Altium Designer 6 Integration Platform

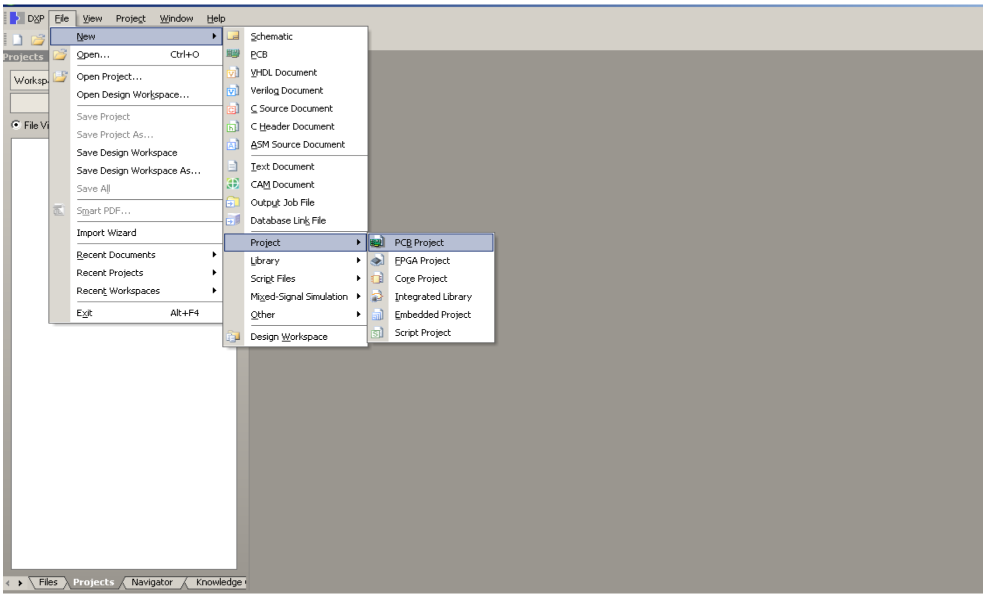
Project

Altium Designer 6 allows you to access all documents related to a design via its user interface. The basic of any design is the **project**. The project links the elements of your design together, including the source schematics, PCB, netlist, any libraries or models you want to keep in the project.

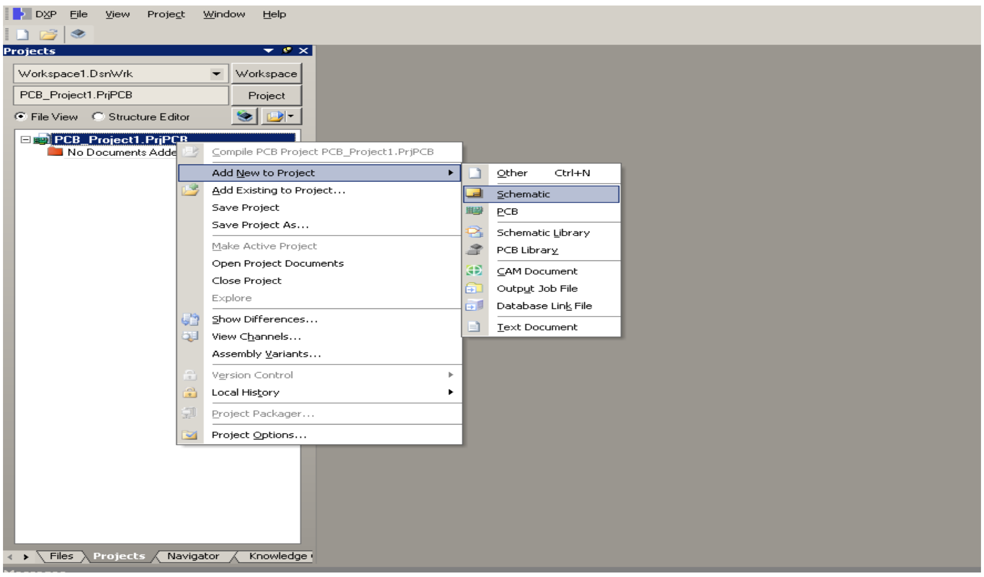
There are different types of project in Altium Designer: **PCB Project, FPGA Project, Core Project, Embedded Projects, Script Project and Library Packages** (the source for an integrated library). Related project can also be linked under a common Workspace, giving easy access to all files related to a particular design. You can add document to your project, example add Schematic, PCB, Cam Document, PCB Library, Schematic Library, etc.

Creating a New Project

1. Select the **File>>New>>Project>>PCB** Project menu to create new PCB project.



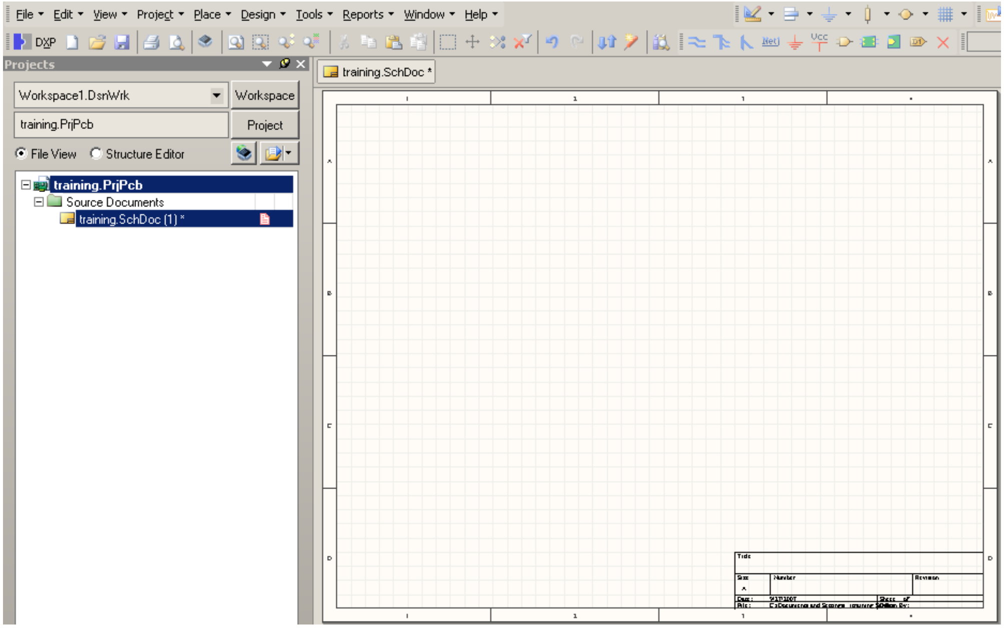
2. Now add the schematic by **right click** at project header (**PCB_Project1.PrjPCB**) choose **Add New to Project>>Schematic**



BW

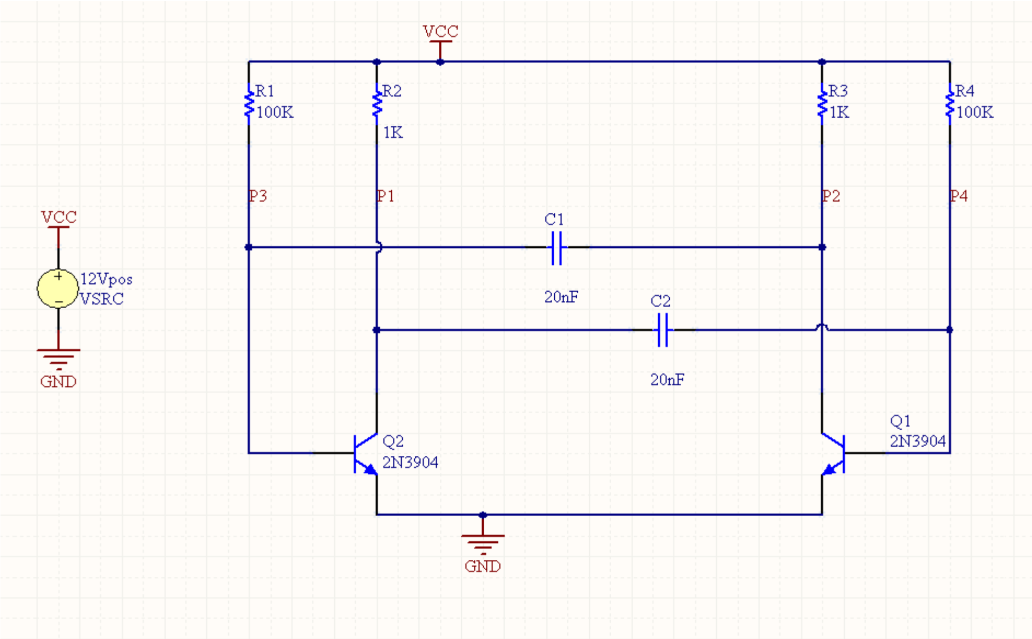
3. You also can add new schematic by go to **File>>New>>Schematic**

Save the Project and Schematic file to your folder.



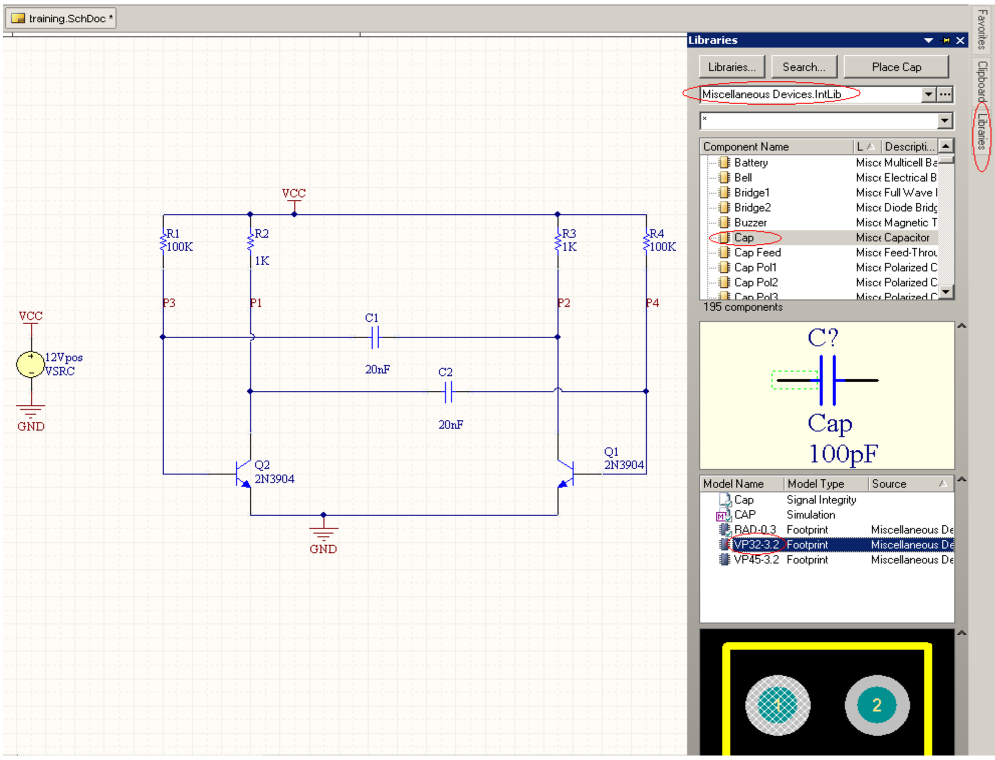
Draw Schematic

4. To draw schematic as shown below, we need to browse the schematic symbol from Library.



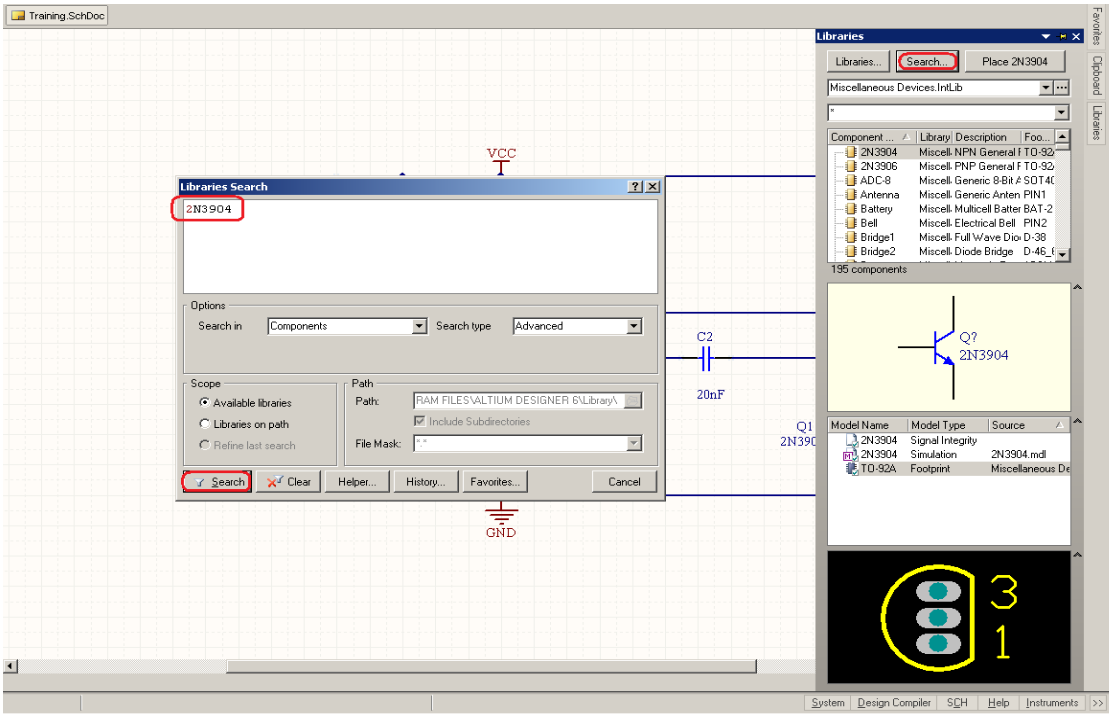
5. Library is placed at Right side on project panel. Click at **Libraries**, and search your schematic symbol. Library from Altium is Integrated Library, it means inside the library, it have **schematic symbol, footprint, simulation model, signal integrity, and PCB3D**. Library is grouped based on the name of manufacture. i.e. microchip, Atmel, maxim, etc.

6. Inside one integrated library, it contains few schematic symbols, and each schematic symbol, it have Footprint, PCB3D, Simulation model and Signal Integrity (note: **not all schematic symbol have simulation and signal integrity model, depends from manufacture**).



7. **Double Click** on 'Cap' or press **"Place Cap"** button, and now you are on component placing mode. Before the placement, press **Tab** to view the properties and edit the attribute for this component. Do the same step for each component. To exit from the current mode, **Right click** or press **Esc** key.

8. If you don't know where is your schematic symbol is placed, you can **search** it use search engine provided by Altium. Key in your keywords component, i.e. **2N3904** and click **Search**. After search engine found your keyword component, you can press **stop button** to stop searching. When you want to place the component and the library is not installed, it will automatically prompt you to install, choose to install the library. Do the same step for another unknown schematic symbol.



Notes:

- To place Wire, select **Place>>Wire (P, W)**. 
- To place P1, P2, P3, and P4 use net label. Net label can be accessed by go to **Place>>Net Label (P, N)**. 
- To place Power port **(VCC and GND)**, click on the icon 
- To place the **12Vpos** click on this icon , and choose **12Vpos**.

Shortcut Key

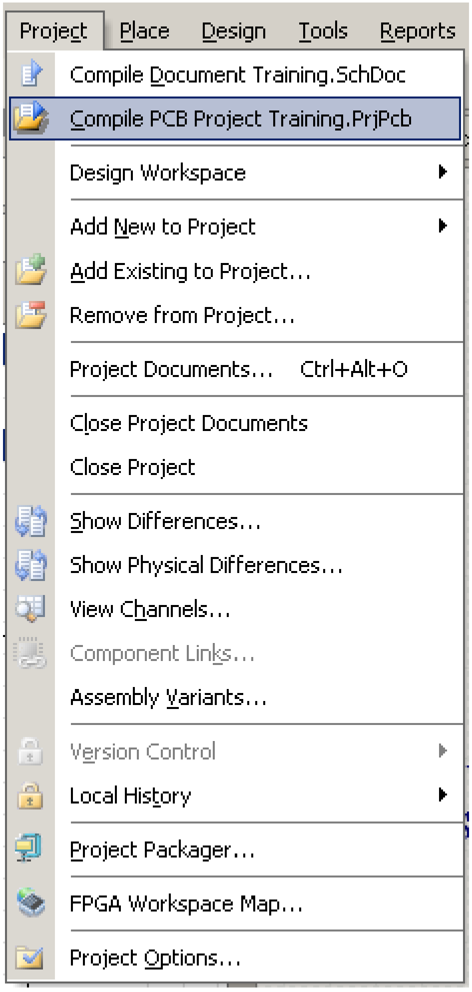
- **Page Up** => Zoom In
- **Page Down** => Zoom Out
- **TAB** => Edit properties before placing
- **Esc** => Escape from current process
- **CTRL+A** => Select all
- **CTRL+C** => Copy selected objects
- **CTRL+X** => Cut selected objects
- **CTRL+V** => Paste copied/Cut objects
- **Spacebar** => Rotate object under move object command
- **Shift+Spacebar** => Change Wire mode under Wiring/ Routing mode
- **X** => Flip or mirror Component in X Direction/axis
- **Y** => Flip or mirror Component in Y Direction/axis
- **Backspace** => Remove the last vertex when placing a wire/bus/line/polygon

Common Mouse Controls

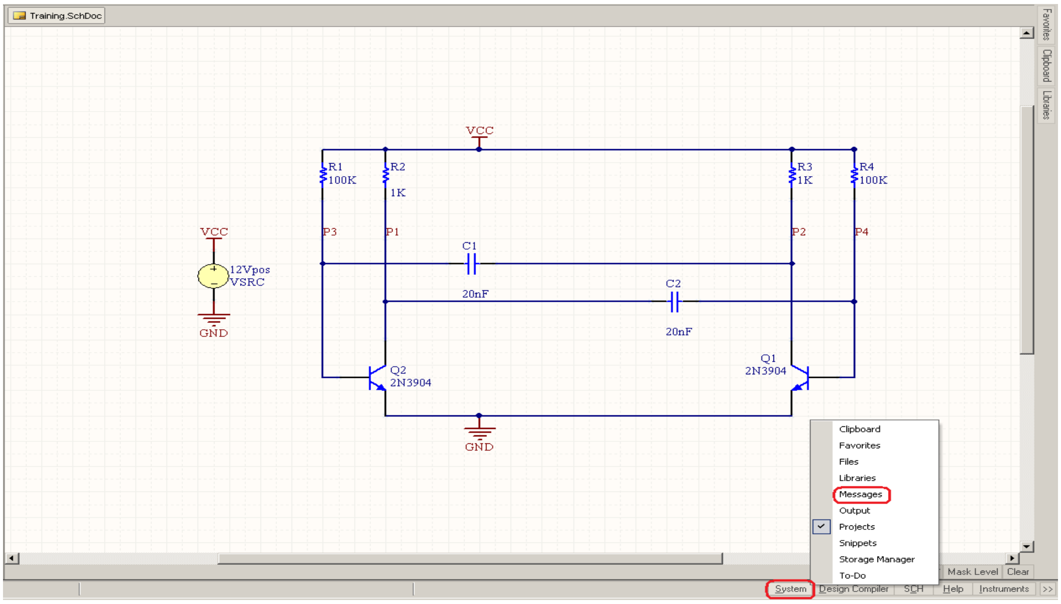
- **Left Click on Object** => Select object
- **Left Click and Hold on object** => Select object and move.
- **Middle click and hold** => Pan Mode in PCB only
- **Right click and Hold** => Pan mode in both PCB and Schematic
- **Right Click** => Bring up optional Feature in both Schematic and PCB
- **Right Click on Object** => Bring up Additional Object Feature and Properties in both Schematic and PCB
- **CTRL+Scroll up/down** => Zoom in / Zoom out
- **Double click on object** => open properties of selected object

For Shortcut key, you can access from View>>Workspace panel>>Help>>Shortcut.

9. When schematic drawing is done, Compile the project, select **Project>>Compile PCB Project qqq.PrjPCB**. (qqq is your Project name) **(P, C)**

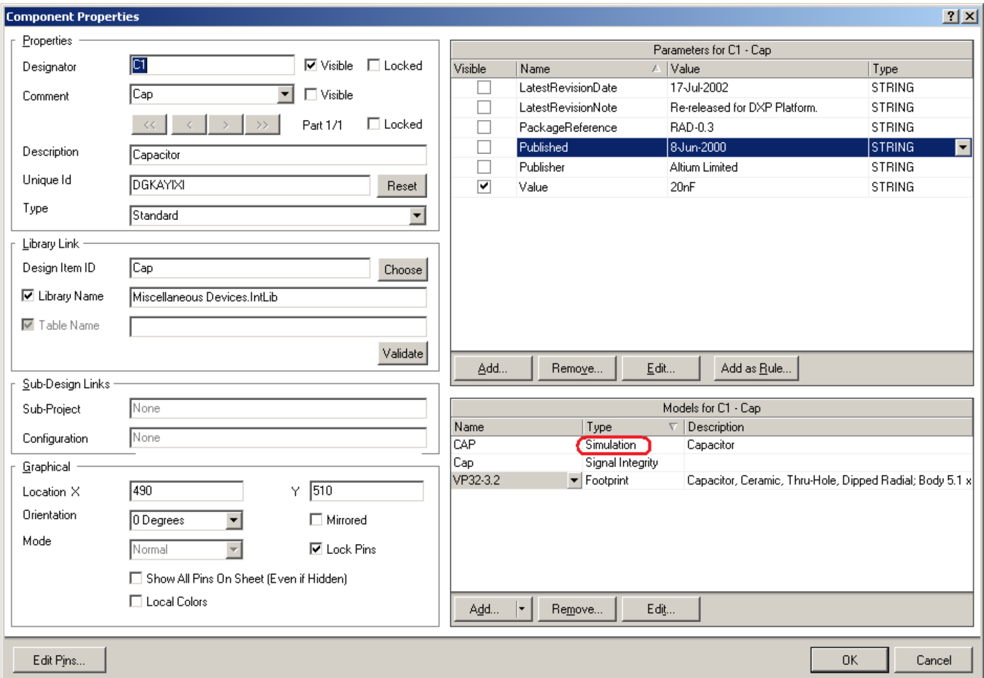


10. You can see your compile result from Message panel. Click at **System**, and choose **Messages** or go to **View>>Workspace Panel>>System>>Messages**



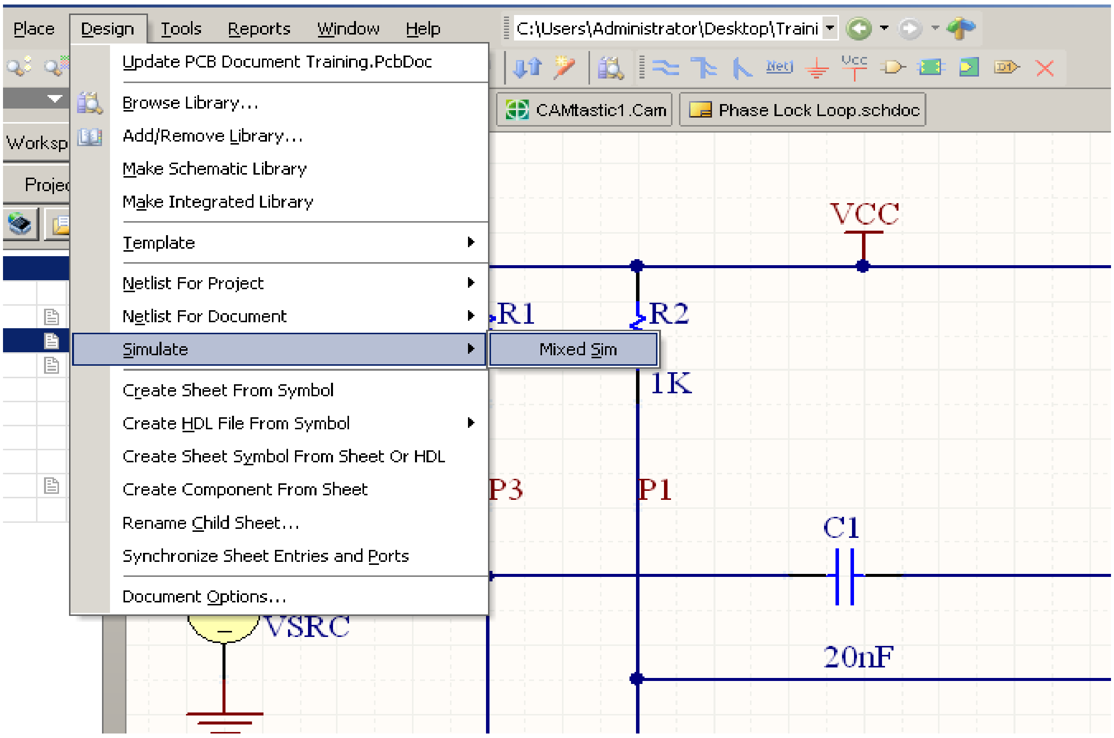
Run Simulation

11. To do **simulation**, make sure all of your schematic symbol have simulation model. From default library, Altium provide simulation for basic component.
12. **Double click** on component and component properties dialog will display

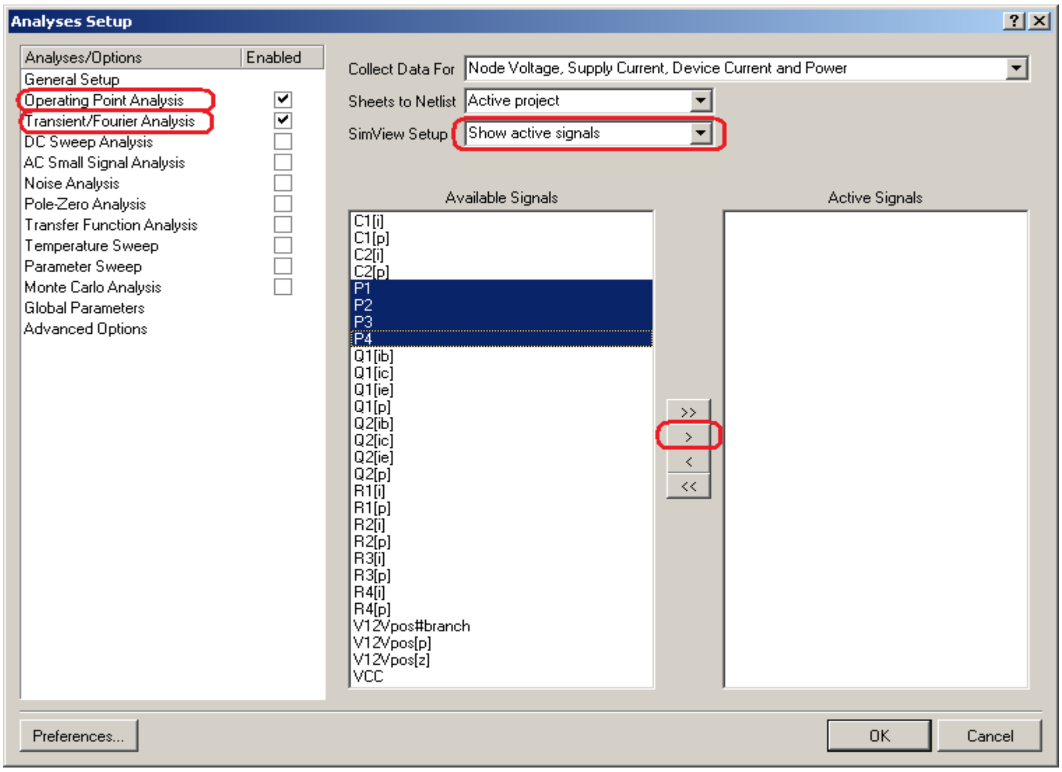


13. Inside the schematic properties, you can add the parameter for your schematic symbol, add the footprint, PCB3D model, simulation file, signal integrity and can add rule for your PCB design.

