

21/2/2024 Netlist and Board Mechanical R.kumar

Mapping PCB Library with schematic :

- Open schematic library, go to tools → footprint manager.
- Here all the component list is shown that each and every element/component we have to map with PCB library.
- For that select one component then click add. Then we have to give name or we can give it by simply browsing by Browse option.

- Then click OK and OK. Then we can able to see 3D and 2D option left corner down.
- After adding all components give Accept changes (create ECO) option.
- Then it will open one window there give 'Validate changes' option. Then check whether all are green and Tick mark or not
- Then 'Execute changes' option. If there is any error it will show. After that give 'close'.
- Then in schematic library if you click on any component (For ex: C1) In <sup>Parameters</sup> Properties (right side) if you click 'show' option at footprint → 'Click here to display' then it will show the footprint.
- After mapping we need to check
  - ERC checking
  - Netlist Verification
  - Single pin Report
- ⇒ For ERC checking right click on 'project name' and validate PCB project training.
- It will show error if pin is not connected (or) footprint is not assigned to schematic etc.
- ⇒ For netlist verification we need to generate the netlist for that File → report → Netlist schematic then it will show the folder and press save.
- It will ask for two options to save 1) project 2) Document and we have to select 'RIN' netlist in netlist format then 'OK'.
- open that file using Notepad by right clicking.



- ⇒ For single pin Report right click on project → Add new to Project → output job file
- Then again right click on project and save
- After that we have to add Add new Reps in Report outputs.
- Then select Report single pin ULTs → [project]
- Then select any folder structure then give generate content.

### Import Netlist to PCB:

- right click on project → Add new to project → PCB
- Give any name for example (training) by right clicking 'pcbdoc'
- After that Design → Import changes from Training.Prjpcb (project name) → 'Yes'
- Then check for errors and validate changes, until it show "No Differences Detected" while we go Design → Import changes from (project name) is clicked.
- Then click 'OK'.

### Agenda:

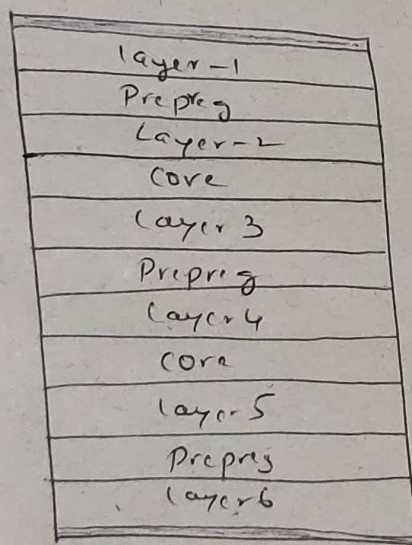
- 1) Board outline
- 2) Board Clearance
- 3) Drigin and Dimensions
- 4) Cutouts and Slots
- 5) Mounting Holes and Fiducials
- 6) Restriction Areas.



- Board outline: We can Define Board shape by Importing DXF file.
- That should be provided by mechanical team.
- Go to File → Import → DXF/dwg and open DXF file.
- One separate window will open. In that we have to take care of scale (mm, mil, inch) mm.
- In source layer name select all layers and mechanically and click 'OK'.
- Board clearance: Go to Design → Rules → Board outline clearance → right click on it → New rule → Under that open Board outline clearance.
- It consists of Arc, Track, SMD pad, THpad, via, Fill, Poly, Region, Text as 0.254 (default)
- After that we have to set the Origin.
- Origin can be either the left bottom corner of the board or the left bottom mounting hole.
- For setting origin edit → Origin → set and control the cursor to place origin where ever you want.
- Mounting Holes: Mounting hole is a hole, or slot on a PCB, used either to connect to chassis or for mechanical standoff. Like screws or bolts can fit in that holes and we should be very careful while giving clearance to that hole.
- Because if any component is there nearby to mounting hole and screw head may affect the component.



- Fiducials: These are alignment targets on an SMT board for the vision system in a pick and place equipment.
- Three global fiducial locations for each side of a board with SMT devices are required.
- Placement of global fiducials toward the extreme corners of the PCB offers the most benefit for alignment accuracy.
- A fiducial should be located no closer than 5mm to the board edge or tooling hole.
- Fiducials are small pads that are placed on a PCB to aid Pick & Place machines during component placement.
- Layer stackup:



A PCB layer stackup is an arrangement of copper and insulating layers that make up a PCB. These layers are arranged in a way to get multiple printed circuit boards on the same device.

- At their most basic, multilayer PCBs consist of at least three conductive layers.