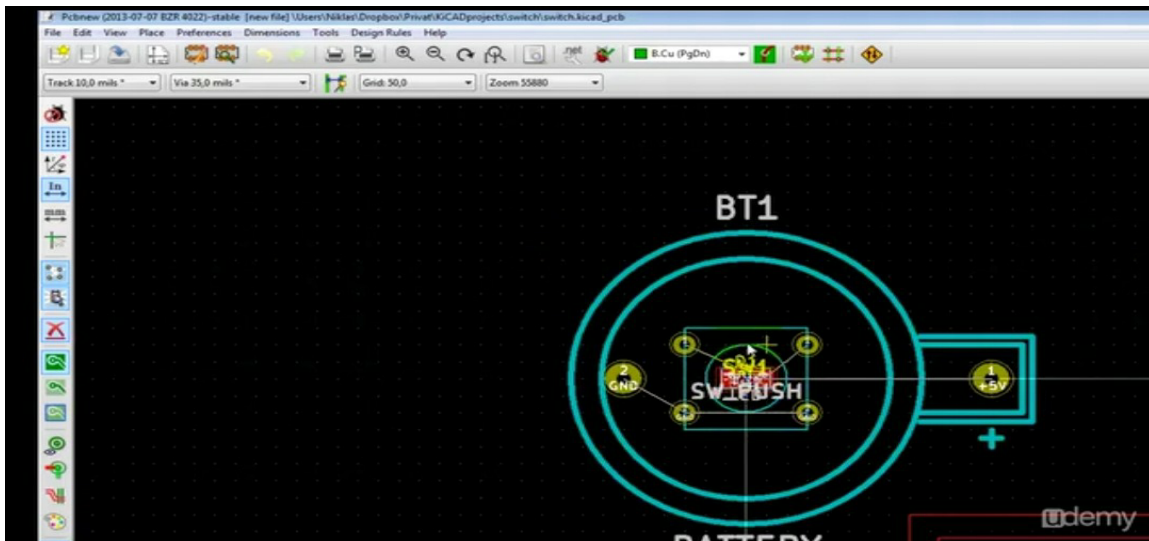


## DAILY ASSESSMENT FORMAT

Date:	10-06-2020	Name:	Abhishek
Course:	PCB Design using Kicad	USN:	4a17ec001
Topic:	1]Start a new project 2]Netlist and Footprint	Semester & Section:	6 & 'A'
Github Repository:	Abhishek-online-courses		

### FORENOON SESSION DETAILS

#### Image of session



Lectures

More



Section 1 - Up and running.



1

✓ Start a new project.



Video - 17:38 mins

2

Netlist and footprint association and placi...

Video - 16:12 mins

## Report –

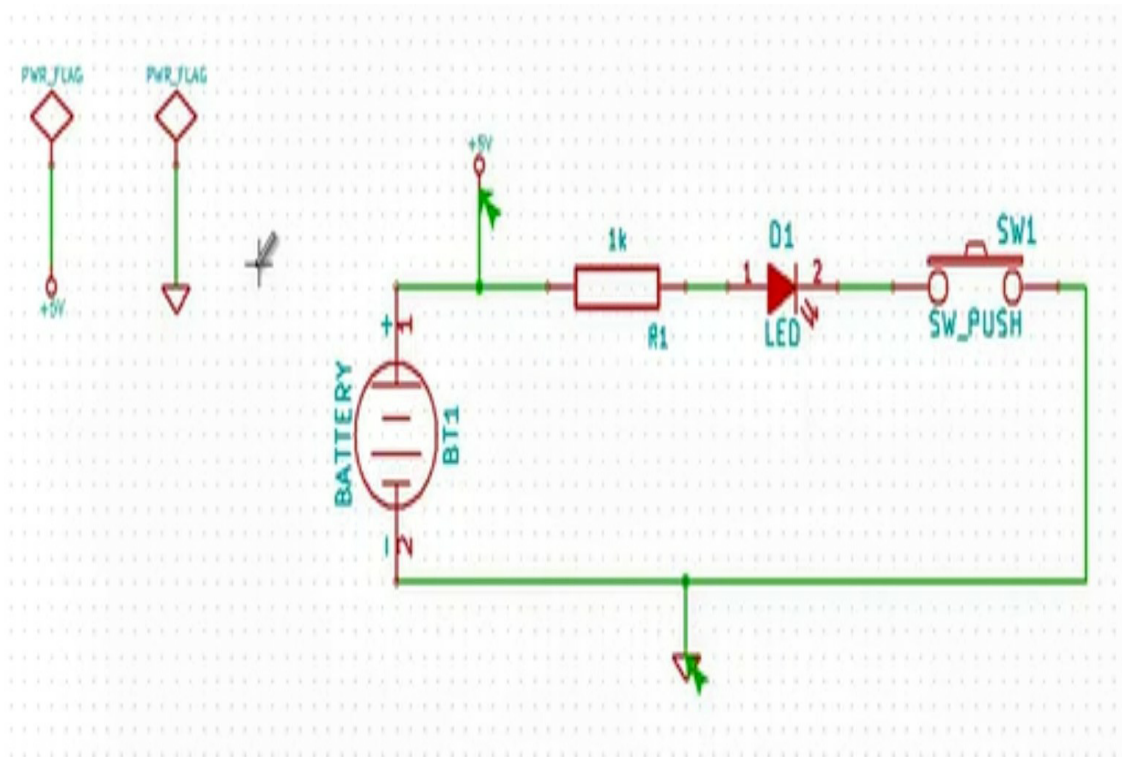
**KiCad** is a free and open source Electronic Design Automation (EDA) software package used to draw schematics (known as schematic capture) and for PCB design and layout.

### Start a new project :

- Steps in creating a new project,
  - ✓ Start Kicad and create a new project.
  - ✓ Create a new directory to hold the project files
  - ✓ A new project is created.

- **Schematic :**

In schematic, the actual circuit needs to be drawn using the different device symbols.



## Netlist and Footprint :

- **Footprint :**

- ✓ Once schematic is done, then the symbol needs to be connected to the required footprint.
- ✓ The connection is done via the pin number given to the pins in the symbol and the pad number given to the pads in the **footprint**.

- **Netlist :**

- ✓ Netlist is a file that tells KiCad which component terminals are connected to which other component terminals.

Generating a netlist is very important as it defines the actual layout of a pcb.