14. For parameter, you can use it for creating your **BOM (Bill Of Material) report**. All information about the component, you can add from here and when create the BOM, you can add this parameter to your BOM report.
15. After schematic is completed, go to **Design>>Simulate>>Mixed Sim (D, S, S)**

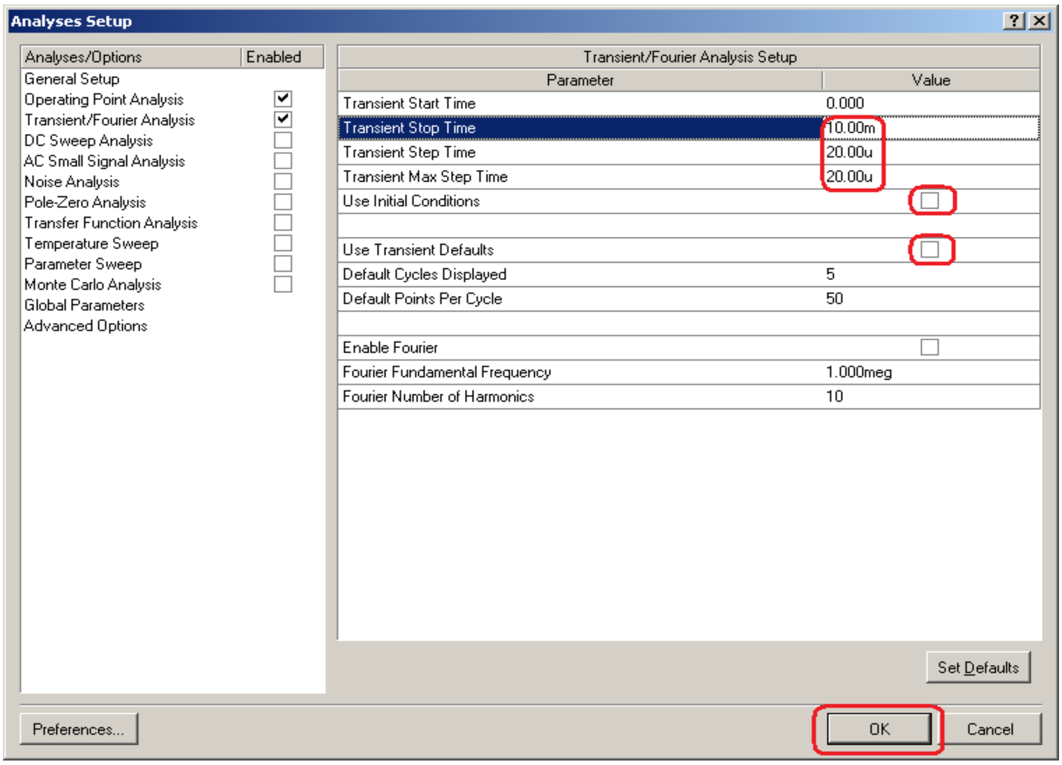


16. Select **P1, P2, P3, and P4**, **add to active signal by pressing '>'**, Change **SimView Setup** to **Show Acive Signals**.
17. Enable **Transient/Fourier Analysis and Operating Point Analysis**

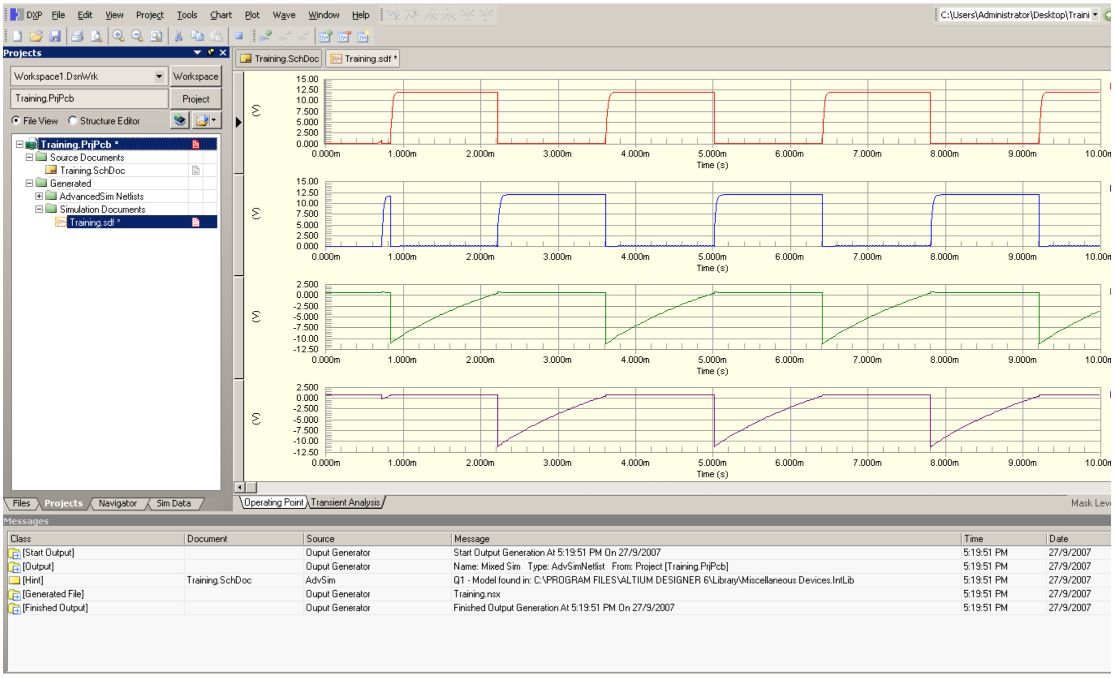


18. Click on Transient//Fourier Analysis, **Uncheck “Use Initial Condition” and “Use Transient Defaults” boxes**.

19. Change the value same as shown below.

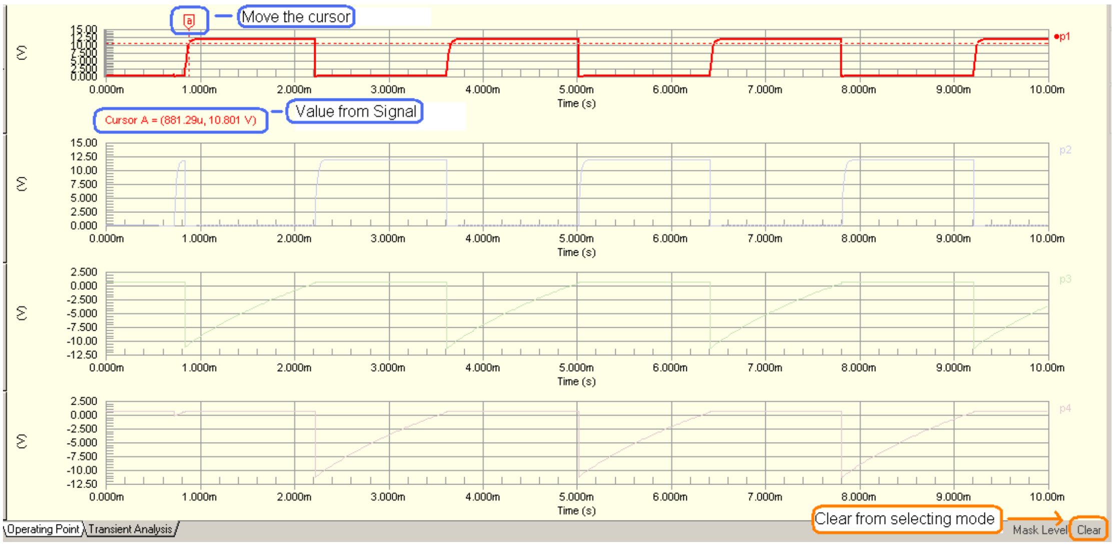


20. Press **OK**, and Altium will generate **simulation file (.sdf file)**

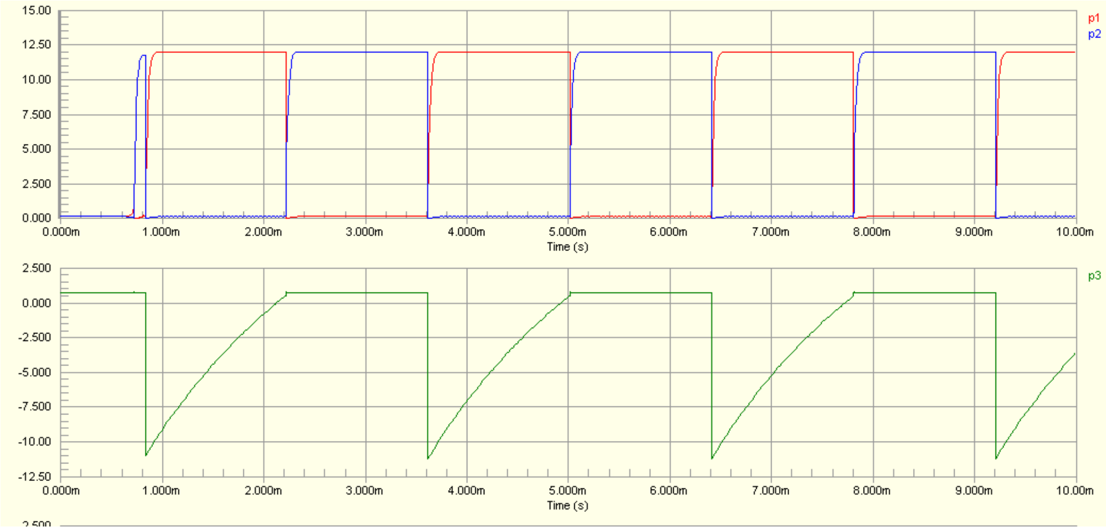


This is signal behaviour from schematic design based on your net label (**P1, P2, P3, P4**).

21. Click on one of the signal (i.e. P1), go to **Wave>>Cursor A**. move the cursor to point the value from signal.

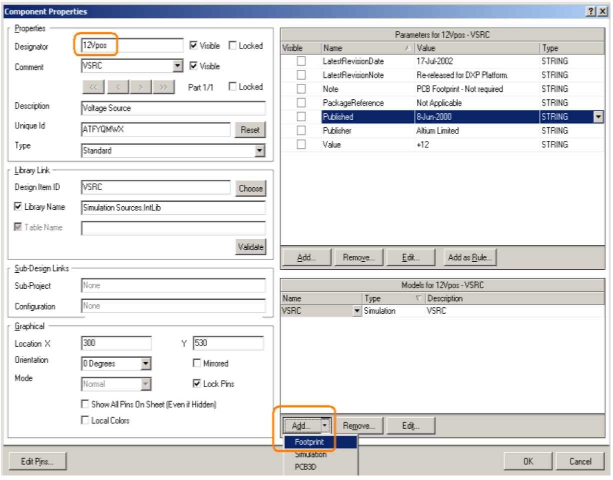


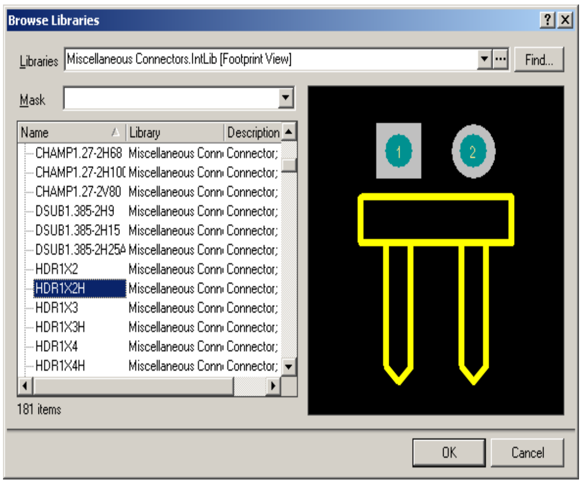
22. Clear from select mode by pressing '**Clear**' button, or use the short cut key **SHIFT+C**.
23. You can compare your signals, click and drag on one of signal to another plot. I.e. click and drag P1, and move to plot P2.



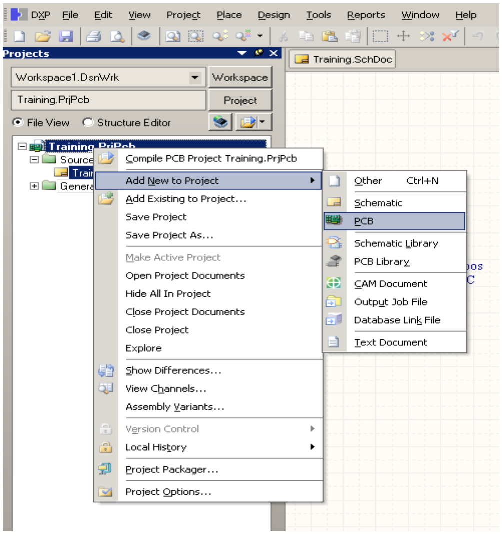
Create PCB

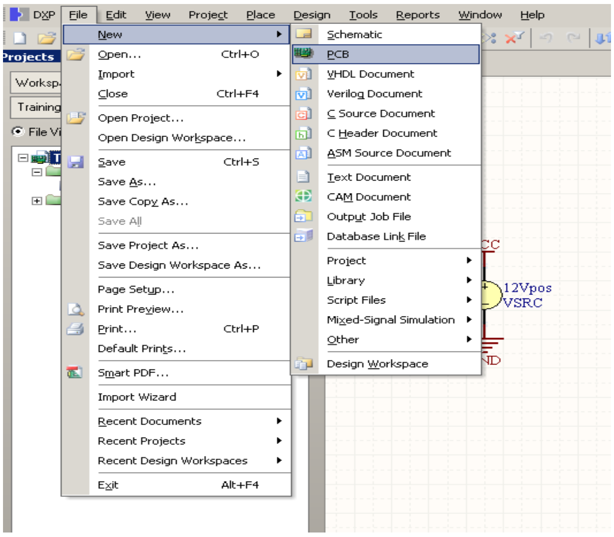
24. **Before you transfer your schematic design to PCB, you have to make sure all of your schematic symbol have footprint model**.
25. Add a **footprint for 12Vpos**, Double click at this component, component properties dialog will display, click at **Add>>Footprint**, Browse to **Miscellaneous connectors.IntLib** and search for 2 pins header (**HDR1X2H**).



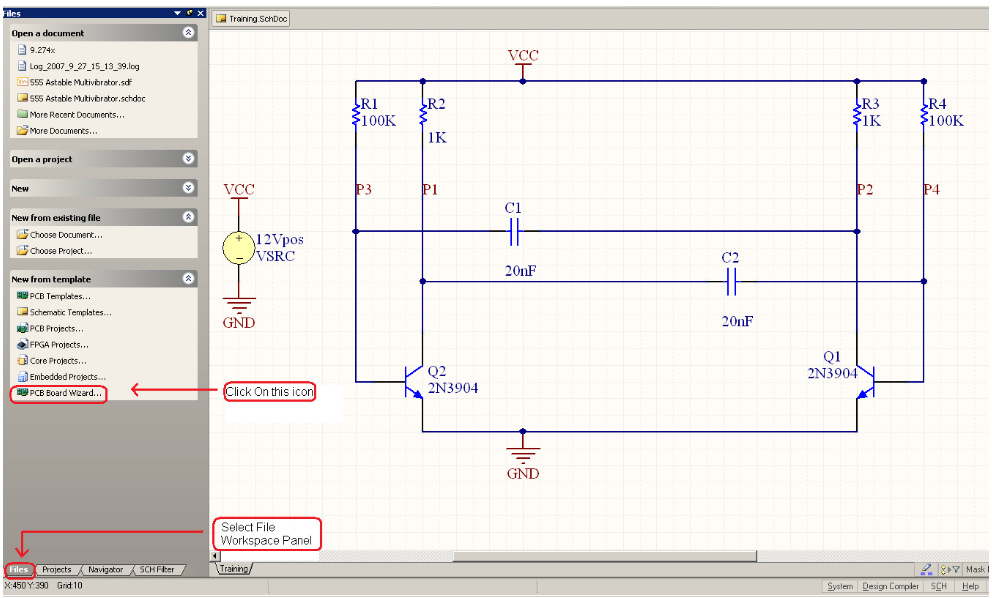


26. There are 2 ways to create new PCB Document, **manual** and use **PCB Board Wizard**,
27. Manual way is add PCB documents directly without set any rule and PCB shape. Go to **File>>New>>PCB**, or **right click at project header**, **Add New to Project>>PCB**

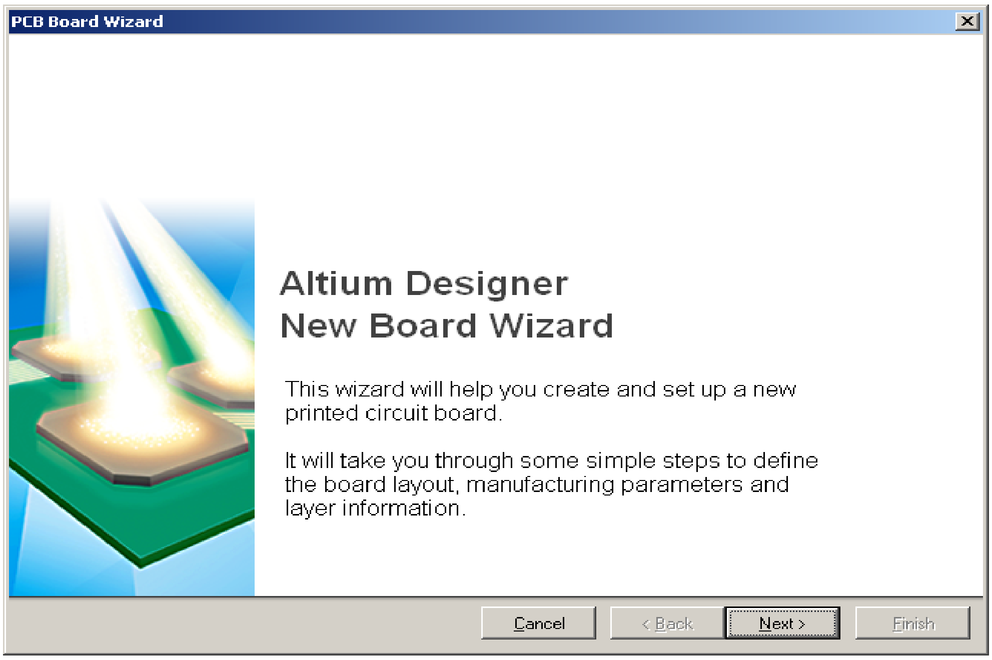


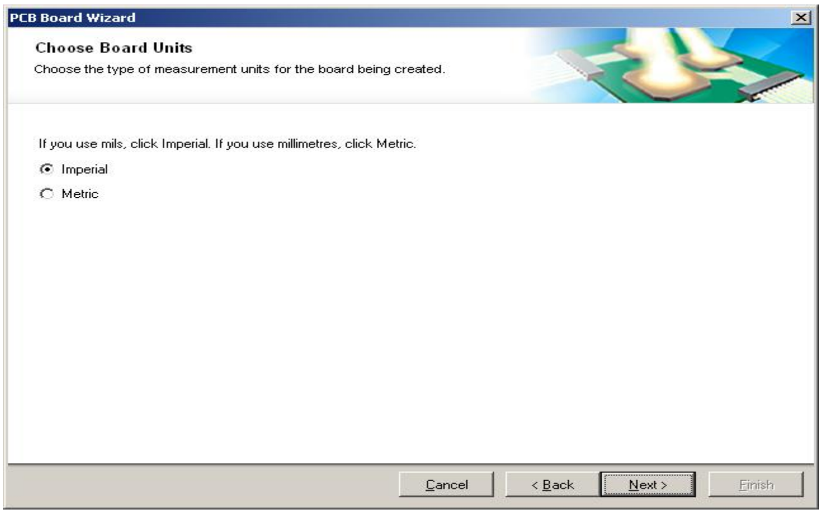


28. To use **PCB Board Wizard**, Select **File Tab** in **Workspace panel**. Click on **PCB Board Wizard icon**. 

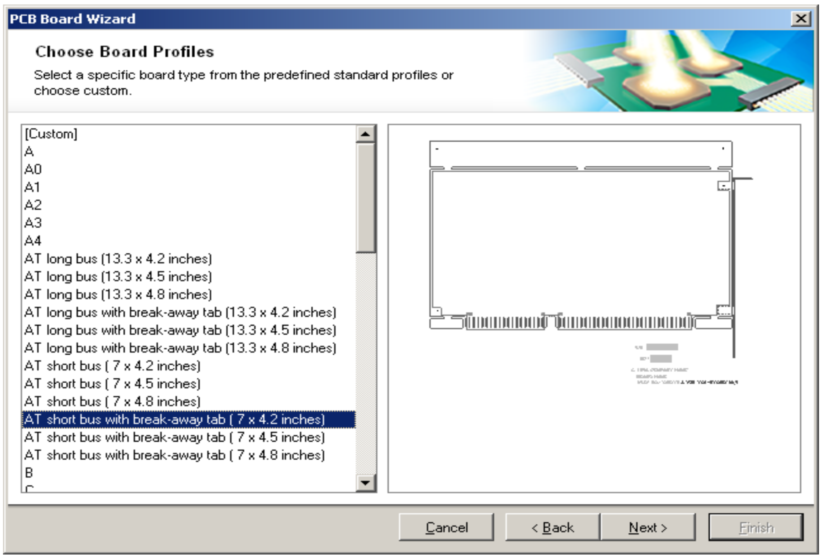


29. Wizard dialog will appear click Next to go to next setup.

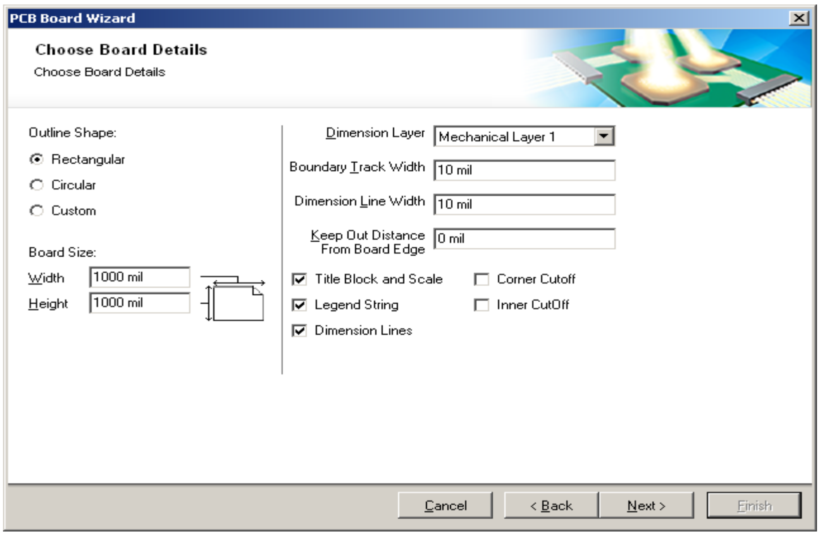




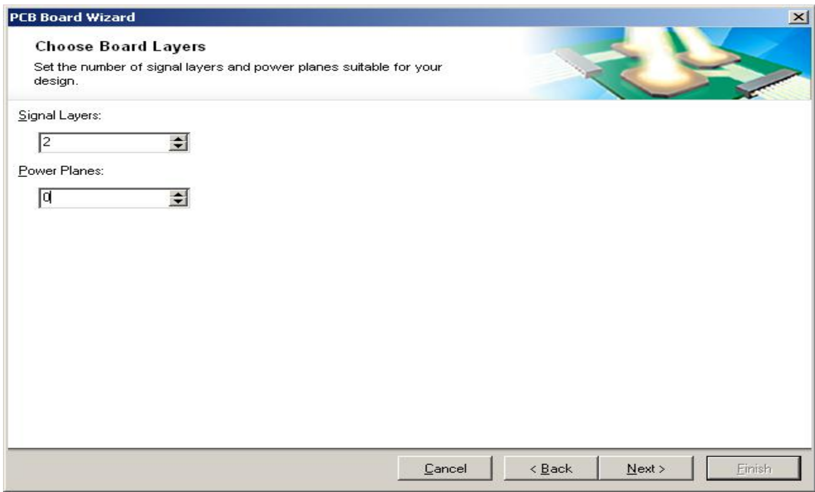
30. Choose the type of measurement units for your board (imperial = inch; metric = milimeter), for example use the imperial. Click next to the next setup



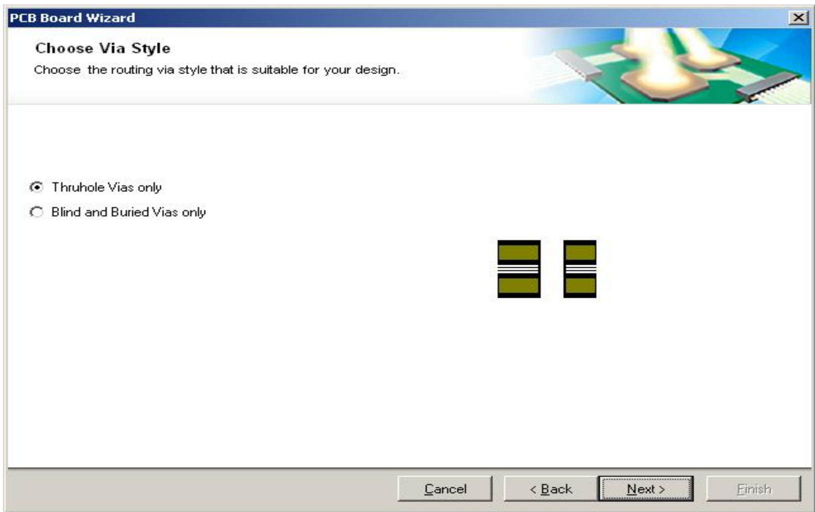
31. Choose your board shapes, Altium provides PCB board shape template. Choose custom if you want to define the board shape itself. Click next to the next setup.



