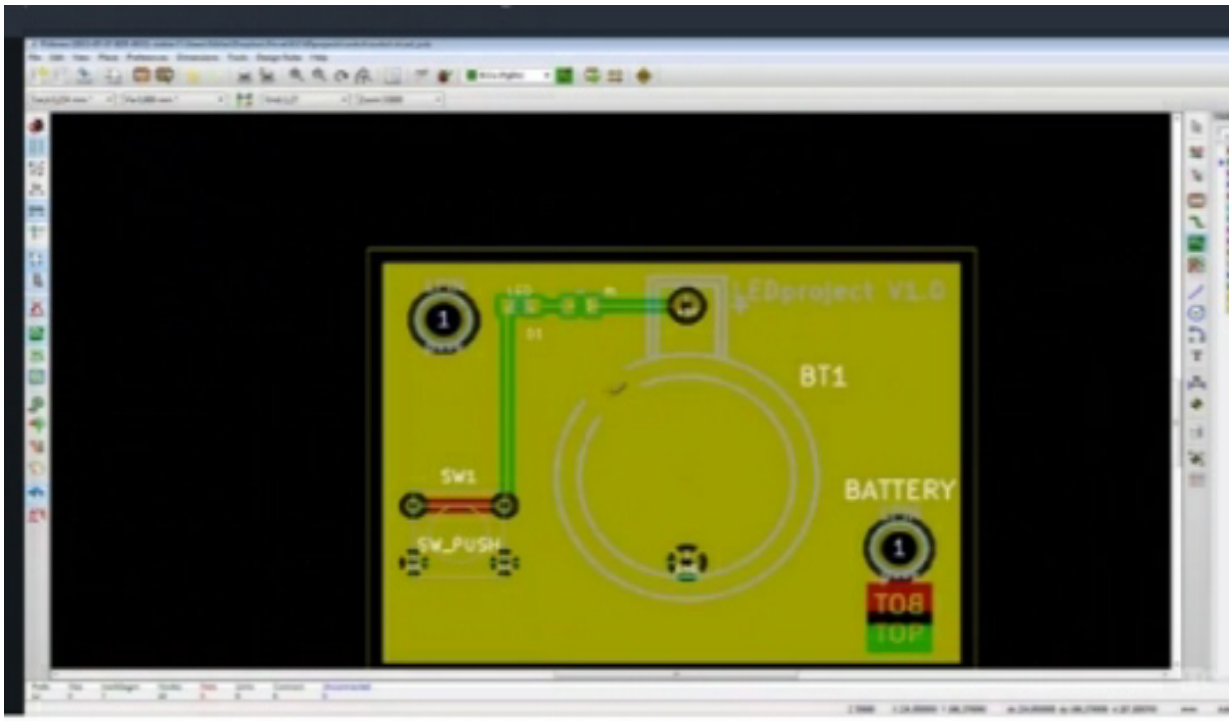
DAILY ASSESSMENT REPORT

Date:	11/06/2020	Name:	Abhishek
Course:	Learn KiCad: Printed Circuit Board Design	USN:	4AL17EC001
Topic:	1] Up and Running • Silk-screen and copper pour • Mounting holes • Create a library and put your own • component in that library	Semester & Section:	6th 'A'
Github Repository:	Abhishek-online-courses		

FORENOON SESSION DETAILS
Image of session



Report

Silk Screen

- The silkscreen is printed to the external surface of a PCB to aid in component identification and orientation. Typically this layer contains the component RefDes to locate components on the board after assembly.
- KiCad refers to the silkscreen layers as:
    - ✓ F.SilkS - Front silkscreen layer.
    - ✓ B.SilkS - Back silkscreen layer.

Copper Pour

- A copper pour or fill refers to an area on a printed circuit board where the original copper is not etched away, and remains in place, usually electrically connected to the Ground signal, producing a “Ground Plane”.
- This has a number of advantages, including decreasing the amount of etching fluid required during manufacturing, as well as reducing the amount of electrical noise and signal crosstalk experienced by the circuit elements.

Virtual Components

- Virtual components are those which have a footprint on the PCB (and may additionally have a schematic symbol) but do not have an associated physical component which needs to be loaded onto the board during assembly.
- Examples of virtual components include:
    - ✓ Mounting holes.

- ✓ Solder bridges.
- ✓ Net ties.
- ✓ Test points.
- ✓ Fiducial markings.

Custom libraries can be created in KiCad using the existing libraries which can be edited howerer the user wants it to be and make use of the custom library in the projects.