

Libraries

generati

20/02/24

PCB Library

Rikumar

- Component Types
- Pad and its Structure
- Pad Placement and Pin Sequence
- Silk and Assembly
- Restriction areas
- Mapping with Schematic

Foot prints: It refers to a specific layout or configuration of a physical component on the board.
for example: If you take Surface Mount Technology

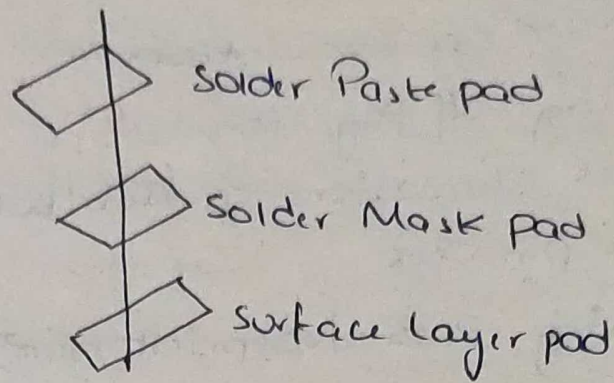


Fig: Surface Mount Technology

If you take Through Hole Technology

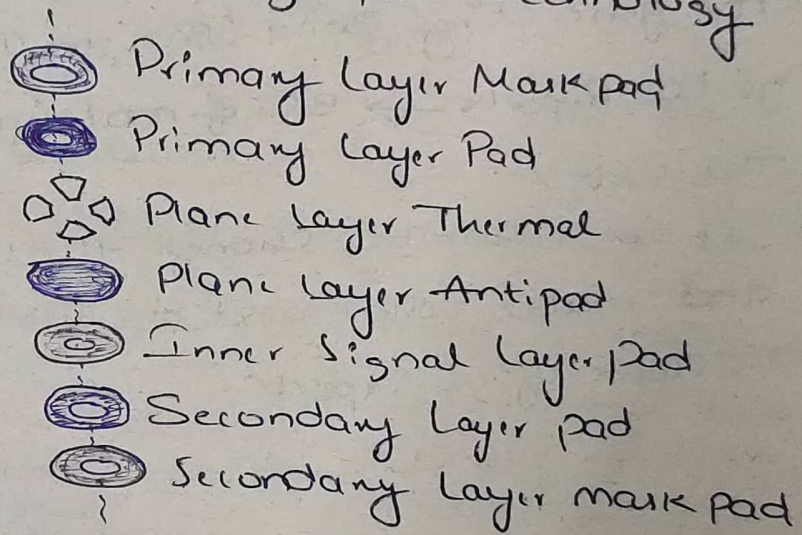
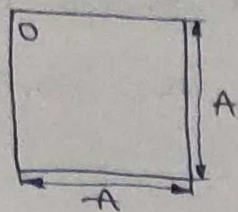


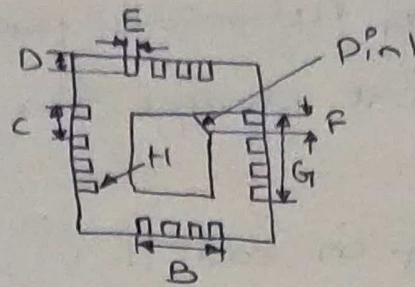
Fig: Through Hole Technology

Building the Library - Foot print:

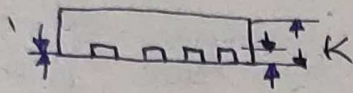
- Extract from Data sheet
 - Land pattern (if available)
 - Lead dimensions
 - Number of Leads
 - Numbering order
 - Pitch
 - Body outline dimensions (max)
 - Height
 - Keep out Areas



Top view



Bottom view



Side view

DIM	MILLIMETERS
A	4.00 ± 0.05
B	2.30 ± 0.05
C	0.650 BSC
D	0.55 ± 0.05
E	0.33 ± 0.05
F	0.45 ± 0.05
G	2.40 ± 0.05
H	0.020 Max
I	0.05 max
J	0.203 ± 0.008
K	0.9 ± 0.1

PCB library for capacitor (SMT):

- Creating PCB library in existing project i.e. schematic lib web
- Project → Add new to project → PCB library
- It also look and works as schematic library
- To add a capacitor here first we need to get land pattern based on the package for 1206 land pattern

→ Each pattern has three packages

Nominal — Class 2

Least — Class 3

Maximum — Class 1

→ If we go for first icon 'T'

3D-Bodies

Keeparts

Tracks

Arcs

Pads

Vias

Regions

Fills

Texts

Other

we can select "All-on"
or any one of
these

→ For creating rectangular pad go to shape select vector.
→ If we want change the units just press 'u' in keyboard.

→ By creating one pad if it is smt then three layers are created automatically (Feature of Altium).

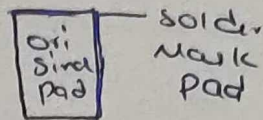
→ Solder mask will be 4 mils extra from original pad.



→ If we want to make 1:1

then go to manual and keep 0 mils by clicking on the layer we want to change.