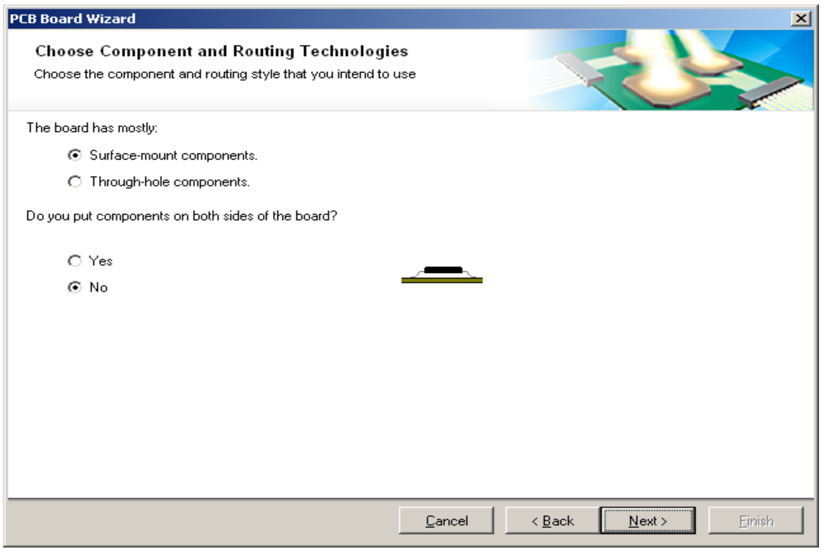
32. After you choose custom, you need to set your board shape properties. The board outline can set to rectangular, circular, and custom. Entry your board shape size and set your boundary board.



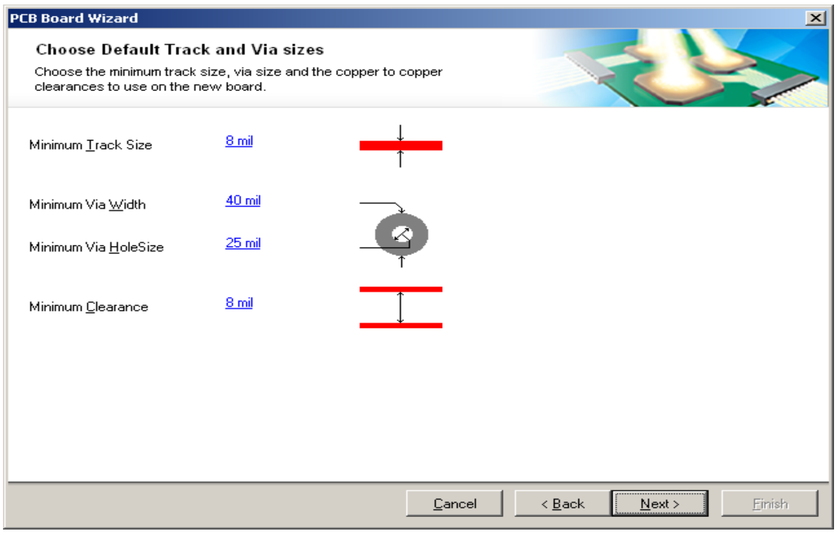
33. Choose how many layer for the board (signal layer up to 32 layer, Power Planes up to 16 layer.). if you want to set your board become 2 layers, set signal layer 2 and Power Planes 0.



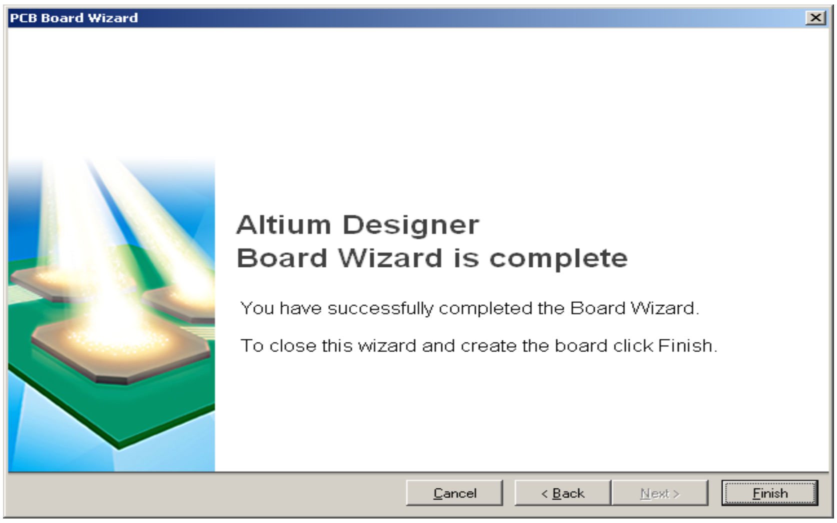
34. Choose your board VIA style, can Thruhole Via only, or Blind and Buried Vias only. Click Next to next setup.



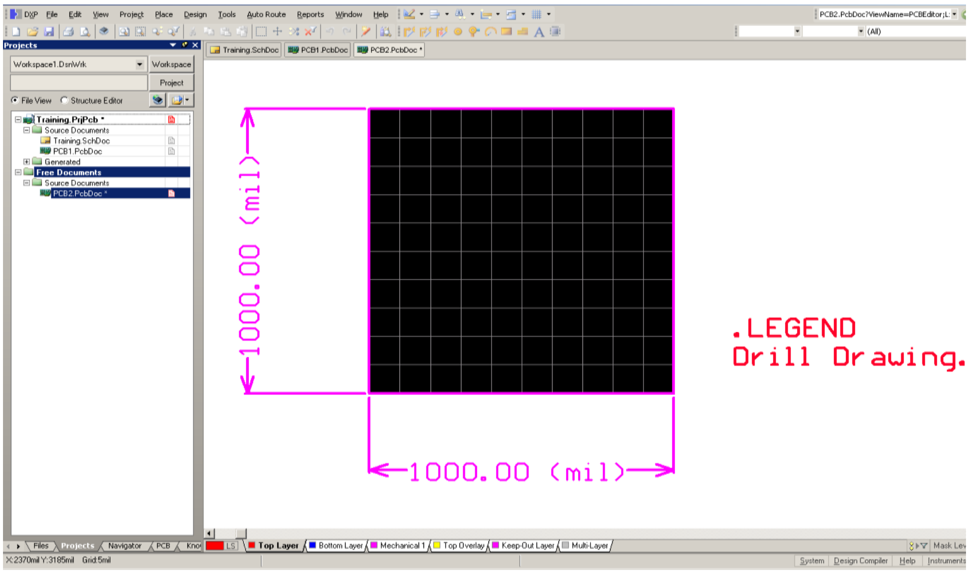
35. Set your component routing and placement. Click next to the next setup.



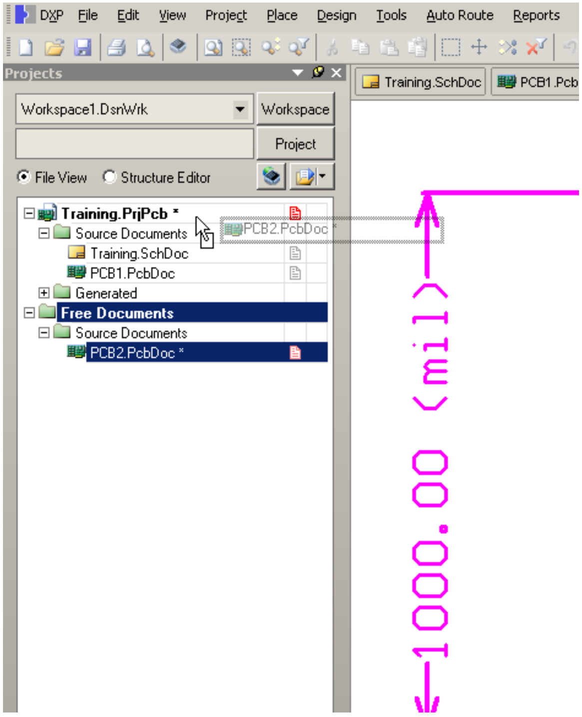
36. Set your Track size, Via properties, and clearance. This value will be your Design Rule on PCB. Click next to your next setup.

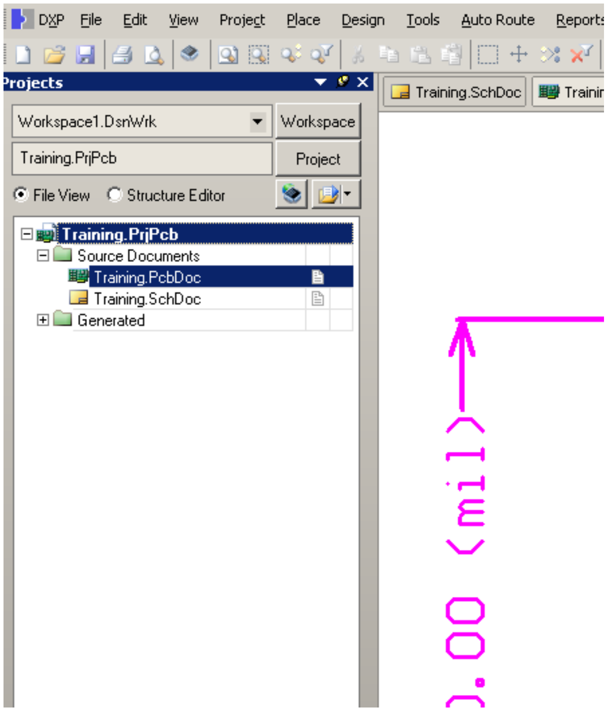


37. Click Finish to create your PCB.



38. Now you have new PCB document and this is free document, click and hold on file in Project panel, then drag to inside the project (Figure (A).)

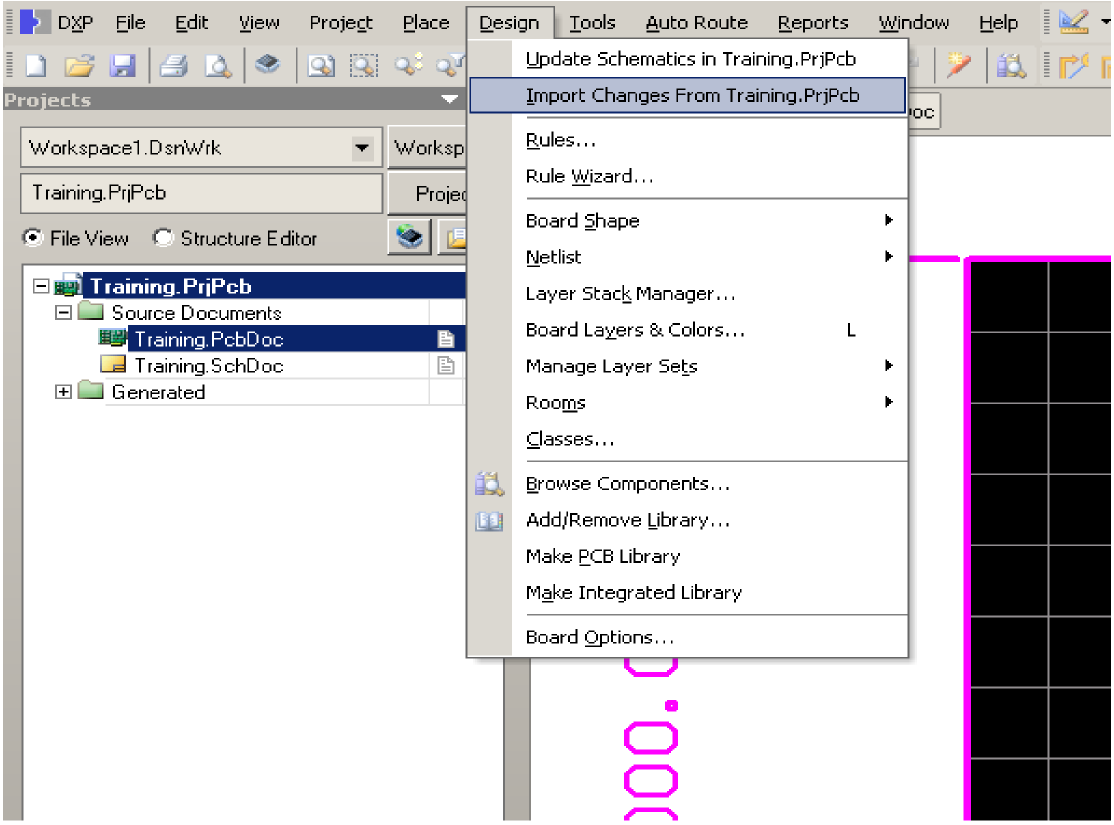




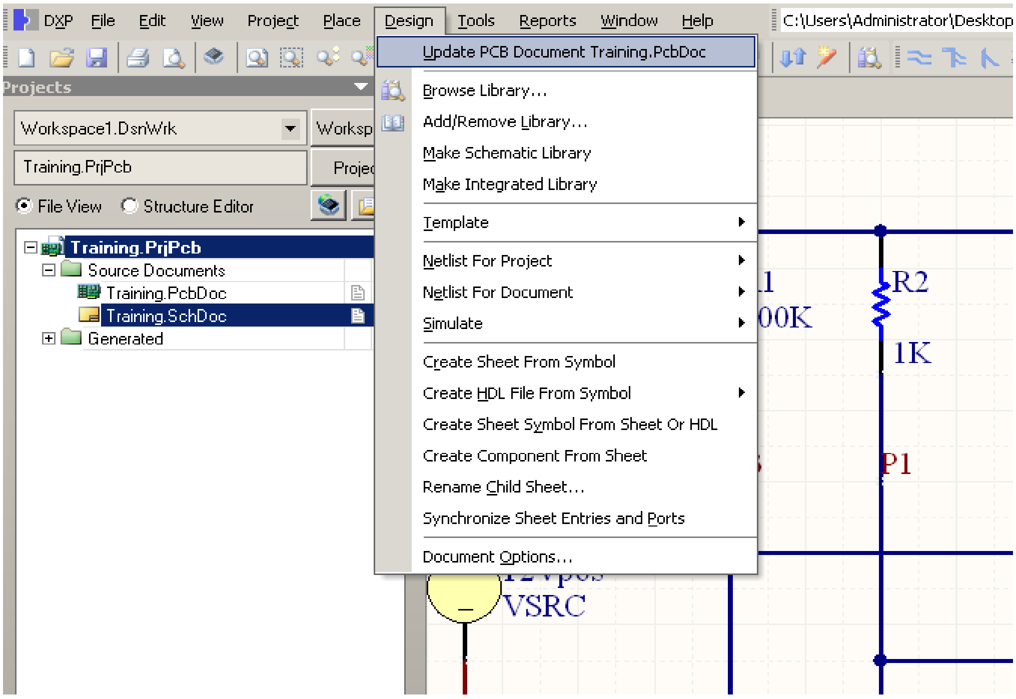
(A) (B)

39. Now the new PCB file is under your project tree, Save Your new PCB file in the same folder with your project.

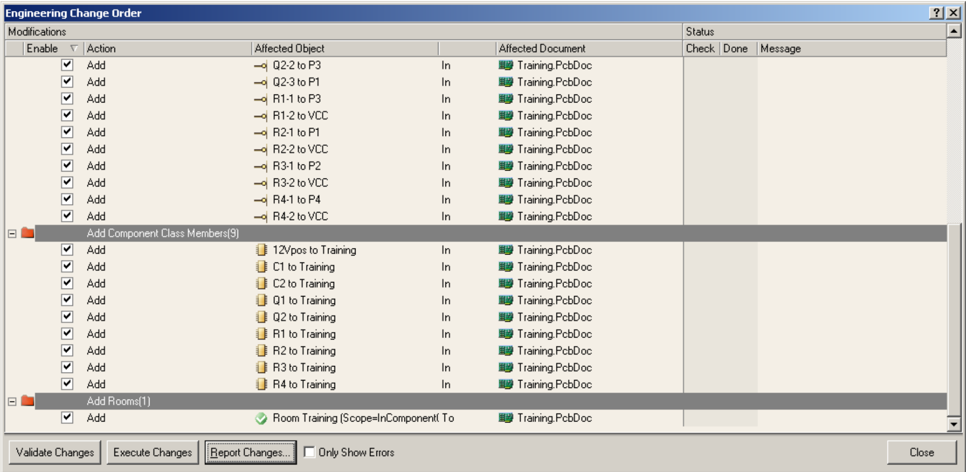
40. There are 2 ways to update design from Schematic to PCB, from PCB view. Go to **Design>>Import Changes from qqq.PrjPcb (qqq is your project name.) (D, I)**



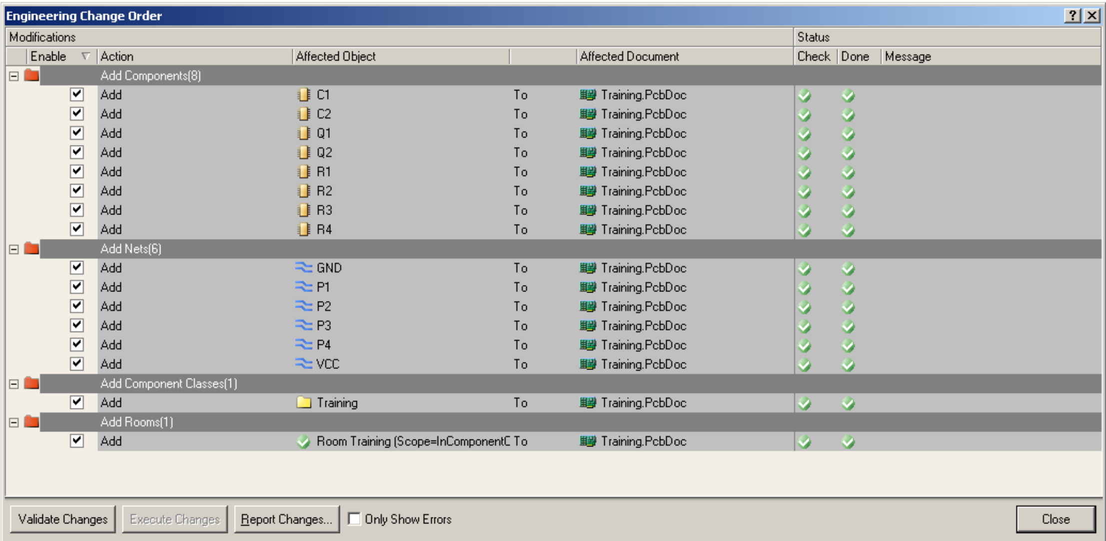
41. From Schematic view, go to **Design>>Update PCB Document xxx.PcbDoc (xxx is your pcb document name). (D, U)**



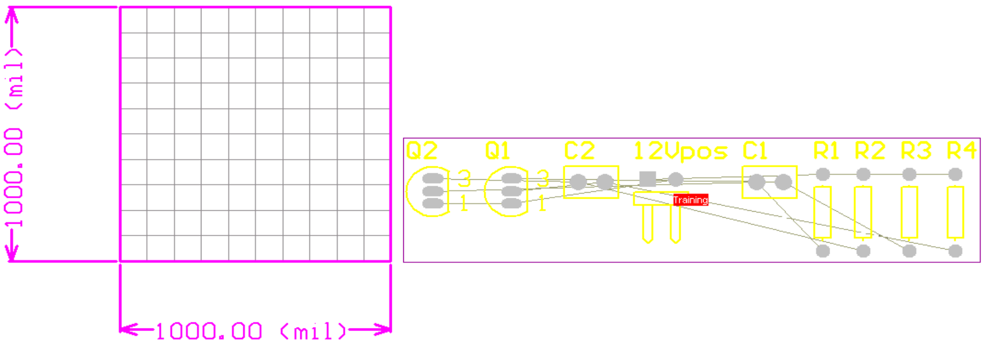
42. Engineering Change Order screen will appear..



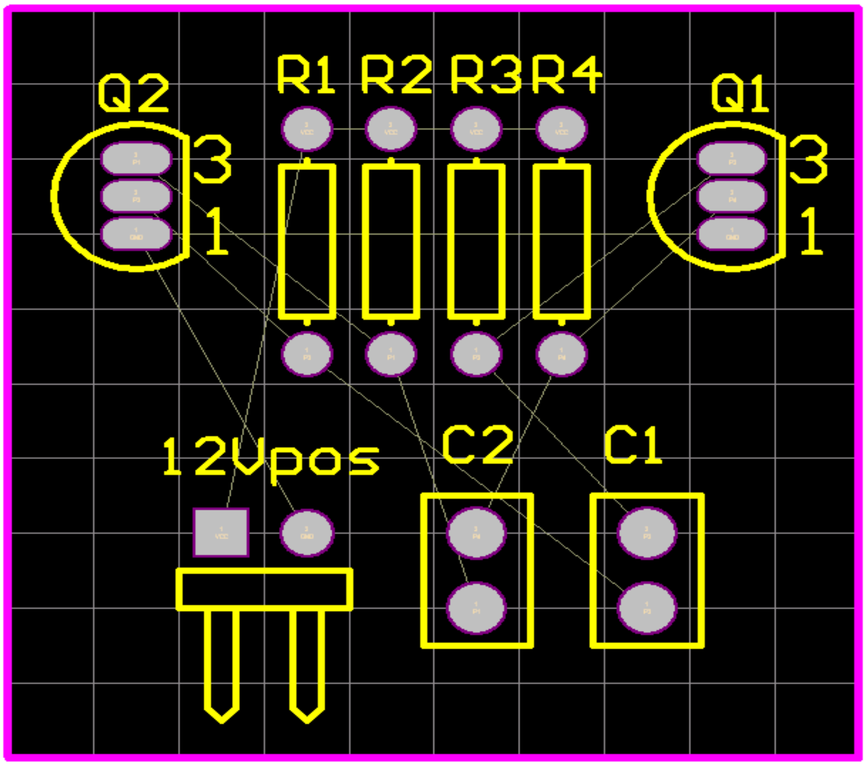
43. It gives information about how many components, Nets, component classes, and room definitions will be updated to PCB.
44. Press Validate Changes,



45. Press Execute to apply changes. Press close to close the windows.
46. Zoom out the PCB to see component, the component is placed out from PCB boundary.



