Steps of AIoT based PCB design

To design a PCB kindly follow the following steps.

Install the KiCAD 0.8 software given to the drive.

1. **Schematic Design:-** Open schematic page – go to new schematic – create new schematic – name it – save it.

    a. Click on Opamp symbol (right side 3rd symbol of the screen) to install the component library.

    b. Check the data sheet to know about the component we require.

    c. Search component library to draw the component.

    d. **If we don't find our required component we have to draw it. To draw the new component -**
    Click symbol editor upside 11th right of the schematic page – file (upper left corner) – new library – global – name the library – new created library will appear on the left side – right click on it – new symbol – name it – take pins from right side of the screen – click on it to put name and number – put it on lines – take rectangle (forth right from top) and make border of pins – double click on border line – fill with background colour – save it – Go to schematic page – go to add library symbol – take created symbol and put it.

    e. Now connect the pins directly or use the input/output symbol from the right screen (12th from up) – name it – select input or output – connect it.

    f. Now if there are similar components or any copied component, the component name could be the same. That can cause some problems while designing. To change the name go up to the schematic page – tools (6th from the left) – click annotate schematic – save it.

    g. Now your schematic is complete. You can take a printout of it from the up left print option.

2. Footprint Design:- First check the mechanical part size of the data sheet – go to footprint editor – search to the left side library search bar if the footprint is available – right click on the footprint – save as – new library (left down) – name it – global – save – add more component in the library – if component footprint is not available – footprint editor – new footprint (upper left) – footprint name – select through hole or SMD – click add pad (right side orange circle) – put on screen and press esc – double click on pad – pad number – set x,y (0,0) – set hole diameter 15% more than component pin/pad size – pad diameter will be 40% more than hole diameter – select the pad – copy – pest – esc – double click on it – change pad number – change x,y as given data sheet – same for the others – go to rectangle shape (right side 6th from up) – file – save as – previously made library or new library. Now go to the schematic page.

3. Layout Design:- Select and double click on component – footprint – select the footprint from library you have created – ok – tools (schematic page up) – click update schematic to PCB - your routing page will open.

4. Routing:- now place the component as required (as close as possible) – go file – board setup – pre-defined sizes – add track width – 0k – select layers – file – board setup – physical stackup – copper layers – select layers as you required – ok – go to routing page – click on layer to route – click route tracks – do routing – after complete routing take rectangle shape – make a border of the board – take via – place on the board edge as a mounting hole and change its size as required – go to view(left up 3rd) – click 3D viewer to see your board in 3D.

5. Gerber File:- To print our PCB we need to create a Gerber file. This gerber file we will send to print PCB. Click Plot (left up 5th) – output directory (select the folder where you want to save it) – generate drill file – select folder –click on Plot (right down) – ok – now go to selected folder and see the gerber files(12 nos of file will generate on the same time) – send them to print your PCB.