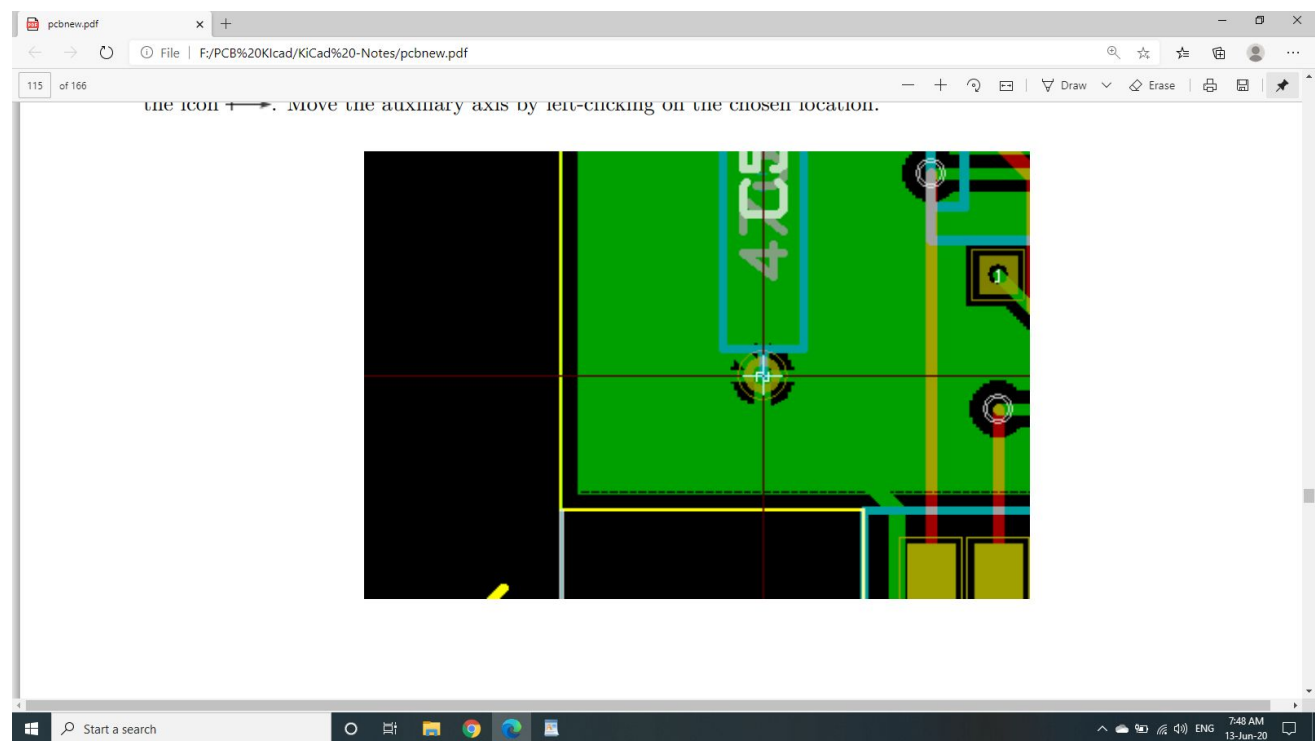


REPORT JUNE 13

Date:	13 JUNE 2020	Name:	Rakshith B
Course:	Kicad on Udemty	USN:	4AL16EC409
Topic:	Design Principles and basic Concepts,Additional Design Considerations	Semester & Section:	6th SEM B
Github Repository:	Rakshith-B		

FORENOON SESSION DETAILS

Image of session



Report –

When a footprint is added to a PCB, the entire footprint is copied into the PCB file (.kicad_pcb). This means changes to the footprint in the library do not automatically affect the PCB. This also means that you can individually edit footprints on the PCB without affecting other instances of the same footprint (either on the same PCB or on other PCBs). However, if you modify the library footprint, the next time you place an instance, it will not match existing footprints of the same name.

In Pcbnew it is possible to execute commands using various means: • Text-based menu at the top of the main window. • Top toolbar menu. • Right toolbar menu. • Left toolbar menu. • Mouse buttons (menu options). Specifically: – The right mouse button reveals a pop-up menu the content of which depends on the element under the mouse arrow. • Keyboard (Function keys F1, F2, F3, F4, Shift, Delete, +, -, Page Up, Page Down and Space bar). The Escape key generally cancels an operation in progress.

After having created your schematic in Eeschema: • Generate the netlist using Eeschema. • Assign each component in your netlist file to the corresponding land pattern (often called footprint) used on the printed circuit using Cvpcb. • Launch Pcbnew and read the modified Netlist. This will also read the file with the footprint selections. Pcbnew will then load automatically all the necessary footprints. Footprints can now be placed manually or automatically on the board and tracks can be routed. Pcbnew 38 / 154 4.3 Procedure for updating a printed circuit board If the schematic is modified (after a printed circuit board has been generated), the following steps must be repeated: • Generate a new netlist file using Eeschema. • If the changes to the schematic involve new components, the corresponding footprints must be assigned using Cvpcb. • Launch Pcbnew and re-read them

To open the Layers Setup from the menu bar, select Design Rules → Layers Setup. The number of copper layers, their names, and their function are configured there. Unused technical layers can be disabled.