47. Put the component and arrange inside the PCB board.



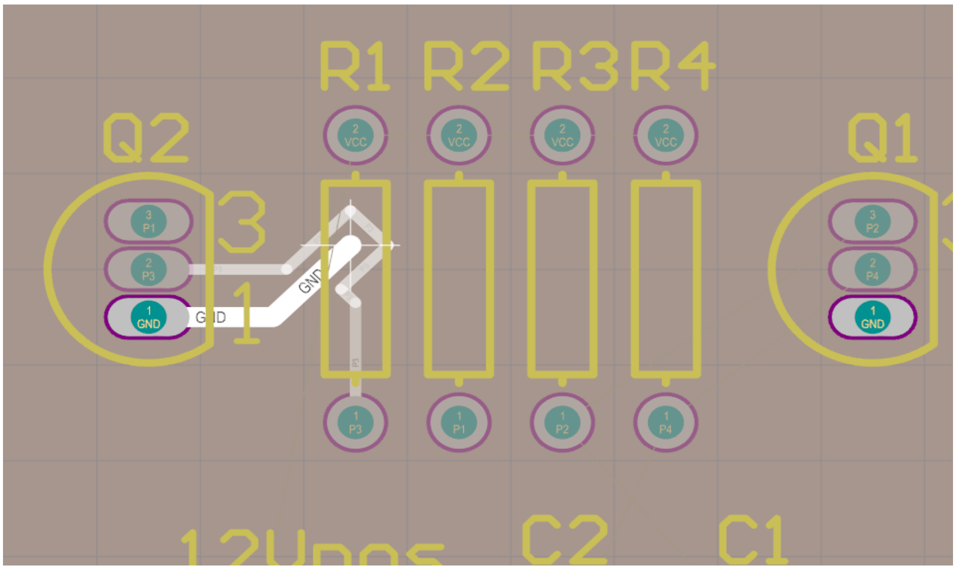
Routing

For Routing, Altium provide 4 types, 2 types for single routing (**Interactive Routing and Smart Interactive Routing**) and another 2 types are for multiple routing (**Differential Routing and Multiple Traces**).

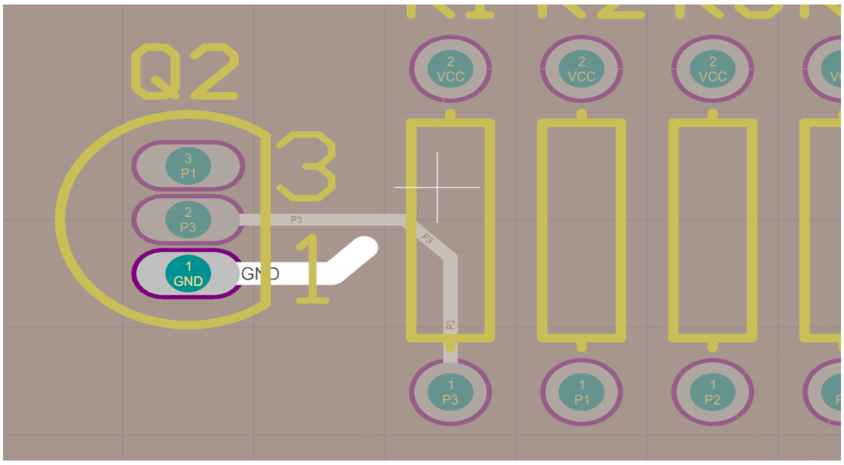
Interactive Routing

48. Interactive routing can be accessed by go to **Place>>Interactive routing (P, T)** or click on  icon.
49. In interactive routing, there're **3 modes**, **Push Obstacle**, **End Ignore Obstacle**, **Stop at First Obstacle.** This option can be accessed by pressing **SHIFT+R**

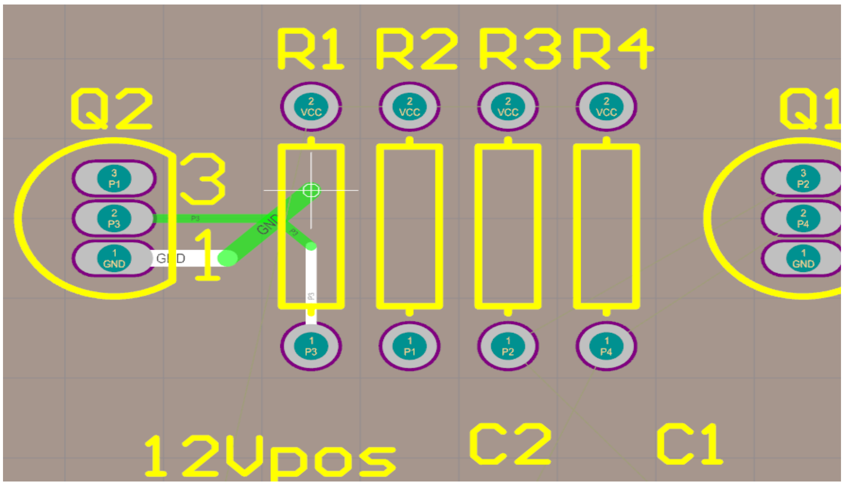
Interactive Routing (Push Obstacle)



Interactive Routing (Stop at First Obstacle)

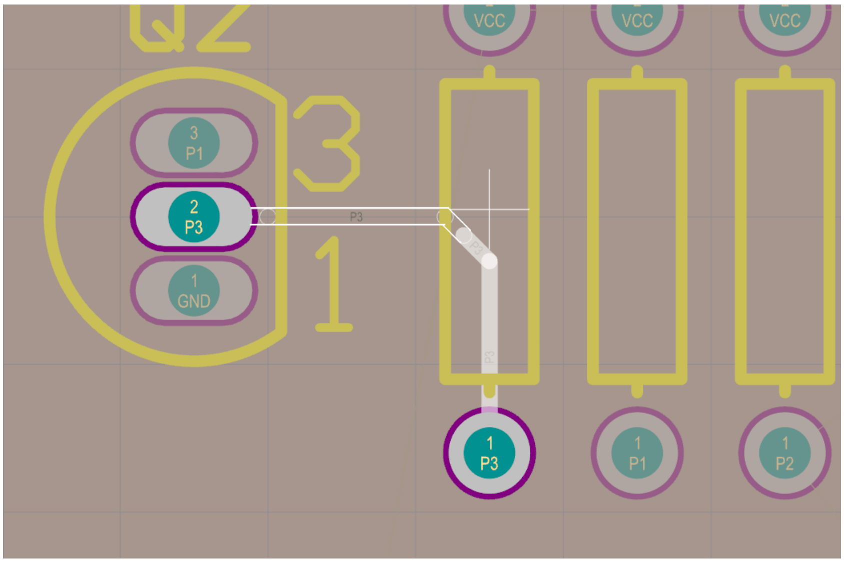


Interactive Routing (Ignore Obstacle)

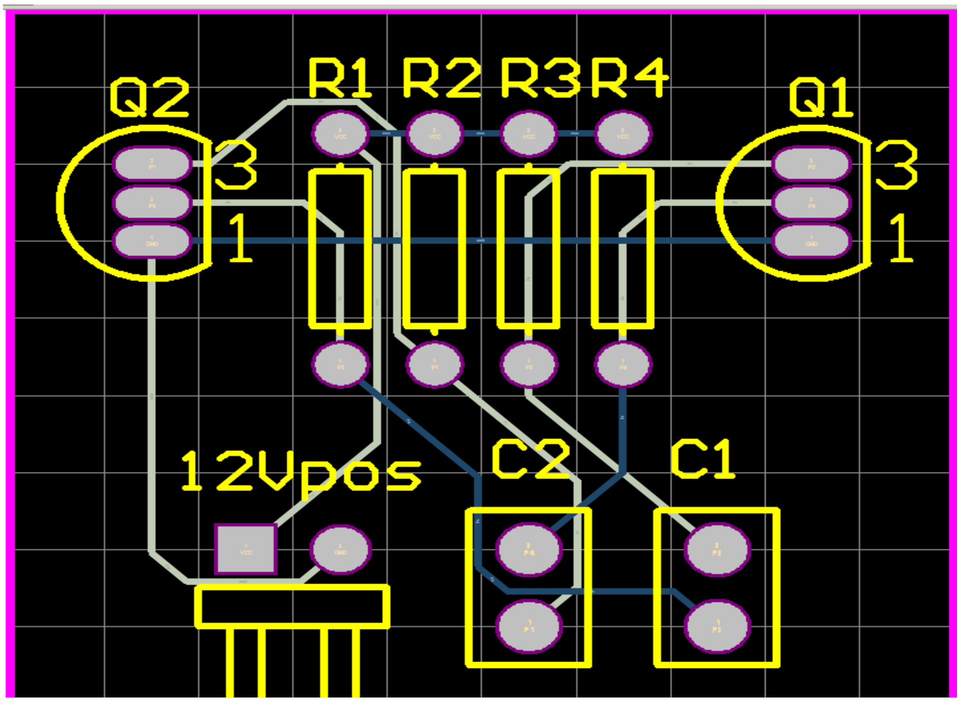


Smart Interactive Routing

50. Smart Interactive routing can be accessed by go to **Place>>Smart Interactive Routing (P, I)** or click on icon.

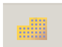


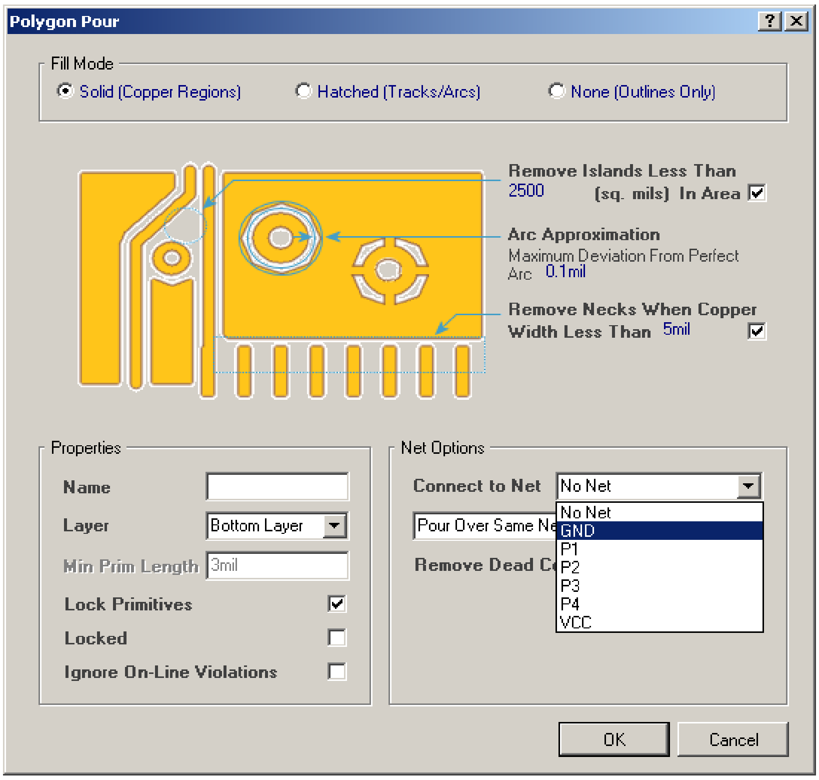
Smart Interactive routing will give preserve angle track to you, if the track is ideal, you can place the track by pressing **CTRL + left Click**. For this project, we only use Singles routing.



Polygon Pour

51. Polygon pour is a copper on PCB. This can be connected to GND, VCC or the other net.

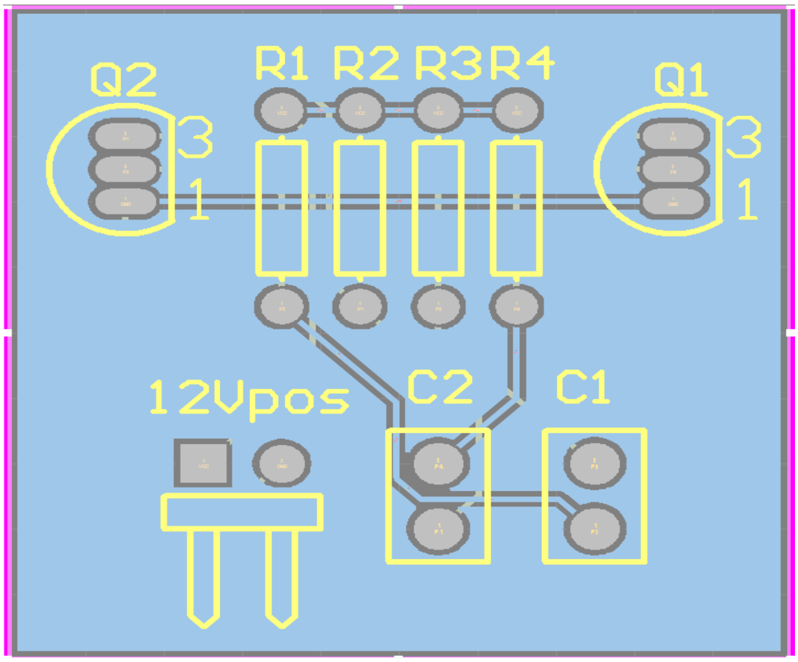
52. To place polygon pour, go to **Place>>Polygon Pour (P, G)** or click on  icon, and polygon dialog will appear.



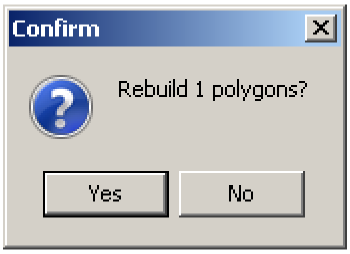
This dialog is for your Polygon properties. You can choose fill mode, properties for size of free island copper, deviation for pad boundary, polygon name, polygon active layer, net connection, etc.

After finish the setup, click **OK** to define your polygon/copper shape.

In polygon mode, for polygon shape, you can choose 45 degree, 45 degree with arc angle, 90 degree, and any degree by pressing **Shift+Spacebar**.



Now your board have copper for bottom layer. If you want to change the layer for this copper, double click on polygon pour and after dialog displays, change the layer become top layer and press OK.



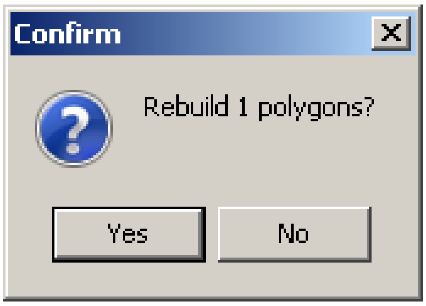
There's one confirmation window message will appears; click Yes to confirm the changes.

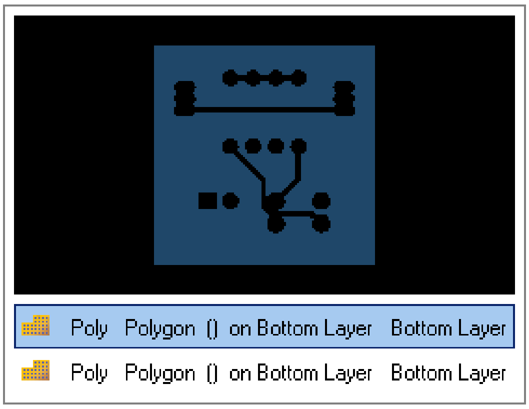
Note: to add the polygon pour to another layer, i.e. add polygon pour to top layer, you can repeat from stepi 50, or you can use copy and paste.

Use Copy and paste to add Polygon Pour on another layer.

53. Select current polygon pour, copy the polygon by pressing **CTRL+C**, define **one point** around the polygon by click on polygon and that point become your reference point when placing the polygon. Now press **CTRL+V** to paste the polygon, and the polygon will have reference point same like as the point that had been choose.
54. The confirmation dialog will appear to confirm to rebuild the polygon.

55. Click **No**, to cancel rebuild the polygon.
56. Then double click on polygon pour, it will show to you 2 polygons on your PCB.
57. Choose any polygon, then Polygon dialog will appear. Change the layer connection become **top layer**, then click OK. Confirmation dialog will appear again, click YES to apply changes.
58. Now you have two polygons on your PCB and in different layer.
59. If you want to edit your track, you don't need to delete your polygon, you can remove it temporary. To remove your polygon temporary, go to **Tools>>Polygon Pours>>Shelve polygon**. And after edit the track, you can restore back your polygon by go to **Tools>>Polygons Pour>>Restore polygons.**

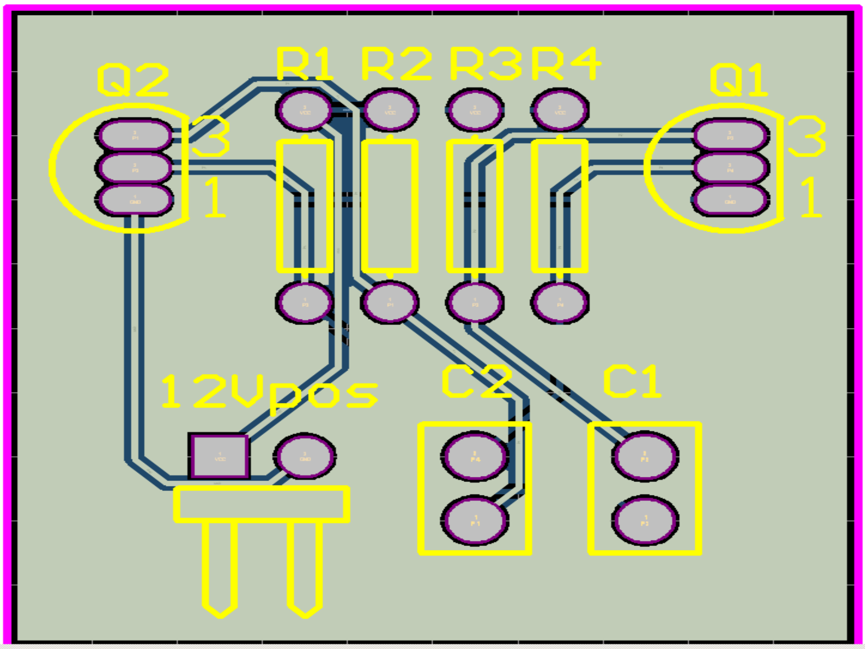




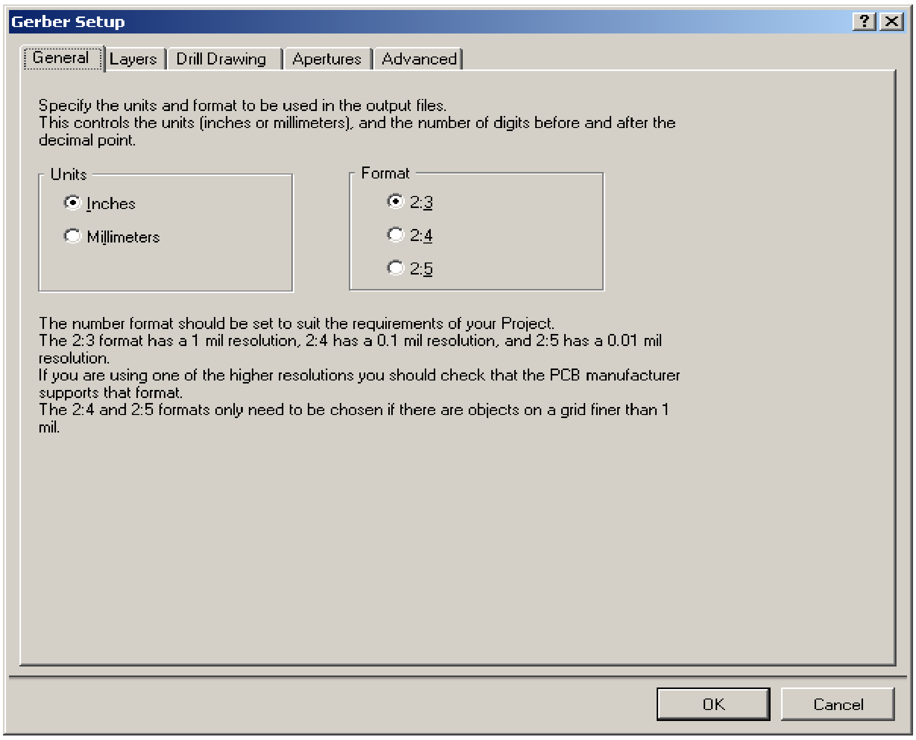
Now you board is ready.

Creating Gerber File and NC Drill file

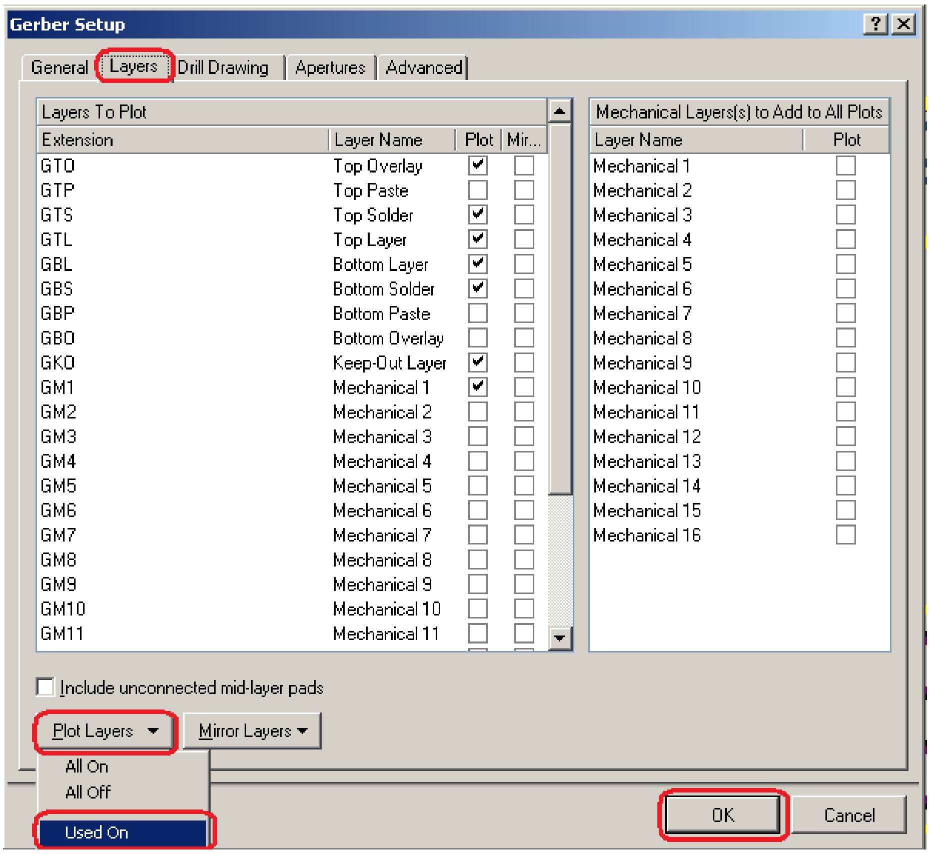
After PCB had been done, next step is creates Gerber and NC Drill File.



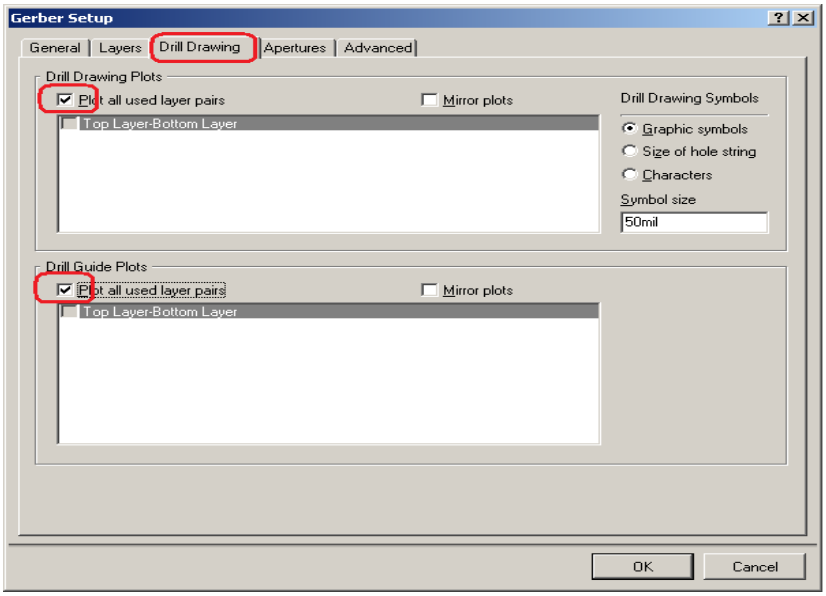
60. On PCB panels, go to **File>>Fabrication outputs>>Gerber Files** to create the Gerber file and Gerber Setup dialog will appear.