→ Then it is like

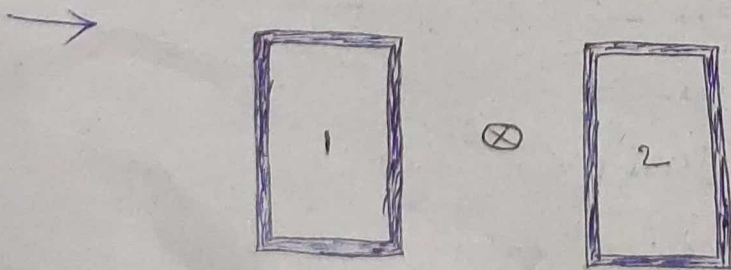


→ As per rule it is 4 mils and by selecting 'Rule' it will automatically change to '4 mils'.

→ update coordinates accordingly to origin ⊕.

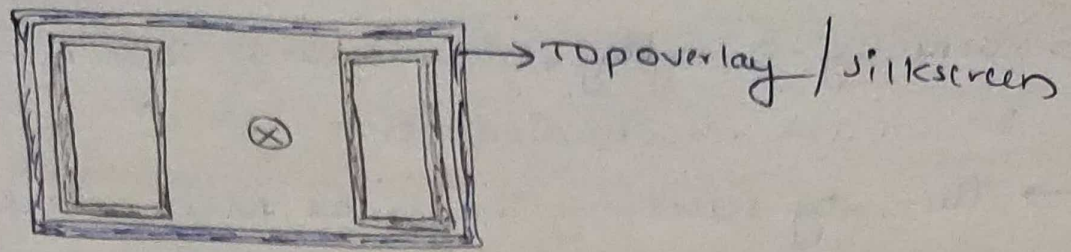
→ If you want to see the dimensions from point to point go to place → dimensions.

→ If in schematic library for capacitor! — 1 — 2 designator is 1 for pin 1 and 2 for pin 2. Then same only we have to use in PCB library capacitor also.



→ Then make silkscreen or top overlay.

→ Silkscreen refers to a layer on the PCB that contains markings, symbols and text for human-readable information.



- After that add string 'Designator'.
- In layers we have option to see what layers should be visible, like we can turn on/off the layers.
- If we want to add more layer, add in component layer pairs so that we can able to see layers by turning on/off.
- Let us take name of layer as Top Assembly / Bottom Assembly. This can add by right clicking on 'component layer pairs'.
- If we are creating layer on Top assembly the line thickness should be more than 6mil and < than is not acceptable.
- If silkscreen 1 of capacitor C1 and silkscreen 2 of capacitor C2 overlaps it gives error.
- Then we have to step file.
- step file is a 3-D model/body. For that we have to download 3-D / step file for capacitor.
- In DigiKey go to required component and click EDA/CAD models → Then download it by selecting 'step'.
- Then by extracting that file in downloads copy that ~~file~~ ^{path} and in Altium → place → 3-D body → paste the path.
- Then select that step file and open

- normally 3-D layer should be in mechanical layer so create mechanical 10.
- Then by selecting layer as Mechanical 10 in visibility drag that onto pads we created by taking help of origin \otimes
- After that just click '3' to view 3-D design.

PCB Design for Through Hole Technology (Three pin connector)

- For connectors there is no specific land pattern available that we want to calculate.
- Give 0.4 mil extra to the pin size so that if it is 1 we should add 0.4 and 1.4 mil ^{pad or} hole we have to create.
- If it is same pin can't insert in ^{pad or} hole right.
- Take Through hole pad and modify (x1y) and hole size change designator as 1
- Then take/copy three pads and change one pad shape to rectangle.
- for sm pad we made origin in the pads and for Through hole we make first pin as origin. Then give pitch (pin to pin gap) size also as (x1y) dimension.
- After that create outline in top assembly and top overlay and add .Designator.
- After that in EDA/CAD modes copy .stp file path after downloading and in Altium place 3-D body.
- Go to Tools → 3D body placement → Align face with board. so pins of connector is aligned with pad holes created.
- And adjust standoff height to perfectly insert the pins. (let as 0)

PCB Design for Through Hole Technology (LMR14030 IC)

- Find land pattern for LMR14030 IC in datasheet
- Here solder mask to pad gap is 0.07 mil
- If they mention like $\left(\begin{array}{l} 8 \times (1.55) \\ 8 \times (0.6) \end{array} \right)$. That means eight pins have same dimensions of length & width.
- Similarly we can add PCB library footprint for this one by clicking 'add' option. Then give name and height.
- And create one pad with (x,y) 1.55, 0.6 and solder mask 0.07 (manually we can add).
- After that copy 8 pads as those dimensions are same.
- Then give Designators 1, 2, 3, 4, 5, 6, 7, 8, 9 as in land pattern.
- Then create the IC according to land pattern and download the 3-D step file and place → 3-D body.
- For IC top and bottom silk screen lines are enough. No need to give entire outline.

*** PCB Libraries completed ✓