12 technical layers come in pairs: one for the front, one for the back. You can recognize them with the "F." or "B." prefix in their names. The elements making up a footprint (pad, drawing, text) of one of these layers are automatically mirrored and moved to the complementary layer when the footprint is flipped. The paired technical layers are: Adhesive (F.Adhes and B.Adhes) These are used in the application of adhesive to stick SMD components to the circuit board, generally before wave soldering. Solder Paste (F.Paste and B.Paste) Used to produce a mask to allow solder paste to be placed on the pads of surface mount components, generally before reflow soldering. Usually only surface mount pads occupy these layers. Silk Screen (F.SilkS and B.SilkS) They are the layers where the drawings of the components appear. That's where you draw things like component polarity, first pin indicator, reference for mounting, ... Solder Mask (F.Mask and B.Mask) These define the solder masks. All pads should appear on one of these layers (SMT) or both (for through hole) to prevent the varnish from covering the pads. Courtyard (F.CrtYd and B.CrtYd) Used to show how much space a component physically takes on the PCB. Fabrication (F.Fab and B.Fab) The fabrication layers are primarily used for documentation purposes to convey information to, for example, the PCB maker or the assembly house.

The selection of the active working layer can be done in several ways: • Using the right toolbar (Layer manager).

- Using the upper toolbar.
- With the pop-up window (activated with the right mouse button).
- Using the + and - keys (works on copper layers only).
- By hot keys.

Selection of the Layers

for Vias If the Add Tracks and Vias icon is selected on the right hand toolbar, the Pop-Up window provides the option to change the layer pair used for vias: Pcbnew 48 / 154 This selection opens a menu window which provides choice of the layers used for vias. When a via is placed the working

(active) layer is automatically switched to the alternate layer of the layer pair used for the vias (unless Shift is held when adding the via). One can also switch to another active layer by hot keys, and if a track is in progress, a via will be inserted.

Drawing the board outline It is usually a good idea to define the outline of the board first. The outline is drawn as a sequence of line segments. Select Edge.Cuts as the active layer and use the Add graphic line or polygon tool to trace the edge, clicking at the position of each vertex and double-clicking to finish the outline. Boards usually have very precise dimensions, so it may be necessary to use the displayed cursor coordinates while tracing the outline. Remember that the relative coordinates can be zeroed at any time using the space bar, and that the display units can also be toggled using Ctrl-U. Relative coordinates enable very precise dimensions to be drawn. It is possible to draw a circular (or arc) outline:

1. Select the Add graphic circle or Add graphic arc tool
2. Click to fix the circle centre
3. Adjust the radius by moving the mouse
4. Finish by clicking again.

Importing the DXF file into KiCad The following steps describe the import of the prepared DXF file as a board shape into KiCad. Note that the import behaviour is slightly different depending on which canvas is used. Using the "default" canvas mode:

1. In the File menu, select Import and then the DXF File option.
2. In the Import DXF File dialog use Browse to select the prepared DXF file to be imported.
3. In the Place DXF origin (0,0) point: option, select the placement of DXF origin relative to the board coordinates (the KiCad board has (0,0) in the top left corner). For the User defined position enter the coordinates in the X Position and Y Position fields.
4. In the Layer selection, select the board layer for the import. Edge.Cuts is needed for the board outline.
5. Click OK.

Using the "OpenGL" or "Cairo" canvas modes:

1. In the File menu, select Import and then the DXF File option.
2. In the Import DXF File dialog use Browse to select the prepared DXF file to be imported.
3. The Place DXF origin (0,0) point: option setting is ignored in this mode.
4. In the Layer selection, select the board layer for the import. Edge.Cuts is needed for the board outline.
5. Click OK.
6. The shape is now attached to your cursor and it can be moved around the board area.
7. Click to drop the shape on the board.

Creating zones on copper layers :

Pad (and track) connections to filled copper areas are checked by the DRC engine. A zone must be filled (not just created) to connect pads. Pcbnew currently uses track segments or polygons to fill copper areas. Each option has its advantages and its disadvantages, the main disadvantage being increased screen redraw time on slower machines. The final result is however the same. For calculation time reasons, the zone filling is not recreated after each change, but only:

- If a filling zone command is executed.
- When a DRC test is performed.

Copper zones must be filled or refilled after changes in tracks or pads are made. Copper zones (usually ground and power planes) are usually attached to a net. In order to create a copper zone you should:

- Select parameters (net name, layer...). Turning on the layer and highlighting this net is not mandatory but it is good practice.
- Create the zone limit (If not, the entire board will be filled.).
- Fill the zone. Pcbnew tries to fill all zones in one piece, and usually, there will be no unconnected copper blocks. It can happen that some areas remain unfilled. Zones having no net are not cleaned and can have insulated areas.

Final preparations The generation of the necessary files for the production of your printed circuit board includes the following preparatory steps.

- Mark any layer (e.g., top or front and bottom or back) with the project name by placing appropriate text upon each of the layers.
- All text on copper layers (sometimes called solder or bottom) must be mirrored.
- Create any ground planes, modifying traces as required to ensure they are contiguous.
- Place alignment crosshairs and possibly the dimensions of the board outline (these are usually placed on one of the general purpose layers). Here is an example showing all of these elements, except ground planes, which have been omitted for better visibility

Overview of Footprint Editor

Pcbnew can simultaneously maintain several libraries. Thus, when a footprint is loaded, all libraries that appear in the library list are searched until the first instance of the footprint is found. In what follows, note that the active library is the library selected within the Footprint Editor, the program will now be described Footprint Editor enables the creation and the editing of footprints:

- Adding and removing pads.

- Changing pad properties (shape, layer) for individual pads or globally for all pads of a footprint.
- Editing graphic elements (lines, text).

- **Editing information fields (value, reference, etc.).**
- **Editing the associated documentation (description, keywords). Footprint Editor allows the maintenance of the active library as well by:**
- **Listing the footprints in the active library.**
- **Deletion of a footprint from the active library.**
- **Saving a footprint to the active library.**
- **Saving all of the footprints contained by a printed circuit. It is also possible to create new libraries. The library extension is .mod.**