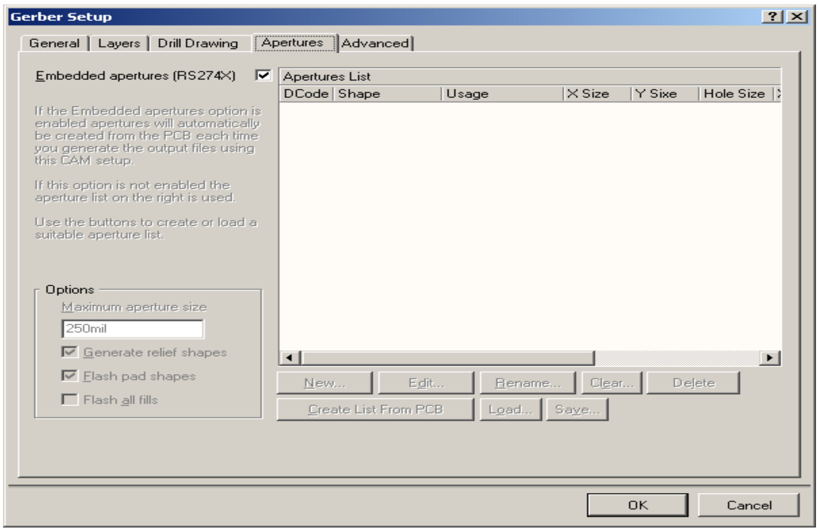
61. Choose your unit measurement and Format, or you can take it from default value. Click to the Layers TAB.



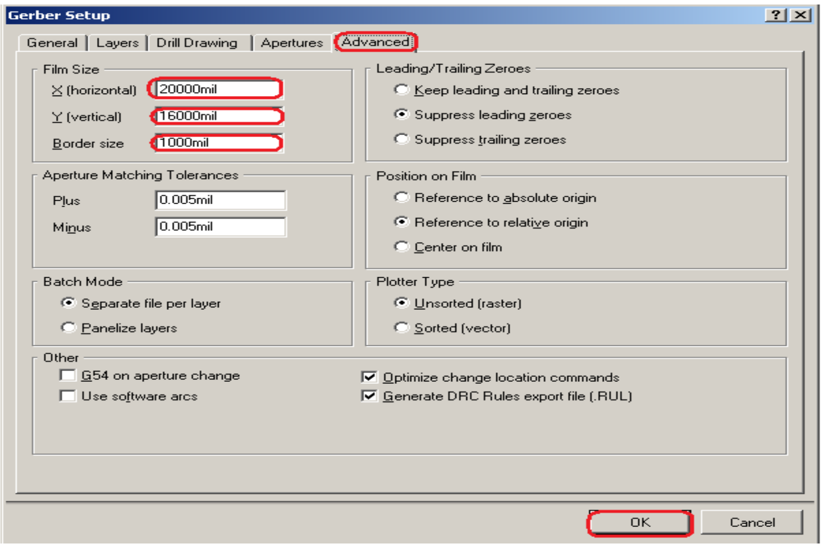
62. On this TAB, choose which layer that you want to create the gerber. If you don't know which layer that you are using now, click on Plot Layers and choose Used on. Then go to next TAB, Drill Drawing TAB.



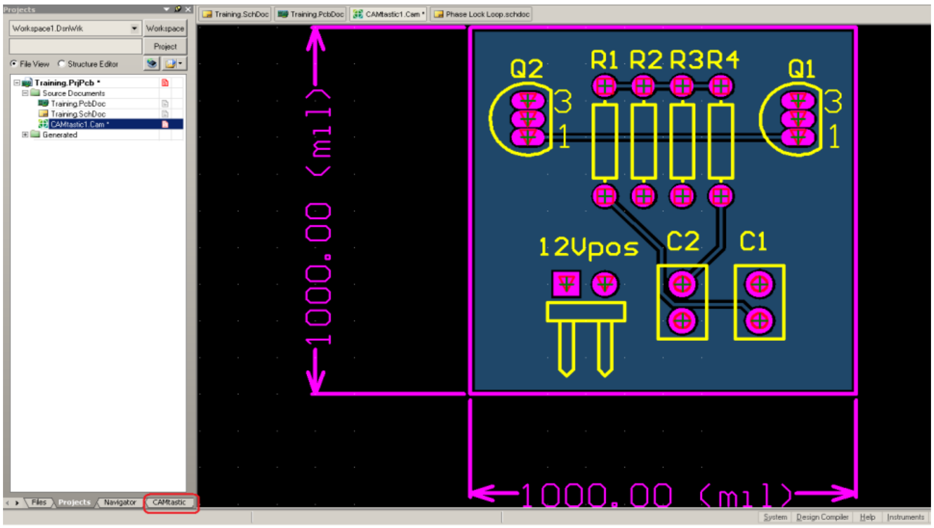
63. Enable Plot all used layer and then go to Aperture TAB.



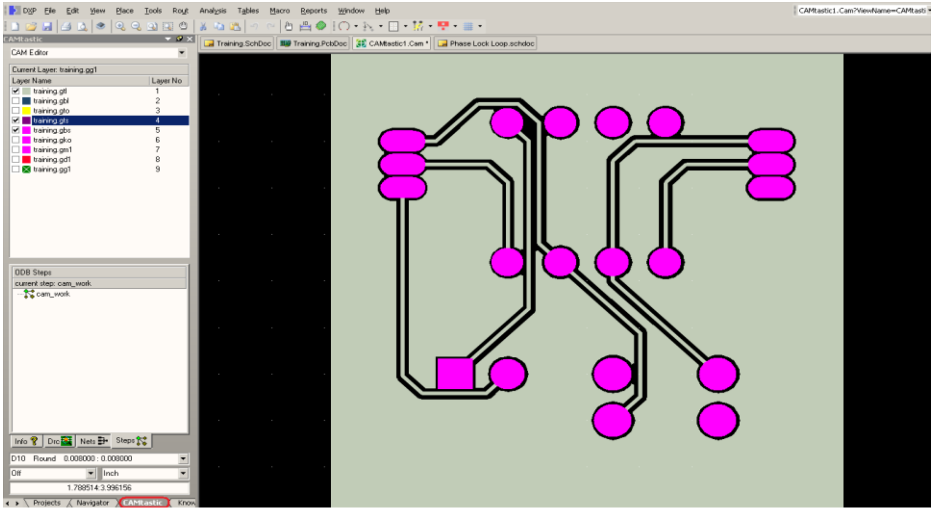
64. On this TAB, we no need to set anything, go to next TAB.



65. The value from film size, can take it from default, but if the board size bigger than the file size, you have to change the film size value larger than film size. Click OK to generate the file.



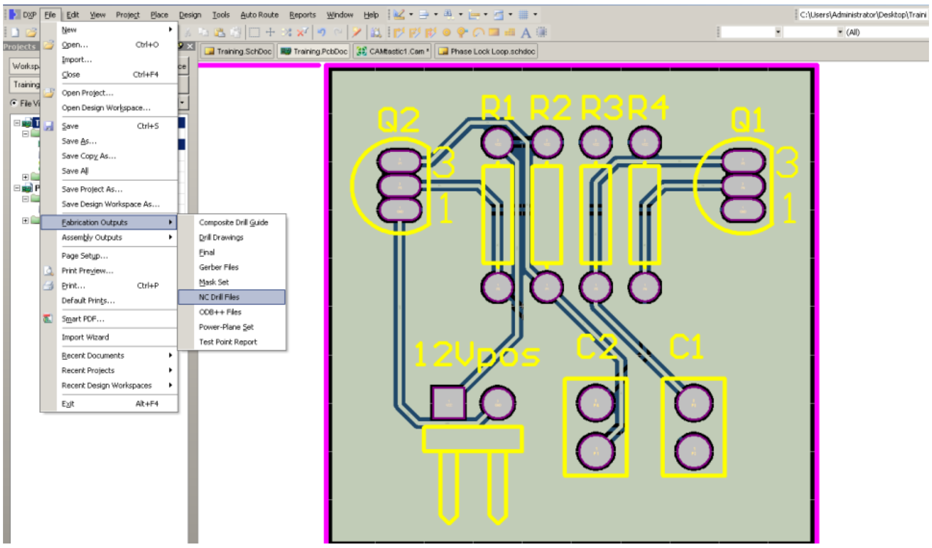
66. Save your Camcastic File on your project folder and go to **Camcastic editor** to see your gerber file layer by layer.



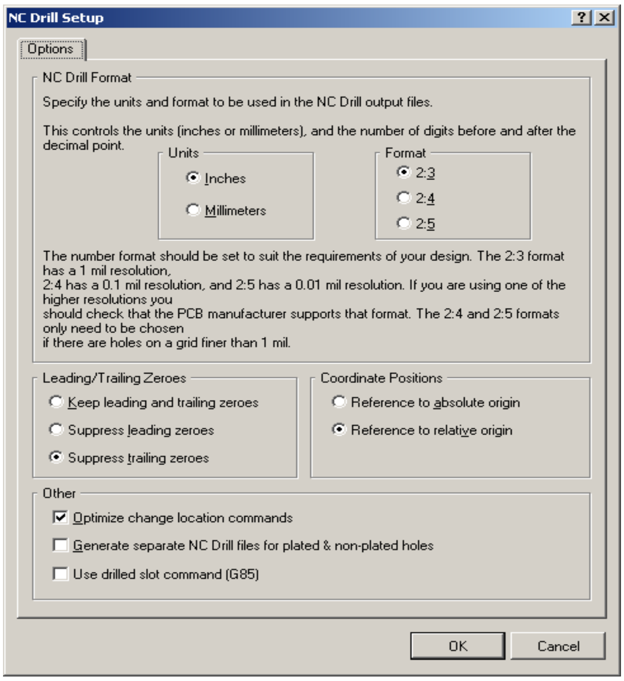
65. Uncheck all the layers box, and enable one by one to check your board.

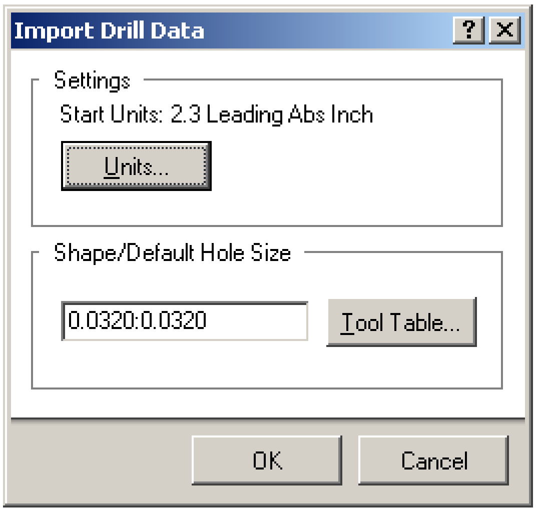
Create NC Drill File.

NC Drill file is a file with information about engineering Drill. We can create this file by go to **<u>F</u>ile>><u>F</u>abrication Outputs>>NC Drill Files.**



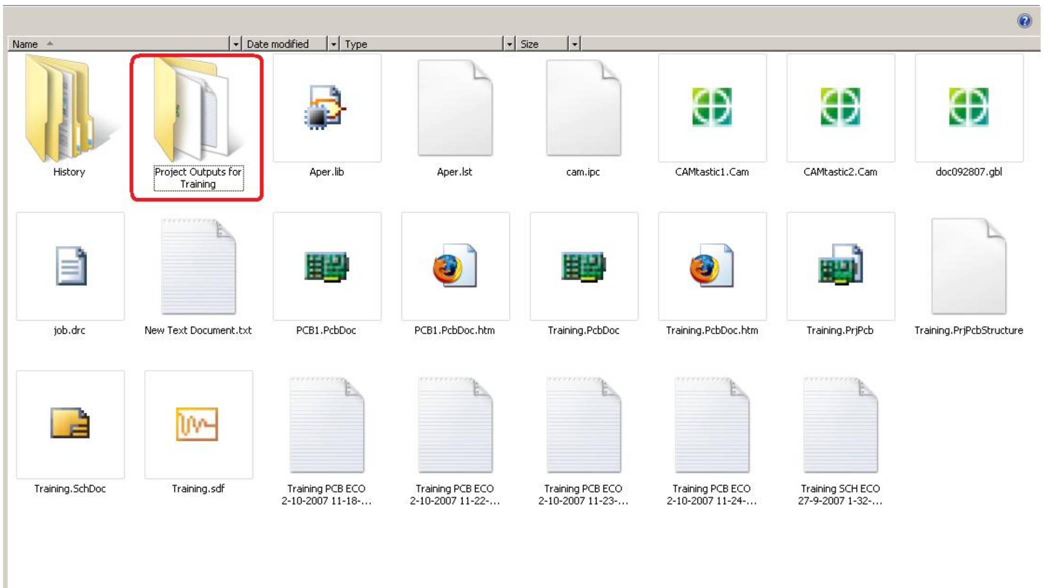
67. NC Drill Dialog will appear and make sure this setup information is same like as your gerber file information or you can take it from default setting. Press **OK** to continue.





68. After press **OK**, another window will appears, this is Import Drill Data dialog. Press **OK** to Continue.

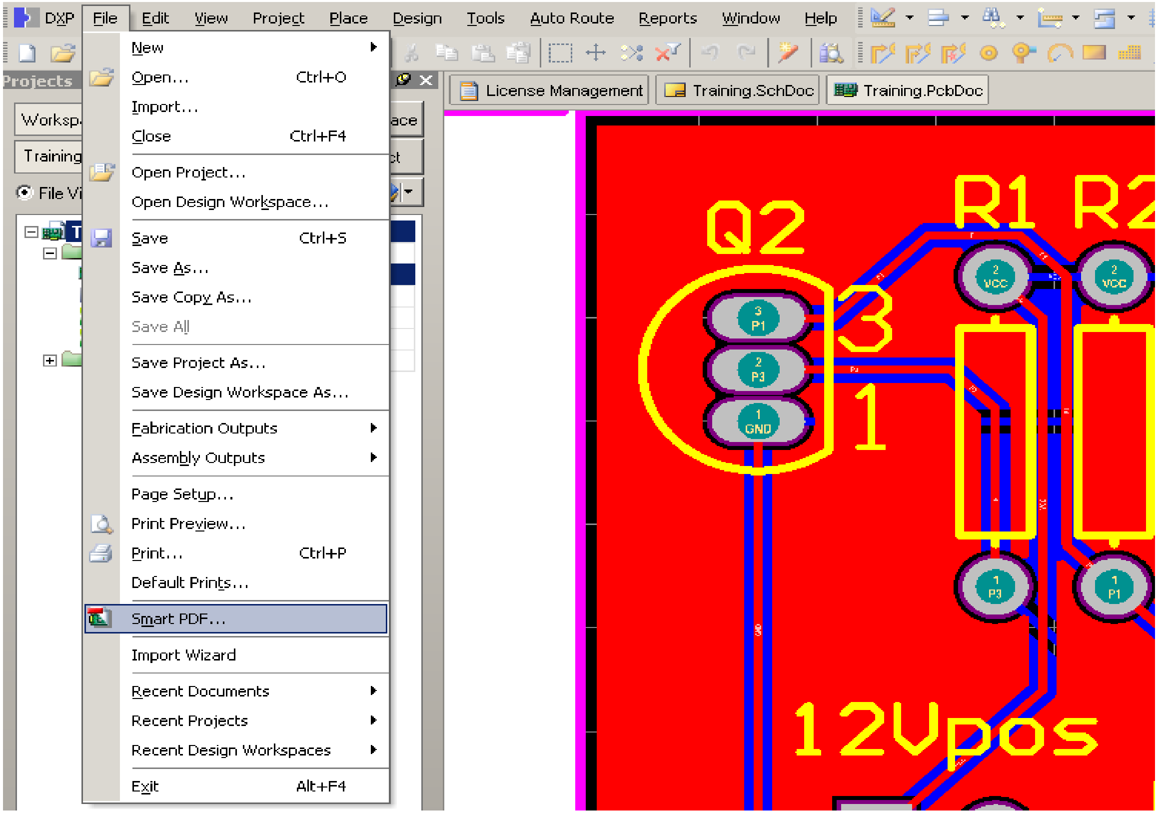
69. A new Camcastic file for NC Drill file will be created, save this file on your project folder.



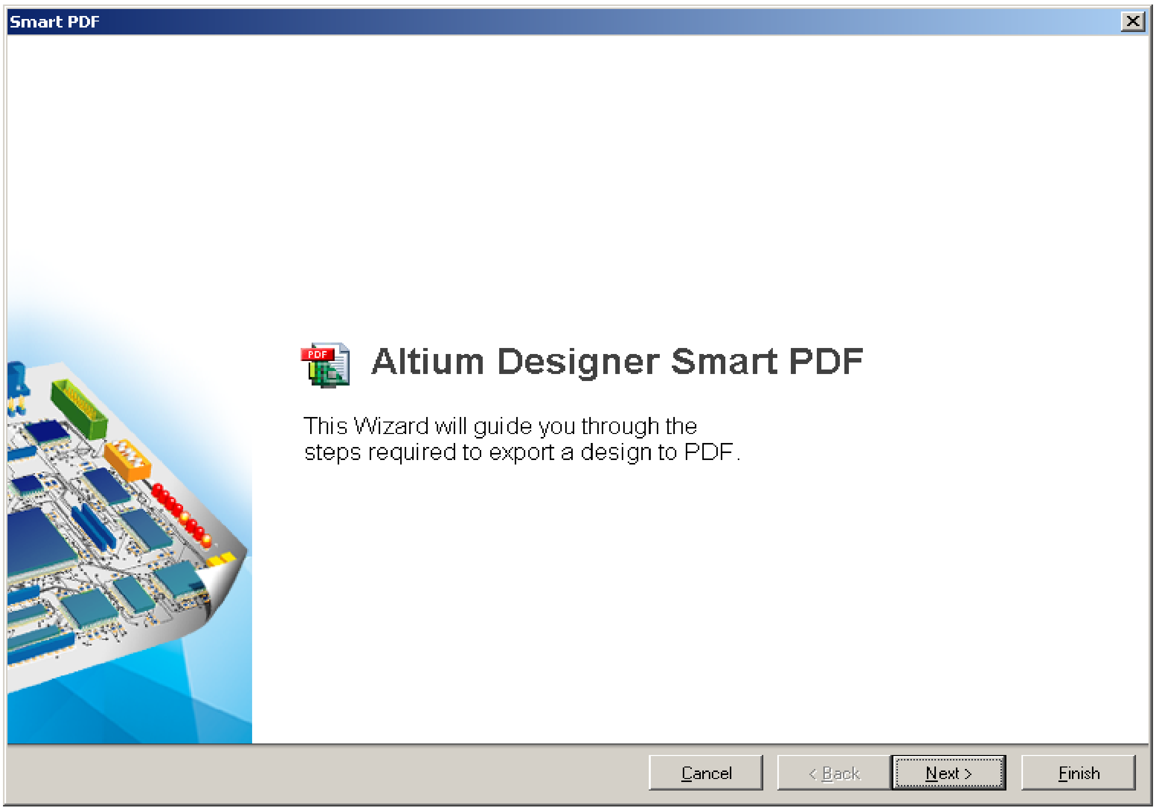
70. Your Gerber and NC Drill Files will be generated and placed in your project output folder.

Documentation

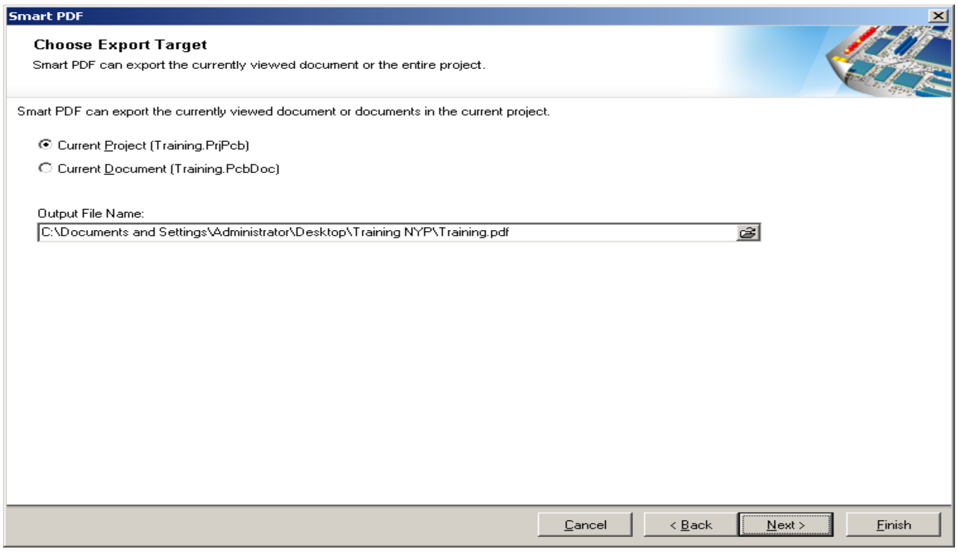
Documentation for project is very important, Altium give a better tool to do documentation for your project. It’s called Smart PDF. This tool is called Smart PDF due to you can browse your component one by one by clicking on component index and this tool also can generate your schematic and PCB file into PDF file without any PDF writer on your system. So you no need to install any PDF writer, what you need is only PDF reader.



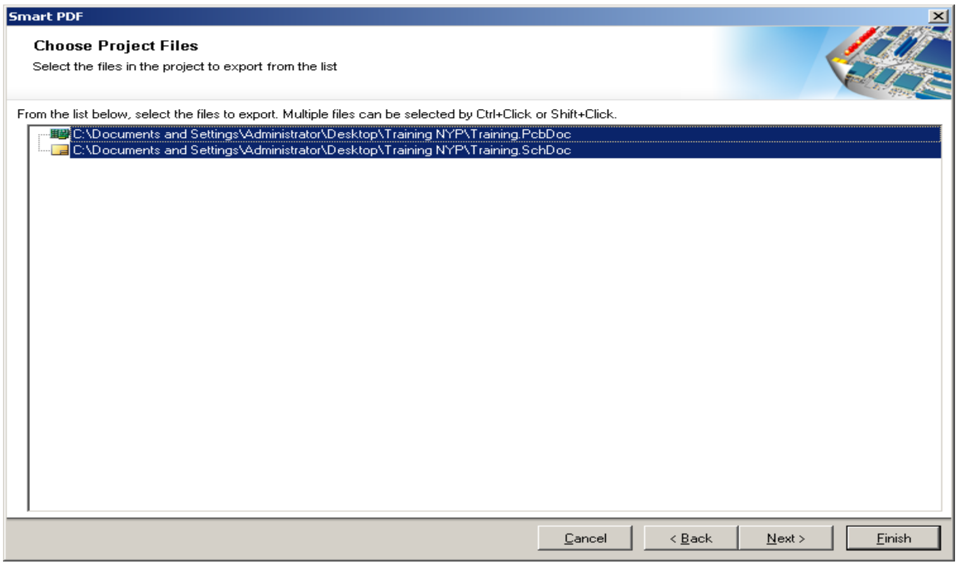
71. To use this feature, go to **File>>Smart PDF (F, M)** and smart pdf setup dialog will display.



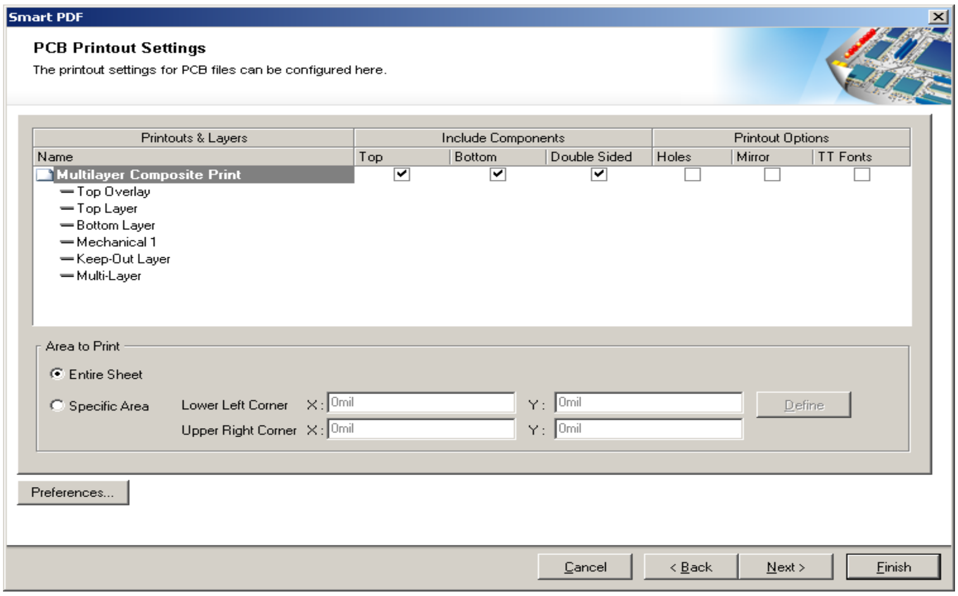
72. Smart pdf setup dialog will display. Click next to go to the next setup.



