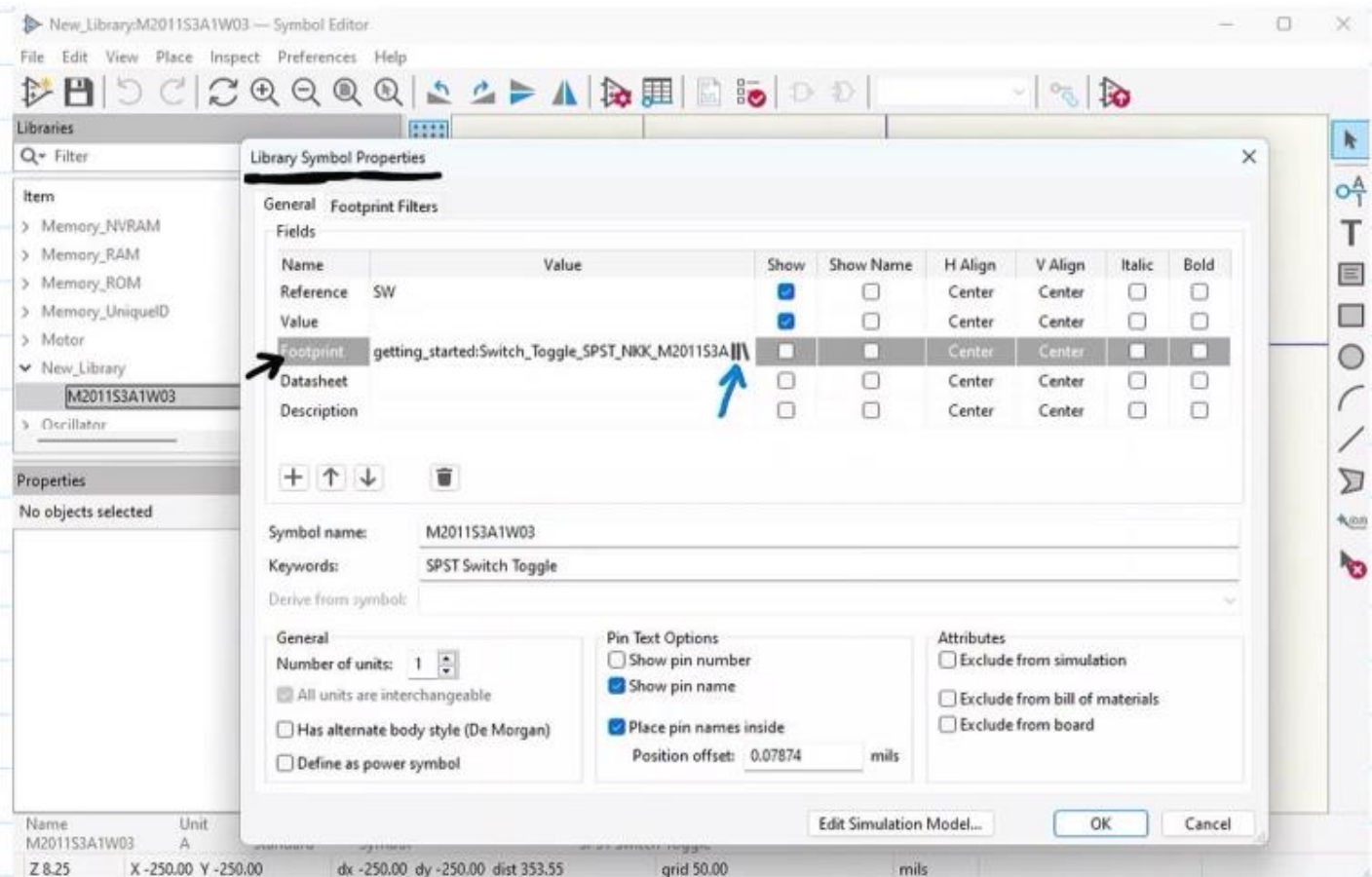




## Attaching Footprint to Symbol :

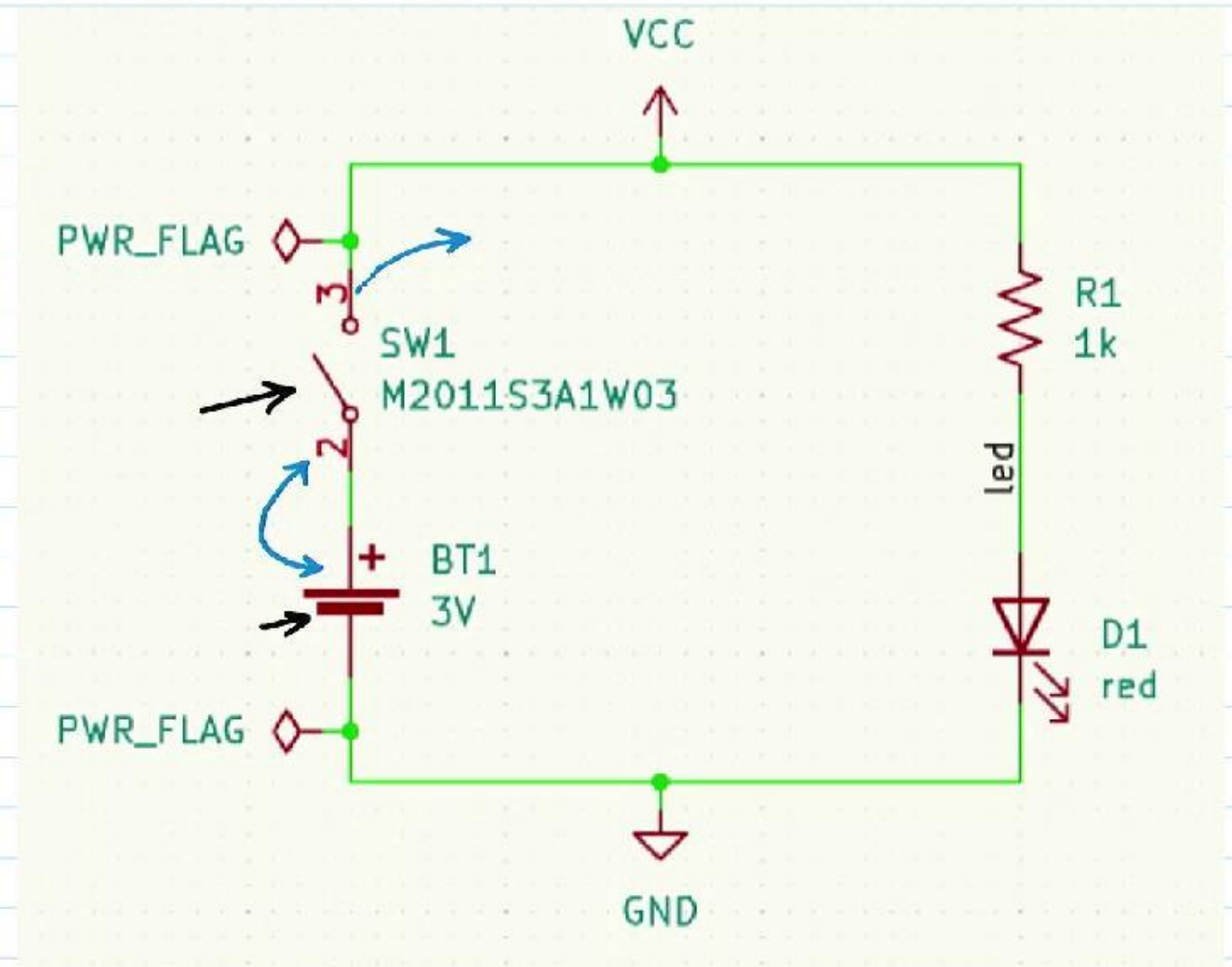
- Go to Symbol editor and open switch symbol.
- Edit symbol properties
- Click on footprint field
- browse for footprint library and select footprint of switch.

Now our symbol is attached to



## Using Custom Symbol into Schematic :

- open Schematic
- Add new symbol
- Make the connections
- RUN ERC



Add Switch to layout.

Tools → Update PCB from Schematic.

- delete unneeded traces.
- Route the new traces.
- RUN DRC.

