

20/2/2024

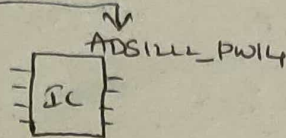
R. Kumar

# PCB Design Flow, Symbol Library and Schematics creation:

## Building the symbol:

→ While we are creating schematic symbol we need to maintain - 100 mils Grid

→ Symbol Name should be max 30 characters



→ Symbol Parameters: Reference Designator

Value

PCB footprint

Manufacturer Part Number

Manufacturer Name

Description

Power/Wattage / Voltage / Tolerance

BOM: "Bill of materials", It is a comprehensive list of all the components used in a particular electronic design or Product.

→ It provides detailed info about Description, Designator Quantity, MFG, MFG\_PN and soon.

(MFG\_PN → manufacturer Part number)

→ For components description better go with "Digikey"

How to create a project in Altium?

File → New → Project → It will ask project name & path to save data.

→ Under project if we want to work on library.

Here are the options

- Integrated Library
- Schematic Library
- PCB Library
- Foot Vio Library

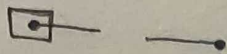


→ We can add and delete the components that we created left-down corner Place Add Delete Edit

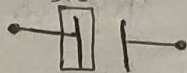
→ For capacitor we can give pin length 100mil and for ICs we can give 200mil.

→ For properties we have  $\left\{ \begin{array}{l} \bullet \text{ Location} \\ \bullet \text{ Properties} \\ \bullet \text{ Symbols} \end{array} \right\}$  for modifications.

→ While creating the pin, make small dot outside so that we can give further connections to that pin.



→ As per standards of Altium while we creating line give color as Blue color.



→ If we are creating a capacitor then in Designator item ID should be copied and pasted from Digkey

→ Designator should be e.g. If we are creating capacitor

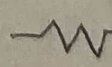
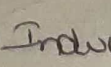
→ "?" is using for future Annotation purpose.

→ Description also copied and pasted from Digkey

→ If we want to add more parameters then click add right down corner and add like Manufacturer

Manufacturer product no.  
Value  
Voltage

→ If we want to reflect the parameters in library or screen just click on the parameter you added.

→ Like that we can create resistor  Inductor 

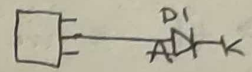
Note: For inductor we are and adjust radius in parameter.

→ For IC make sure that grid should be 100mil so that pin placement will get in a proper way

\* → If you want to change the grid just type 'g' in Keyboard it can change to 10mil, 50mil, 100mil...



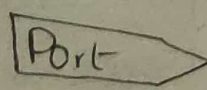
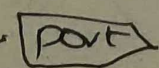
- For IC there is a need of specifying the pin whether it is input or output.
- For that select on pin we added and go to description and change.
- Better keep all i/p pins left side and all o/p pins right side the IC. so that schematic should be clear.
- If we don't know the pin keep it as "passive" only.
- If we want to add the symbols / use the symbols that are created in project go to panels → navigation and double click on component.
- While connecting the components, the netname should be there on wire for example Tx, Rx ...



- Default netname for above fig is 'Net-A' or if we connect wire at K default netname will be 'Net-K'.
- In schematic, if we want to select something go to 'selection filter'. There have many options like
 

Components	Net labels	Texts
Wires	Parameters	Drawing objects
buses	Ports	Other
Sheet Symbols	Power ports	
Sheet Entries		

- for annotation: go to tools → Annotation → Annotation schematics → Update changes list → Accept changes → Validate → execute and close.

-  It is used to connect the page. If we want to clarify whether it is connected or not go to annotation and there we can see. If, in 1st page up to C<sub>9</sub> are there and we added 2 more in next page using , Then annotation starts from

C10 to C11.

→ Netlist Error: Once we complete the schematics we have to generate netlist.

→ Netlist Error occurs due to

- 1) Net Naming Convention
- 2) Missing Off Page Connectors
- 3) Special characters in Attributes
- 4) Single pin Nets
- 5) Missing PCB footprint names

→ For netlist we have to validate first. For that right click on the schematic → Validate PCB project

→ For generating Bom (Bill of materials) we have to go to Reports → Bill of materials. Here we can see the list.

→ If we want to change the list go to columns and enable what and all we want.

→ Then give export.

\*\*\*  
Till now In PCB design flow

✓  
create  
logical  
libraries

✓  
schematic  
Design

✓  
BOM X  
Netlist  
generati