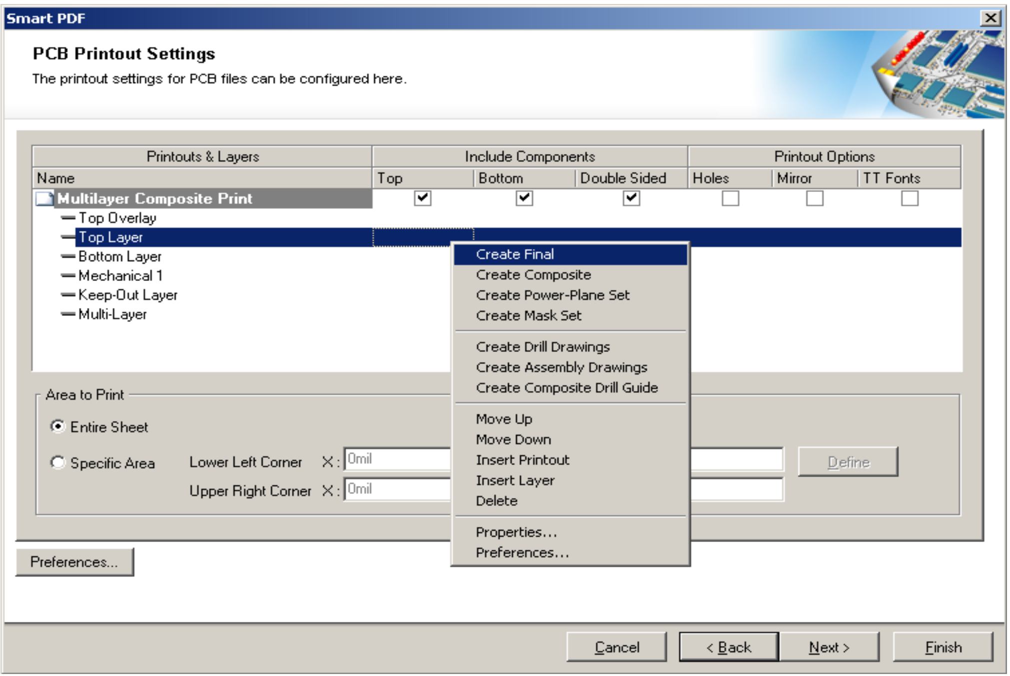
73. Choose your document and select the directory to place your pdf file. Click next.



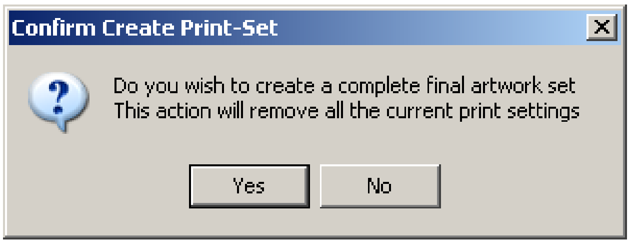
74. Choose your project file then click next.

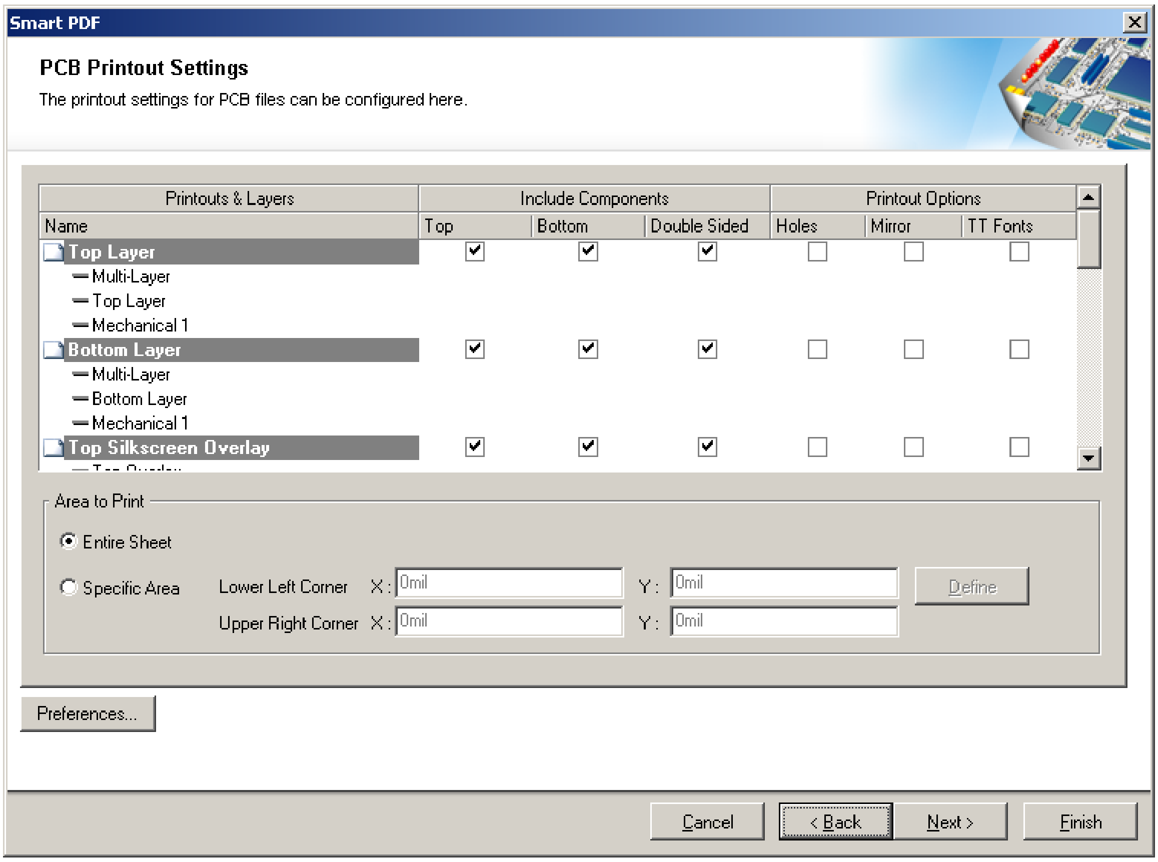


75. This dialog will configure your printout setting. Right click on windows, and click final to create all your entire Schematic and PCB.

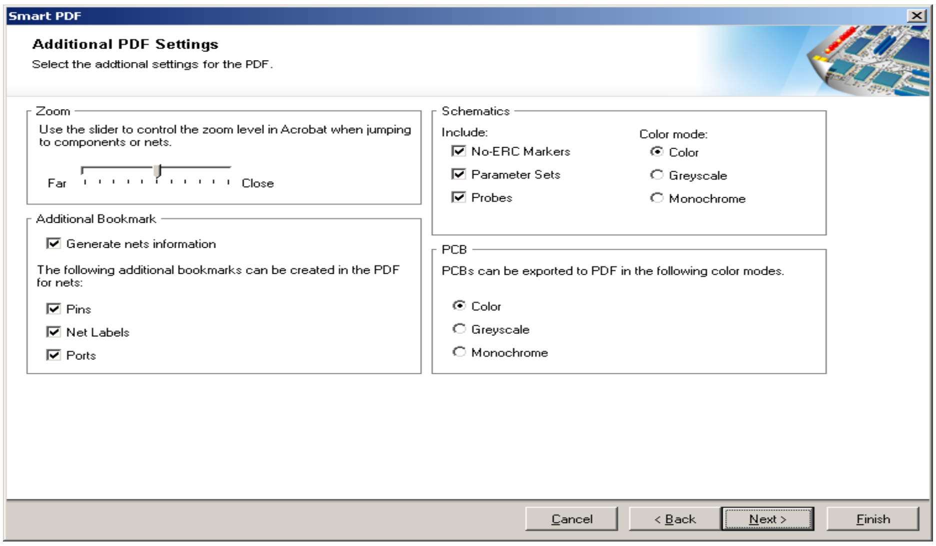


Confirmation window will appear and click **Yes** to continue.

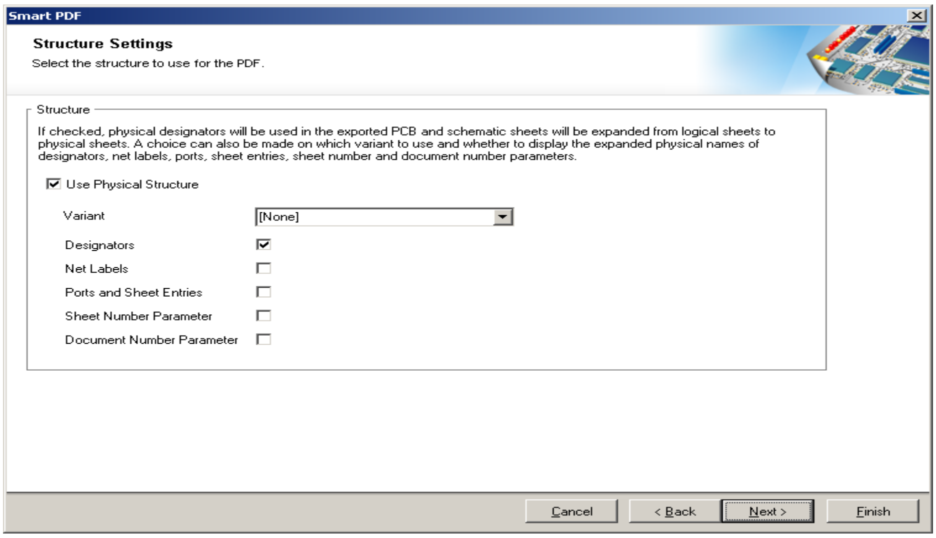




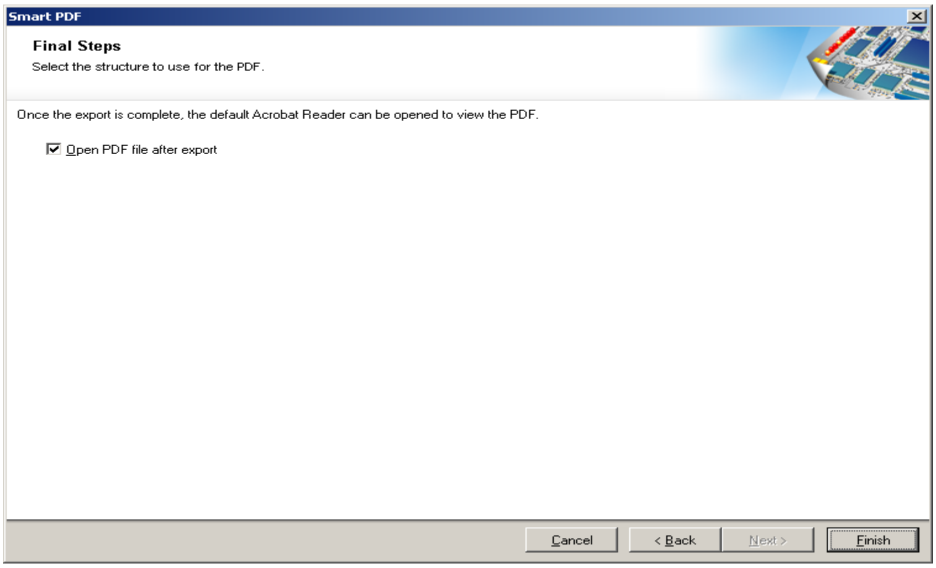
76. Press **Next** to continue the dialog.



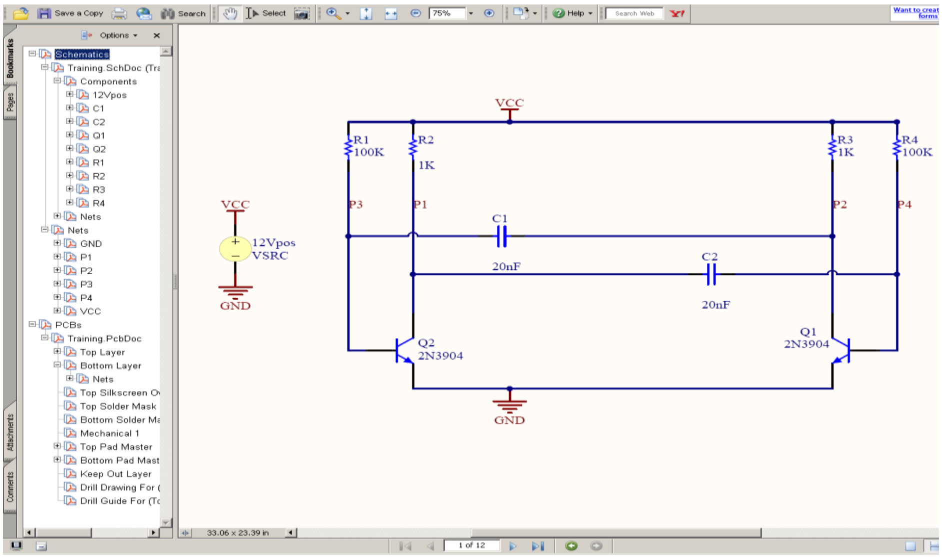
77. You can set your additional setting for schematic and PCB in this dialog or let it default. Click **next** to continue.



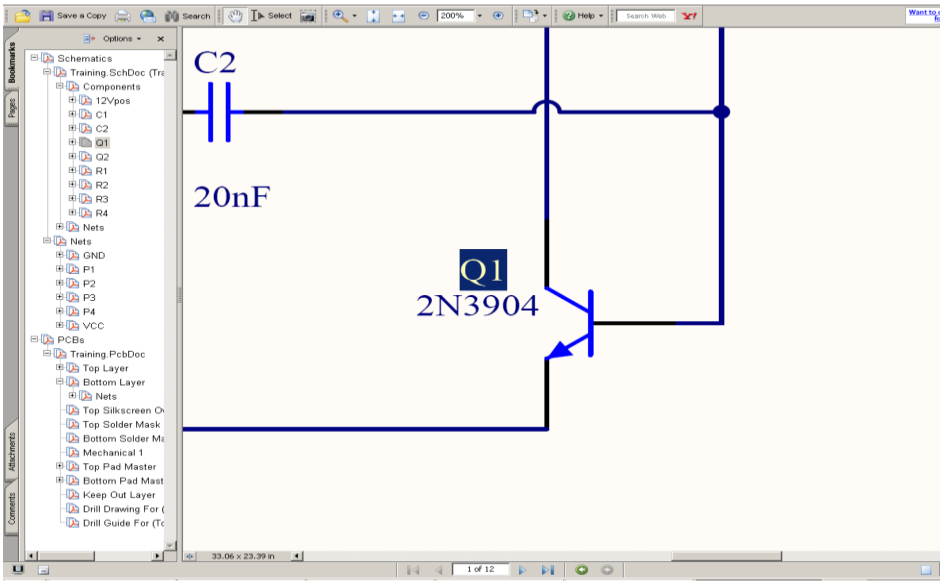
78. You can set this option or can take it from default setup, click next to continue.



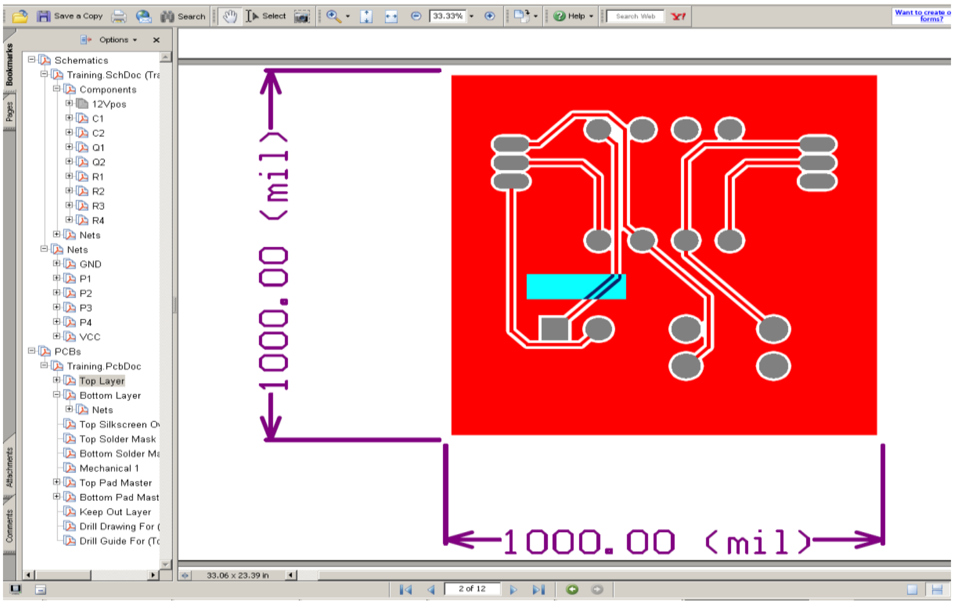
79. This is the Final dialog from Smart PDF setup, click **FINISH** to view the PDF file.



80. Browse your component one by one by clicking on the component name in index properties.



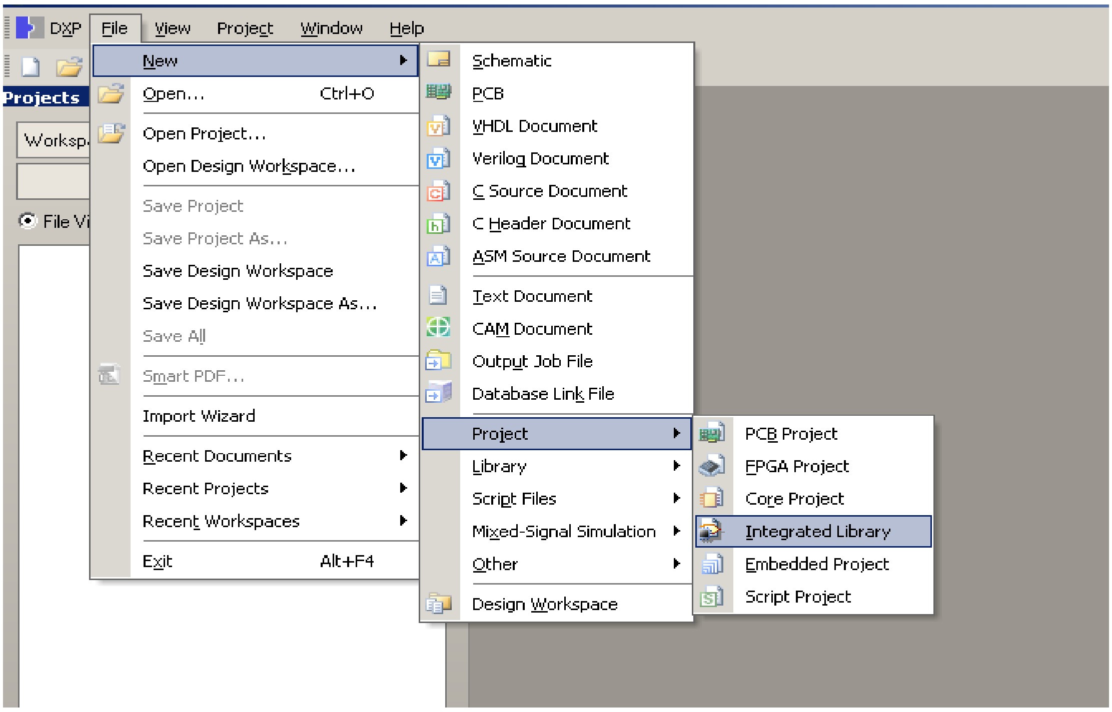
81. Browse your component one by one



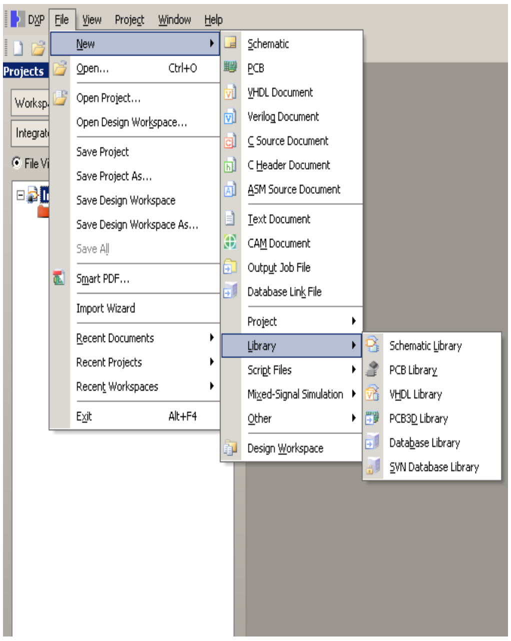
Creating Integrated Library, Footprint, Schematic Library, and link it together.

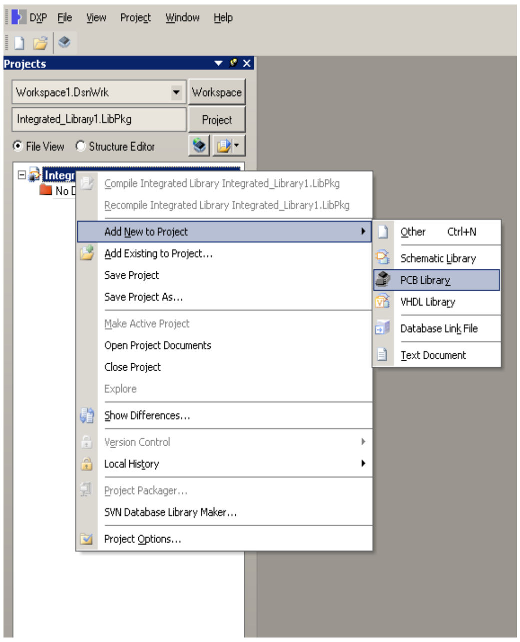
Footprint

81. To create your own footprint, you must in PCB Library Editor. Go to **File>>New>>Project>>Integrated Library**.



82. Add PCB Library to your project. Go to **File>>New>>Library>>PCB Library** or **right click on project >> Add New to Project>>PCB Library**.



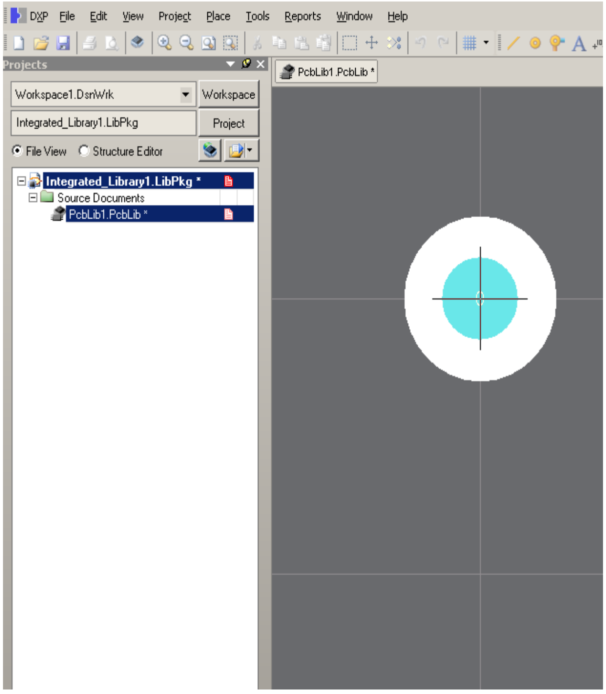


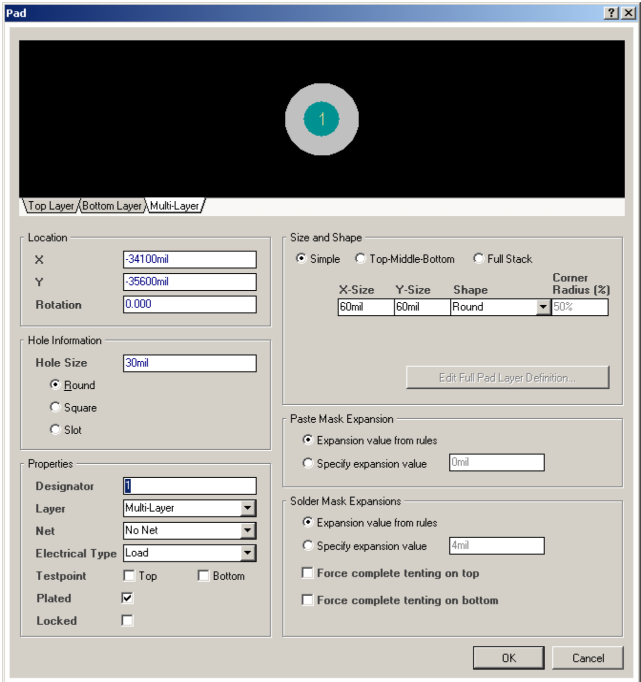
Note: Don't forget to save your integrated library, schematic library and PCB library.

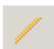
For this example we will build Capacitor, Resistor and LM 75 Temperature Sensor.

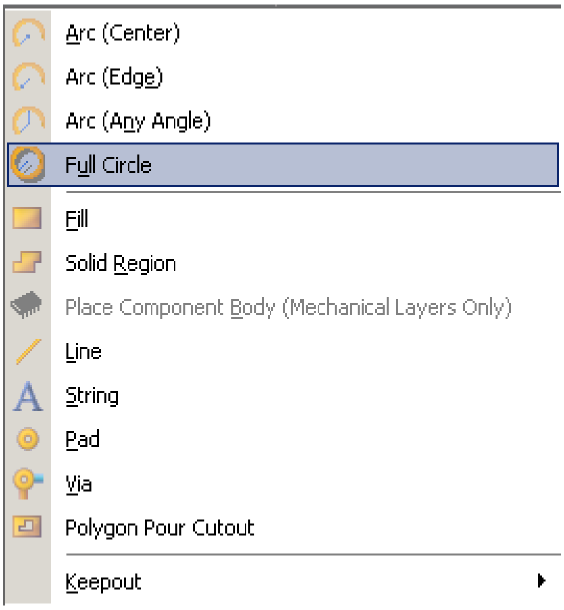
Capacitor

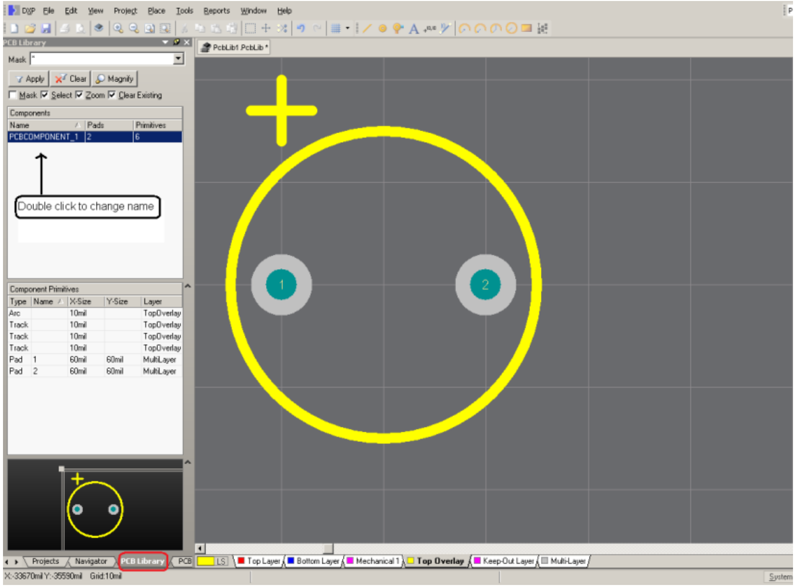
83. Zoom In the windows (**CTRL+Scroll Up** or **Page Up**) until you can see the Visible Grid. To place the pad, go **to <u>P</u>lace>><u>P</u>ad (P. P)** or click on this icon . Before the placement, press **TAB** to edit the properties, change the designator value to **1** and after the first placement, Pad number will automatically increase. On this dialog, you can set up your pad hole properties, example, you can change hole size, hole type, shape, layer connection, etc.





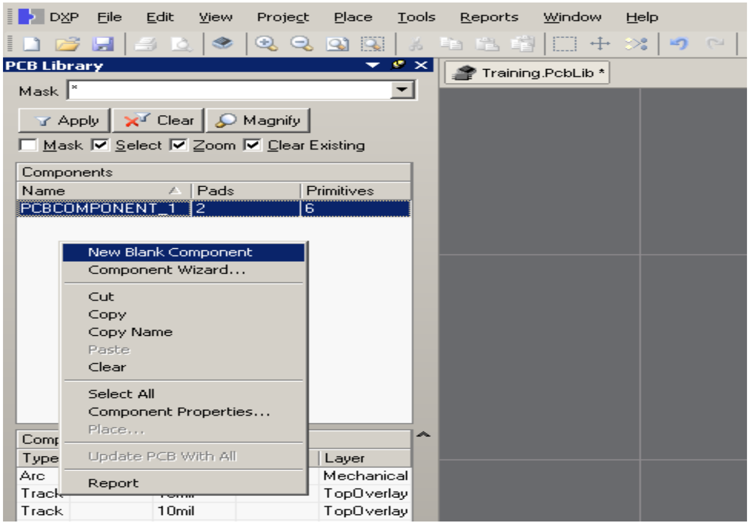
84. Use line to draw component silkscreen, first step is change your current layer to Top Overlay, go to **<u>P</u>lace>><u>L</u>ine (P, L)** or click on this icon . To Draw ARC, Go to **<u>P</u>lace>><u>A</u>RC (P, A)**, there are 3 types ARC, choose which one you feel suitable. Click on Pad number 1, and set it become origin for this component by go to **<u>E</u>dit>>Set Re<u>f</u>erence>><u>P</u>in 1 (E, F, P)**



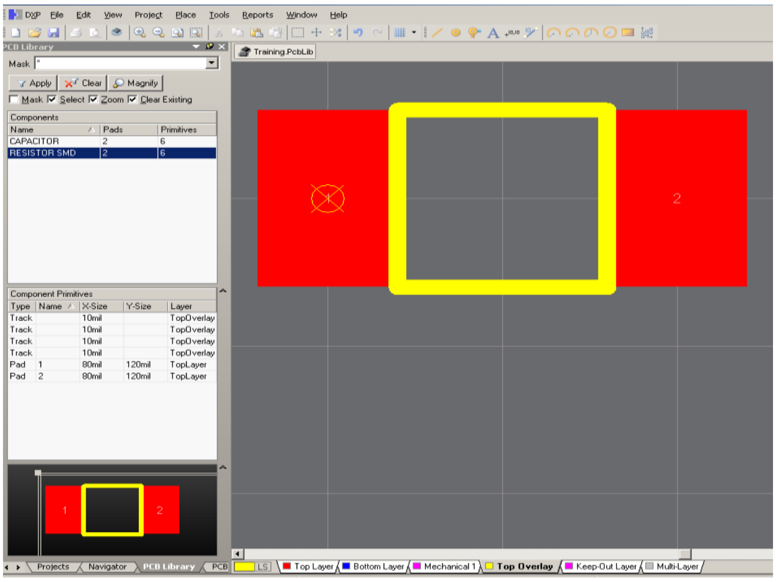


85. **Double click** on component footprint to edit the component properties (name and description) or go to **Tools>>Component Properties (T, E)**.

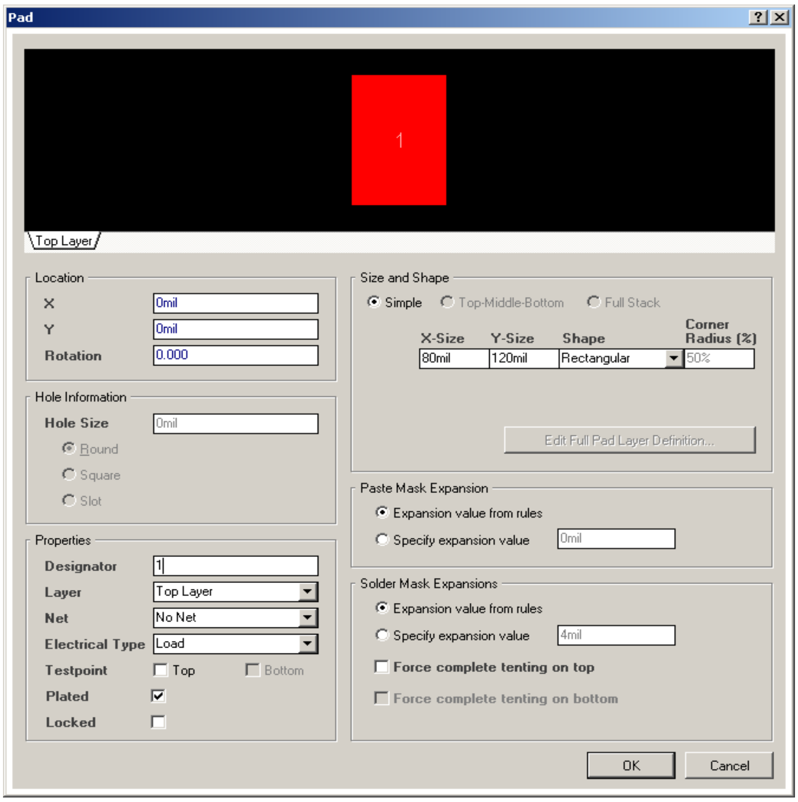
86. To create another footprint, **Right click on PCB Library Editor**, and choose **New Blank Component** or go to **Tools>>New Blank Component (T, W)**.



Resistor



87. This is SMD resistor, to draw this footprint, use the pad and edit the pad properties. Change the properties same as shown below.

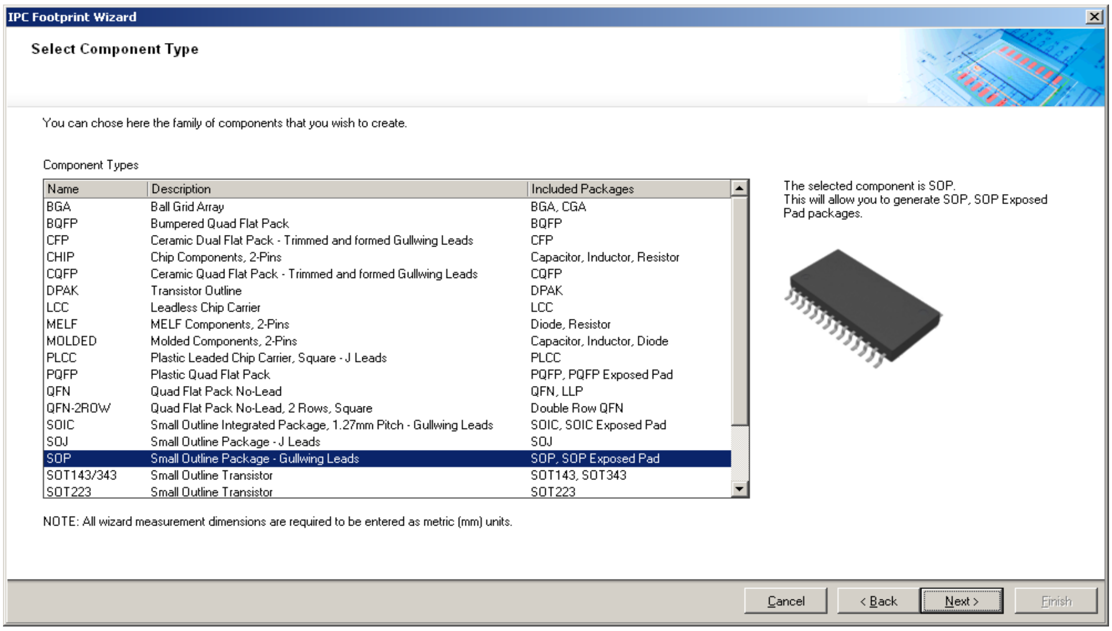


88. To draw **silkscreen**, use line and make sure you are on **Top Overlay Layer**.

Build Footprint based on Component Datasheet using IPC Footprint Wizard.

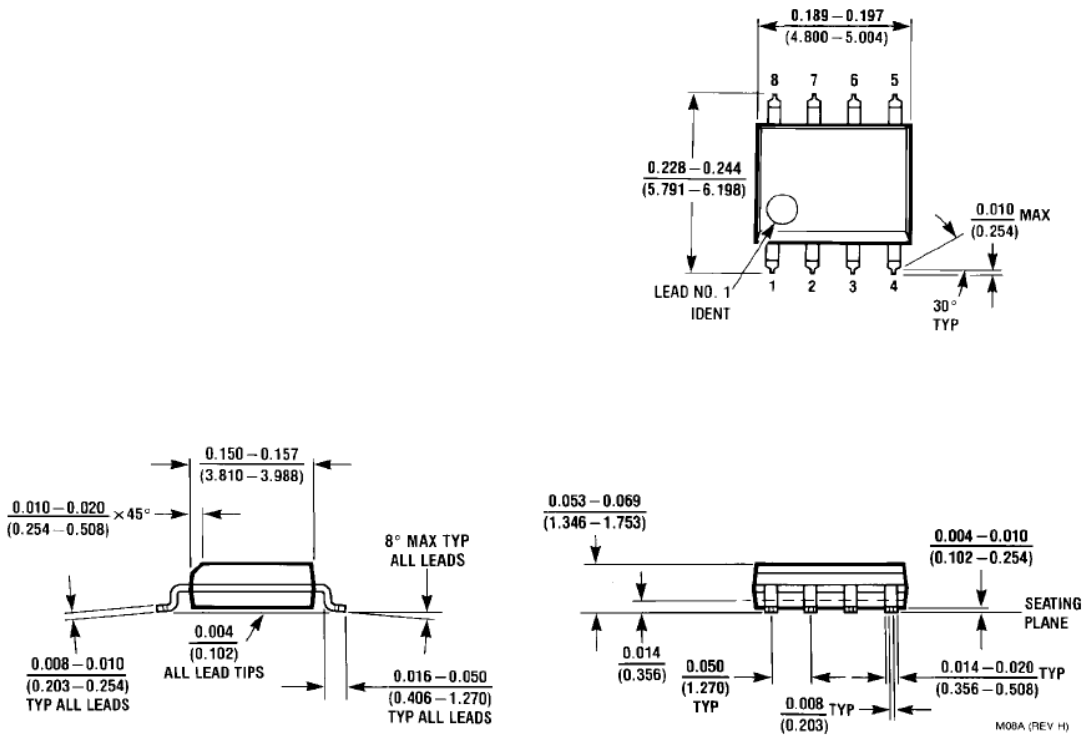
This feature will let you create your footprint based on your component datasheet.

89. Go to **Tools>>IPC Footprint Wizard (T, I)** and Wizard dialog will display. Click Next to continue your setting. **Choose SOP** and click next



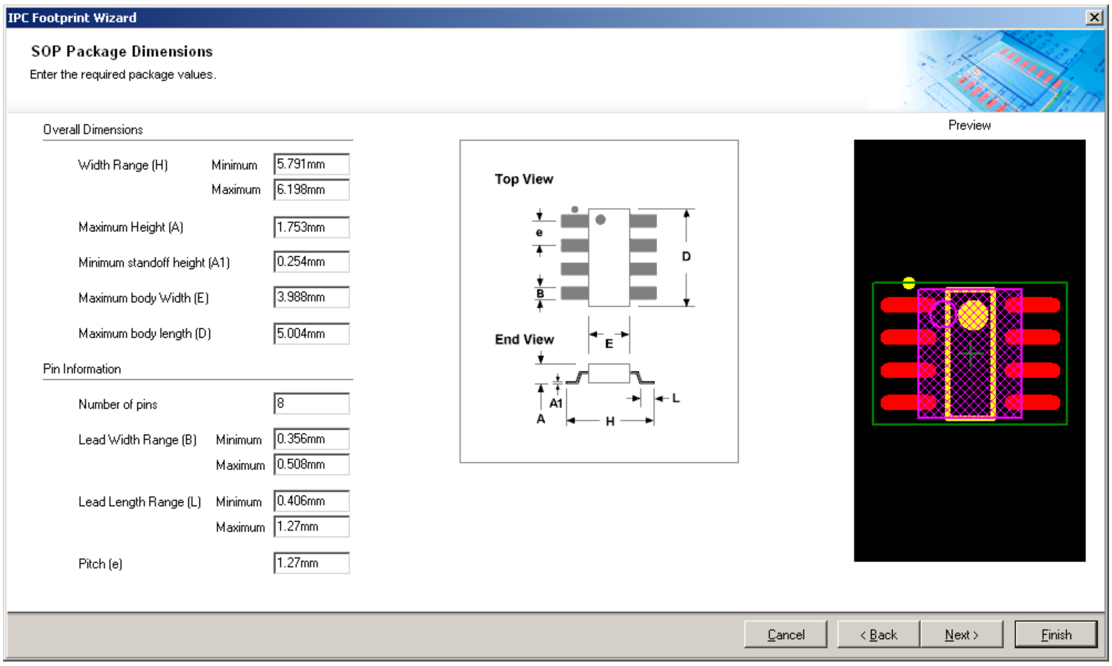
Datasheet Specification (LM 75 Temperature Sensor)

Physical Dimensions inches (millimeters) unless otherwise noted

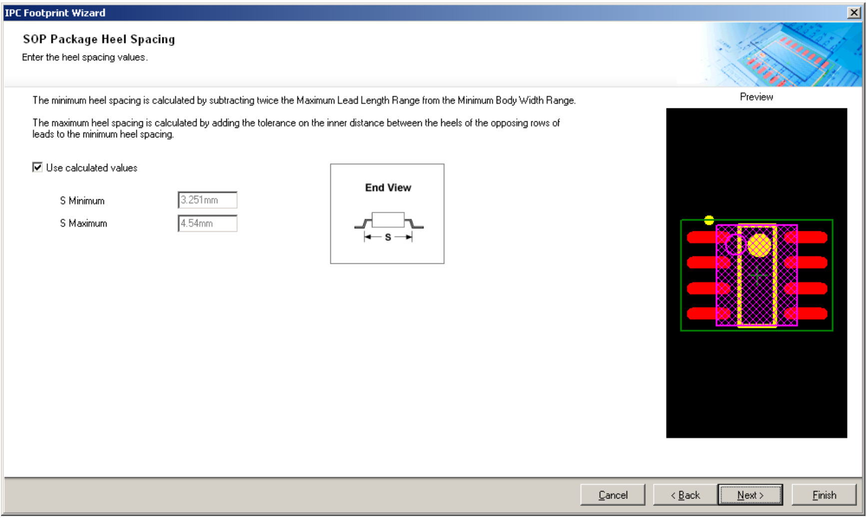


8-Lead (0.150" Wide) Molded Small Outline Package (SOP), JEDEC
Order Number LM75CIM-3, LM75CIMX-3, LM75CIM-5 or LM75CIMX-5
NS Package Number M08A

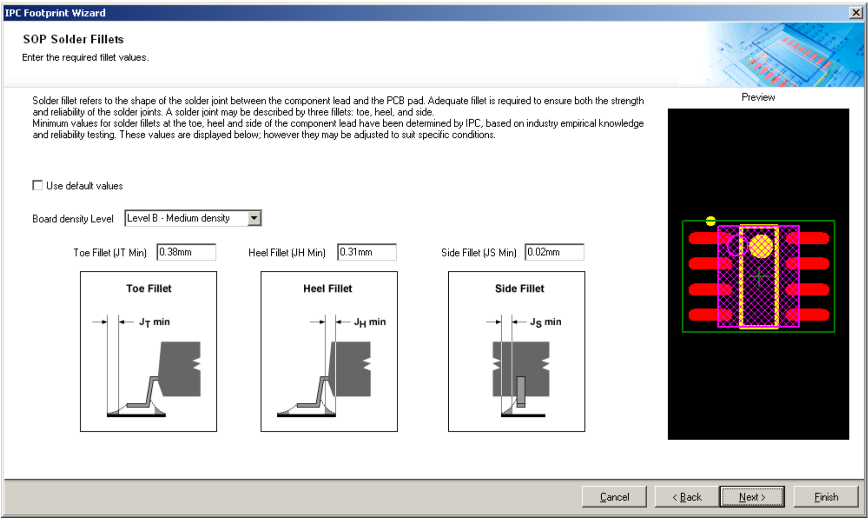
90. Key in all information detail from datasheet. And click **next** to go to the next setup.



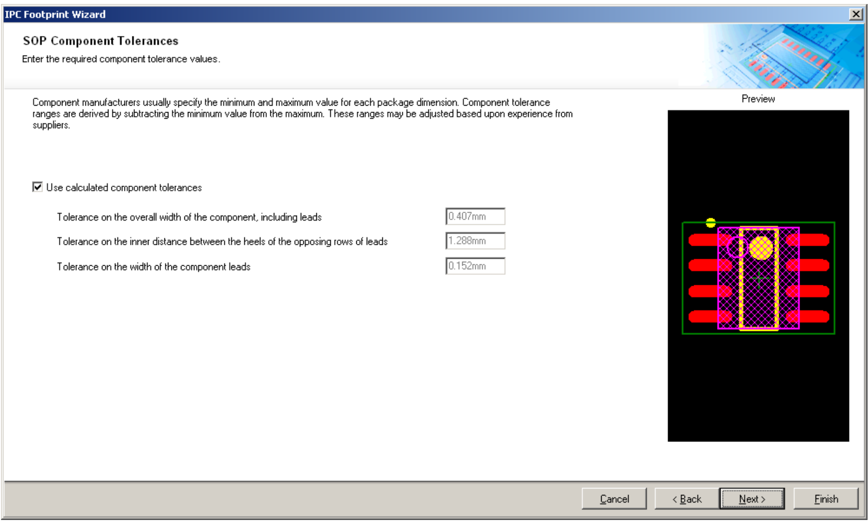
91. After Finish key in the component parameter, press **Next** to go to the next setup.

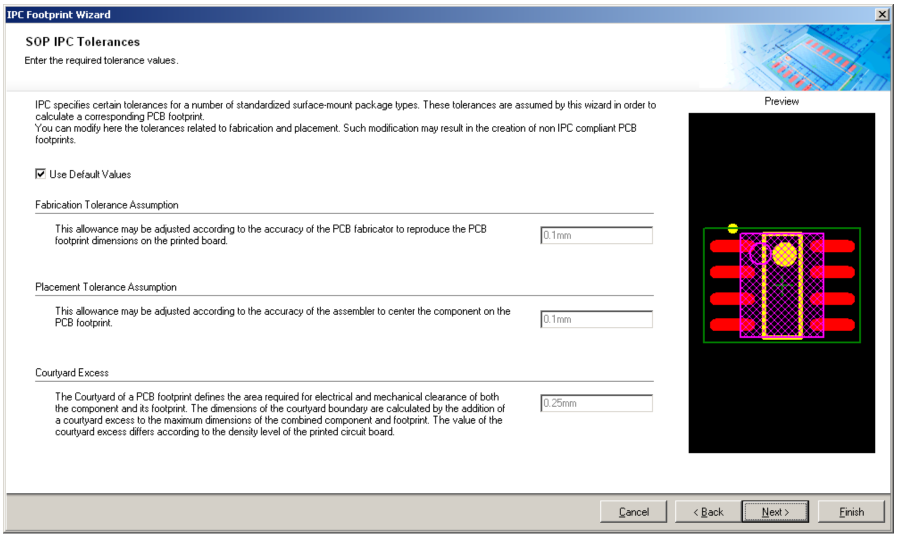


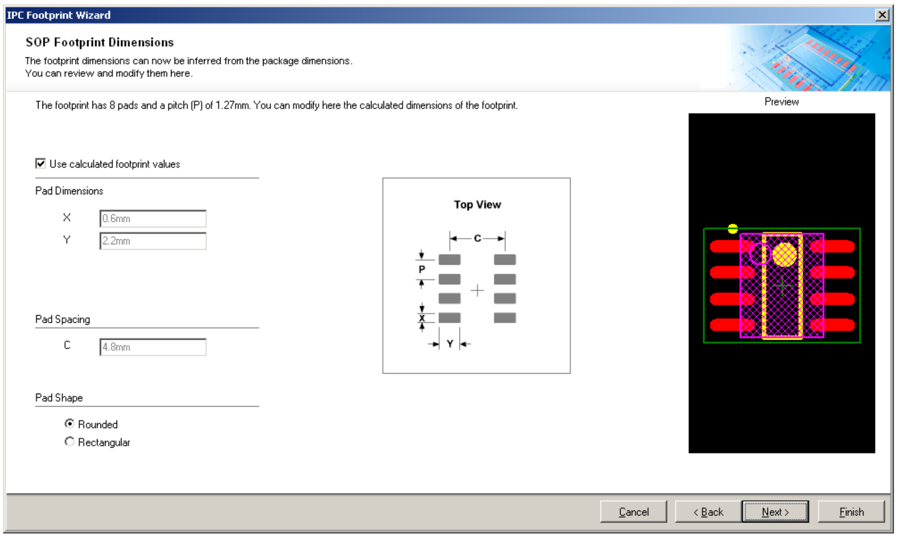
92. For Heel Spacing, you can use calculated values or can key in by uncheck the box.
93. This setup will let you to set the Solder Fillets. You can use from default value. Click **next** to continue.

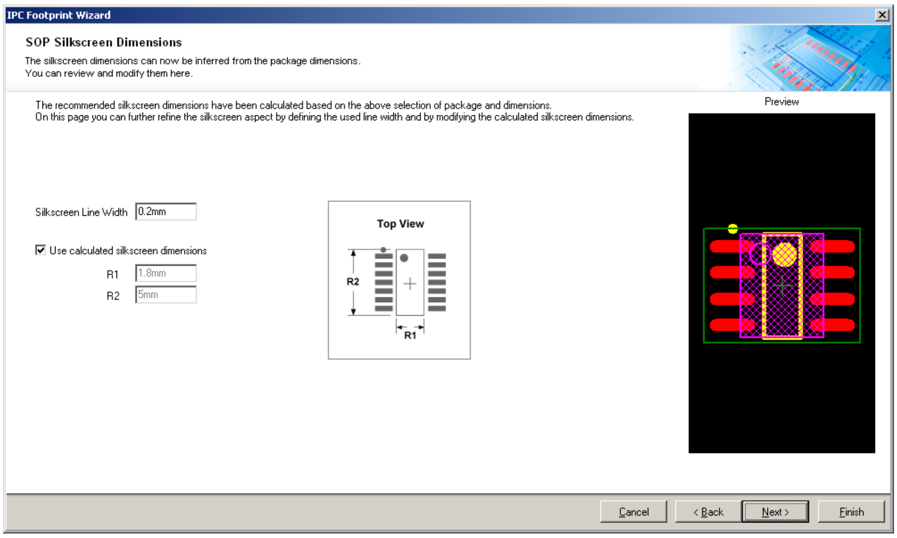


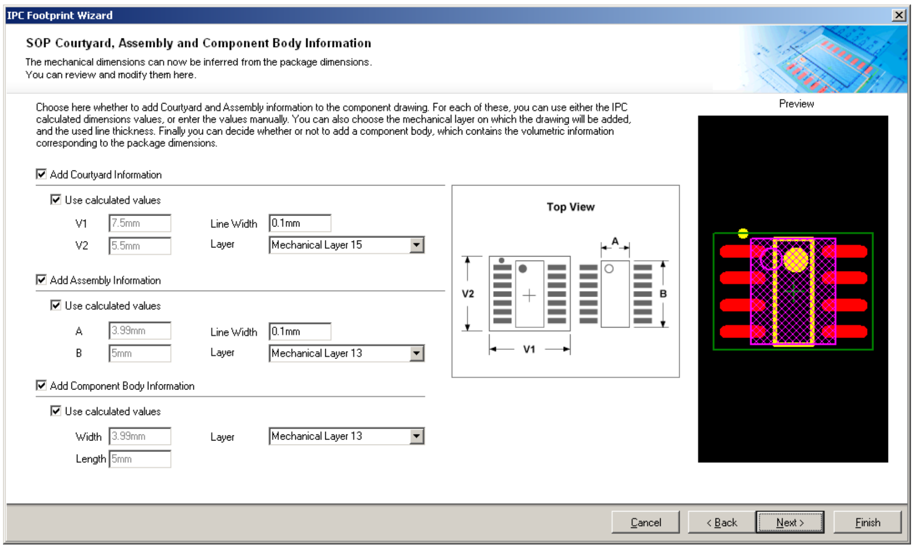
94. You can set the component tolerance for this component. Click Next to continue. Click **Next to take the default setting until Finish**.

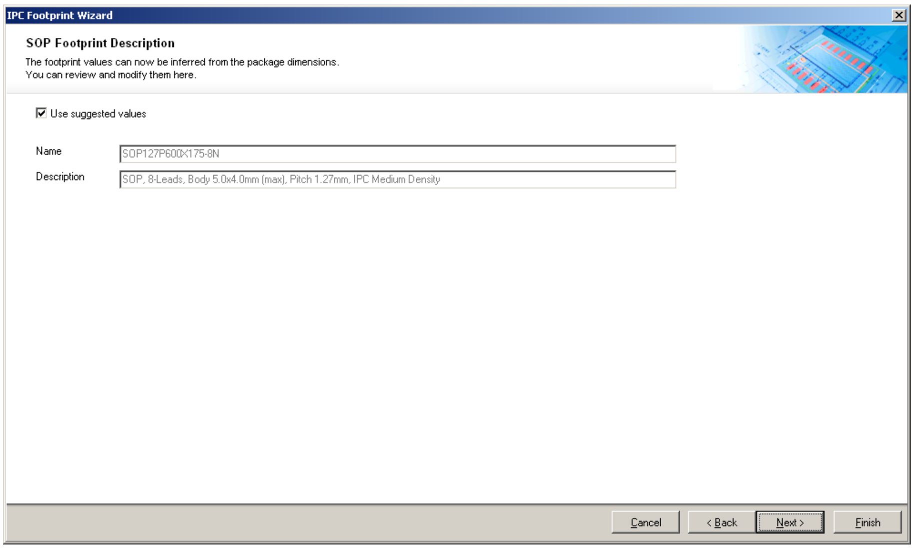


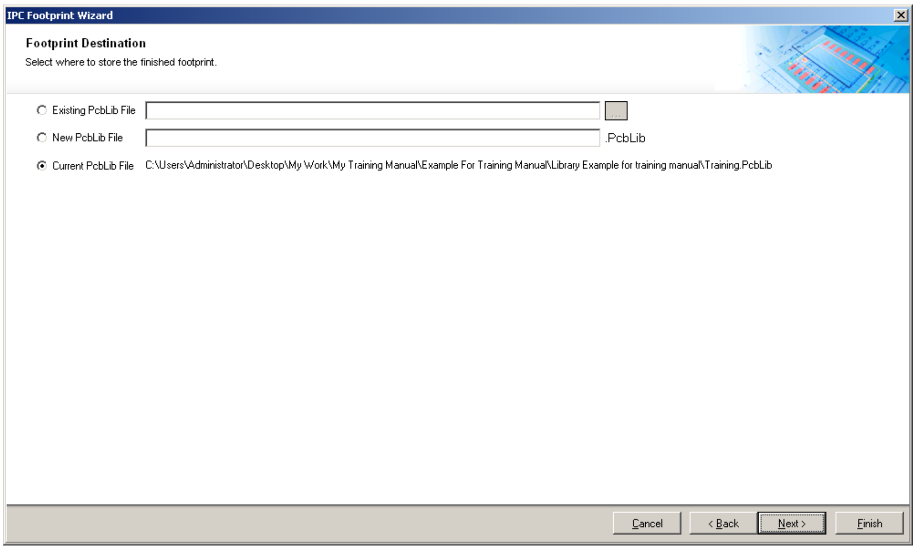

IPC Footprint Wizard
SOP IPC Tolerances
Enter the required tolerance values.
IPC specifies certain tolerances for a number of standardized surface-mount package types. These tolerances are assumed by this wizard in order to calculate a corresponding PCB footprint.
You can modify here the tolerances related to fabrication and placement. Such modification may result in the creation of non IPC compliant PCB footprints.
Use Default Values
Fabrication Tolerance Assumption
This allowance may be adjusted according to the accuracy of the PCB fabricator to reproduce the PCB footprint dimensions on the printed board.
0.1mm
Placement Tolerance Assumption
This allowance may be adjusted according to the accuracy of the assembler to center the component on the PCB footprint.
0.1mm
Courtyard Excess
The Courtyard of a PCB footprint defines the area required for electrical and mechanical clearance of both the component and its footprint. The dimensions of the courtyard boundary are calculated by the addition of a courtyard excess to the maximum dimensions of the combined component and footprint. The value of the courtyard excess differs according to the density level of the printed circuit board.
0.25mm
Preview
Cancel
< Back
Next >
Finish

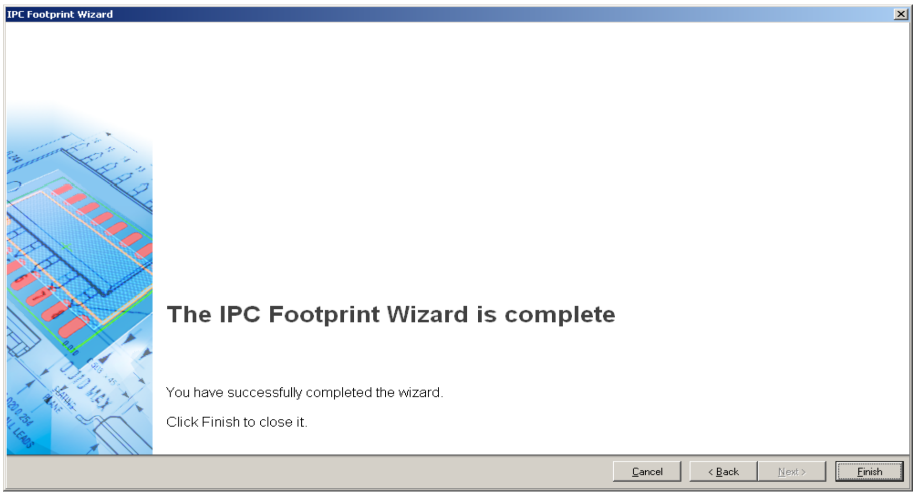

IPC Footprint Wizard
SOP Footprint Dimensions
The footprint dimensions can now be inferred from the package dimensions.
You can review and modify them here.
The footprint has 8 pads and a pitch (P) of 1.27mm. You can modify here the calculated dimensions of the footprint.
Use calculated footprint values
Pad Dimensions
X 0.6mm
Y 2.2mm
Pad Spacing
C 4.8mm
Pad Shape
Rounded
Rectangular
Top View
C
P
X
Y
Preview
Cancel
< Back
Next >
Finish


IPC Footprint Wizard
SOP Silkscreen Dimensions
The silkscreen dimensions can now be inferred from the package dimensions.
You can review and modify them here.
The recommended silkscreen dimensions have been calculated based on the above selection of package and dimensions.
On this page you can further refine the silkscreen aspect by defining the used line width and by modifying the calculated silkscreen dimensions.
Silkscreen Line Width 0.2mm
Use calculated silkscreen dimensions
R1 1.8mm
R2 5mm
Top View
R2
R1
Preview
Cancel
< Back
Next >
Finish

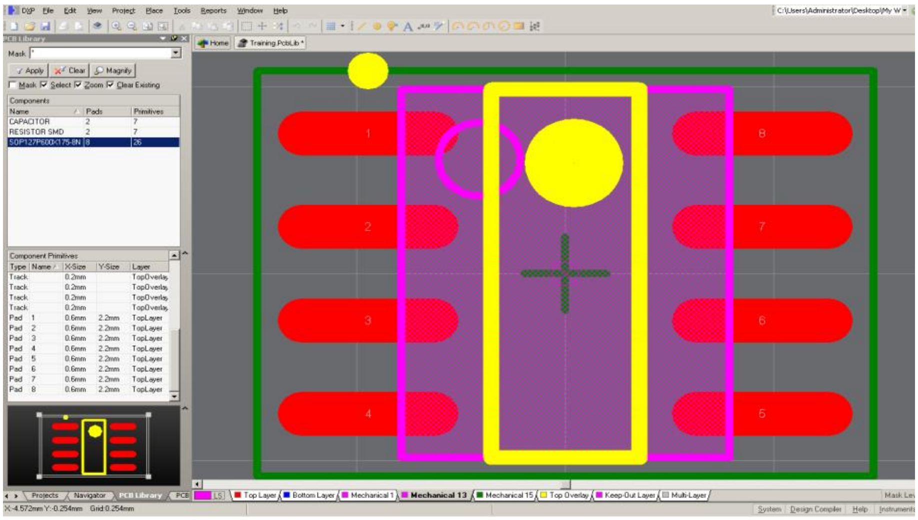

IPC Footprint Wizard
SOP Courtyard, Assembly and Component Body Information
The mechanical dimensions can now be inferred from the package dimensions.
You can review and modify them here.
Choose here whether to add Courtyard and Assembly information to the component drawing. For each of these, you can use either the IPC calculated dimensions values, or enter the values manually. You can also choose the mechanical layer on which the drawing will be added, and the used line thickness. Finally you can decide whether or not to add a component body, which contains the volumetric information corresponding to the package dimensions.
Add Courtyard Information
Use calculated values
V1 7.5mm
V2 5.5mm
Line Width 0.1mm
Layer Mechanical Layer 15
Add Assembly Information
Use calculated values
A 3.99mm
B 5mm
Line Width 0.1mm
Layer Mechanical Layer 13
Add Component Body Information
Use calculated values
Width 3.99mm
Length 5mm
Layer Mechanical Layer 13
Top View
A
V2
B
V1
Preview
Cancel
< Back
Next >
Finish


IPC Footprint Wizard
SOP Footprint Description
The footprint values can now be inferred from the package dimensions.
You can review and modify them here.
Use suggested values
Name SOP127P600X175-8N
Description SOP, 8-Leads, Body 5.0x4.0mm (max), Pitch 1.27mm, IPC Medium Density
Cancel
< Back
Next >
Finish


IPC Footprint Wizard
Footprint Destination
Select where to store the finished footprint.
Existing PcbLib File
New PcbLib File
.PcbLib
Current PcbLib File C:\Users\Administrator\Desktop\My Work\My Training Manual\Example For Training Manual\Library Example for training manual\Training.PcbLib
Cancel
< Back
Next >
Finish

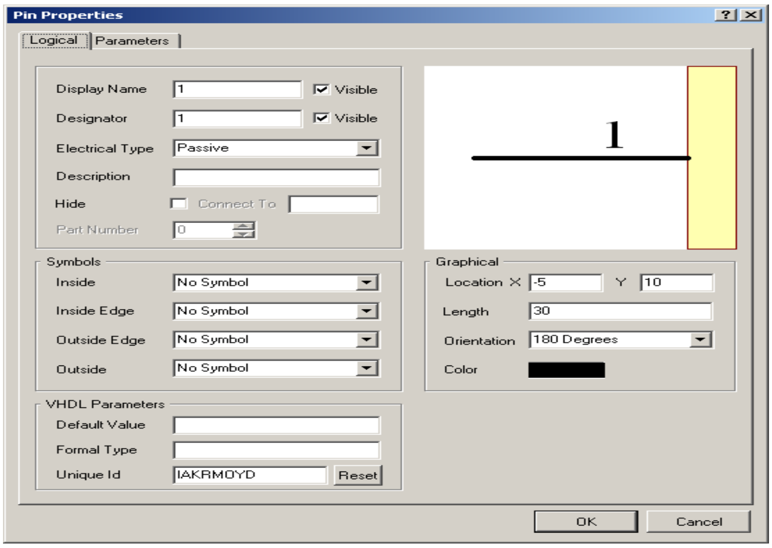


95. Now your footprint is completed, Save it by go to **File>> Save All (F, L)**

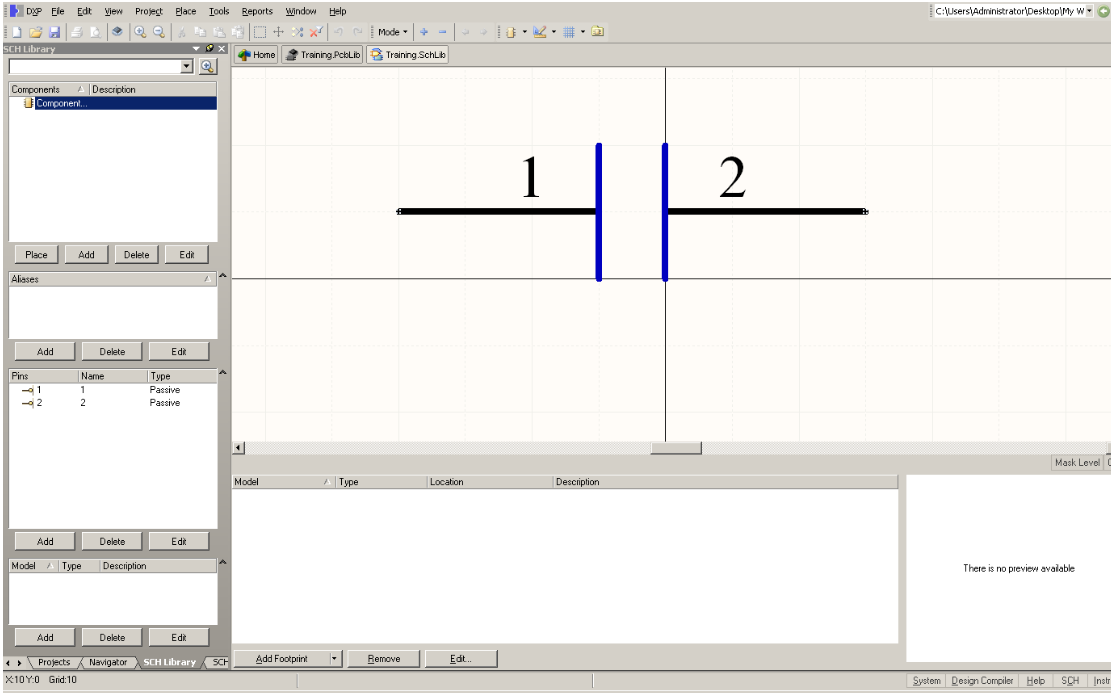


Schematic Library

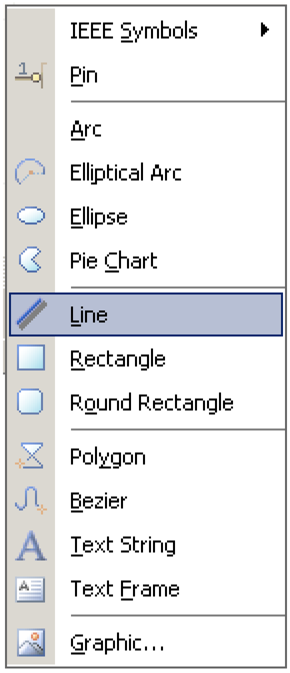
96. On Schematic editor, you can draw your schematic symbol and add footprint. Go **to File>>New>>Library>>Schematic Library (F, N, L, L)**. Place your schematic Pin by go to **Place>>Pin (P, P)**. Before the placement, press **TAB to** change the pin properties.

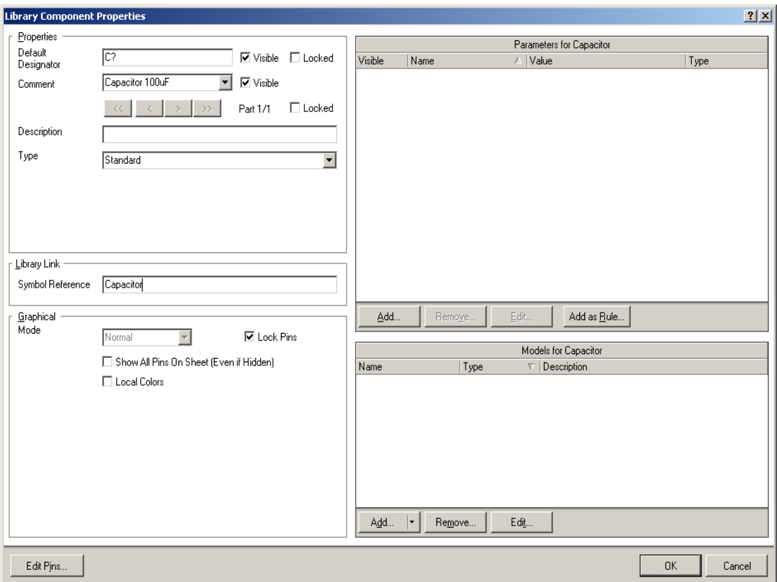


97. Set the first pin number to **1**, and after the first placement, Pin number will **automatically increase**.

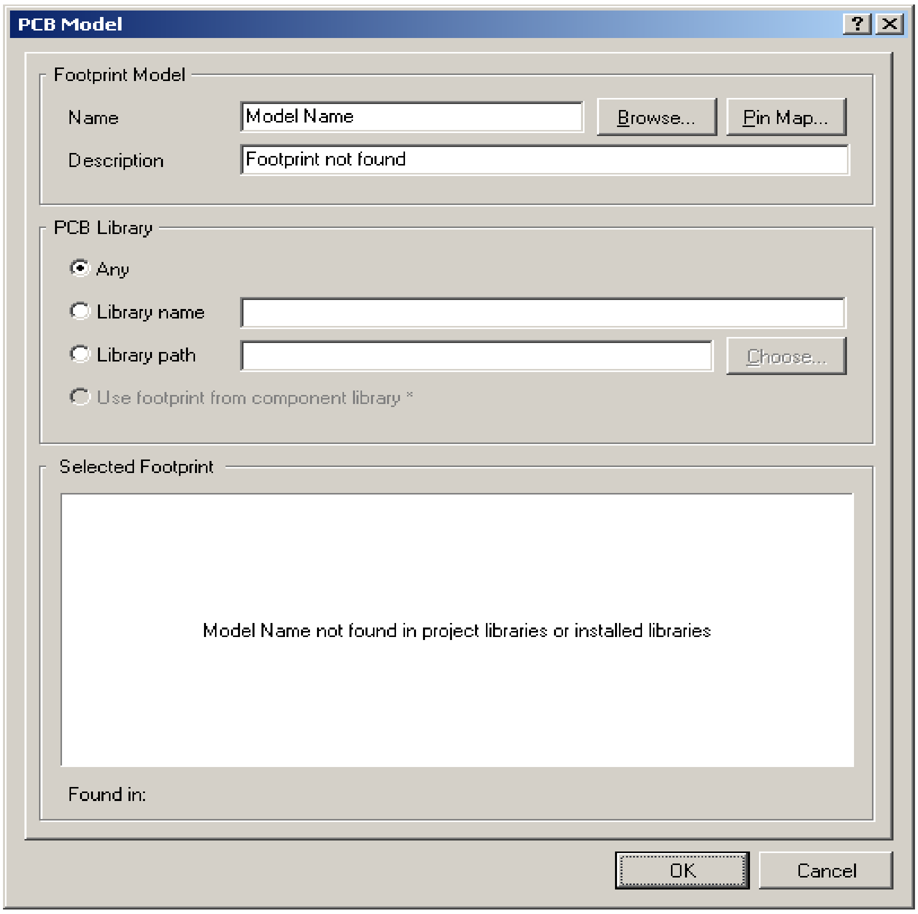


98. To draw a line, use **Place>>Line (P, L)**. These are all tools that can be used to draw schematic symbol. After finish your drawing, you can edit your component properties by double click on component name on component editor and component properties dialog will display. Edit the default designator, edit comment, change the component name, add footprint and add another parameter. Click **OK** to apply changes. To change component name, go to **Tools>>Rename Component (T, E)**

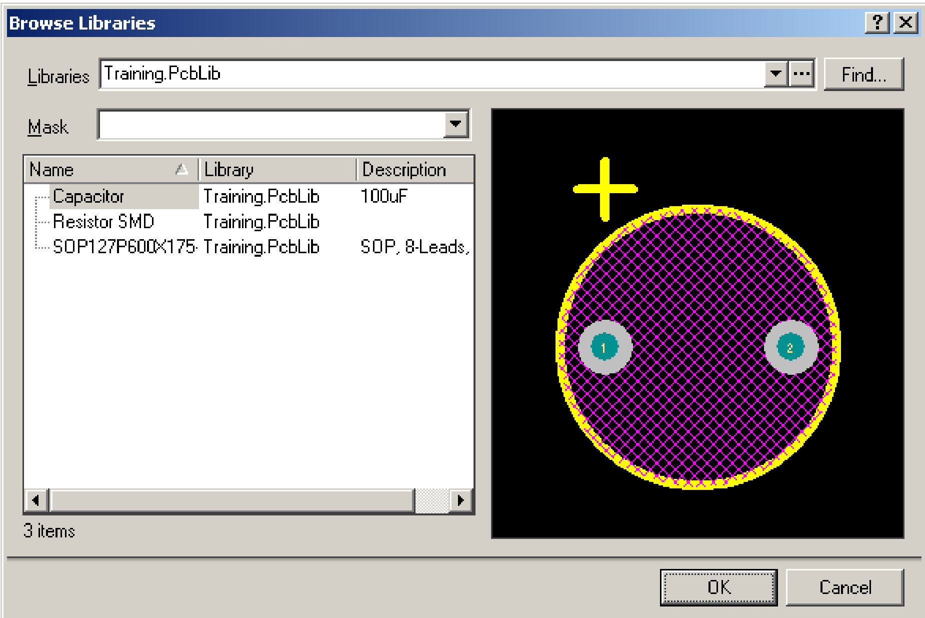




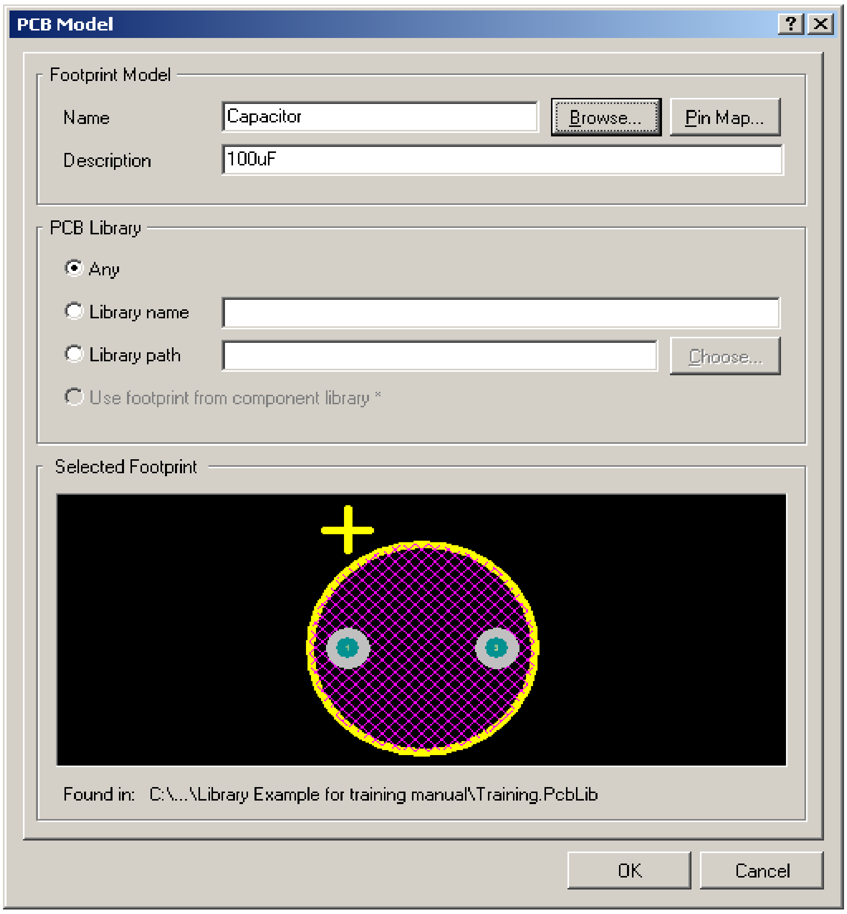
99. To add the footprint, Click **Add Footprint** and footprint model will display.



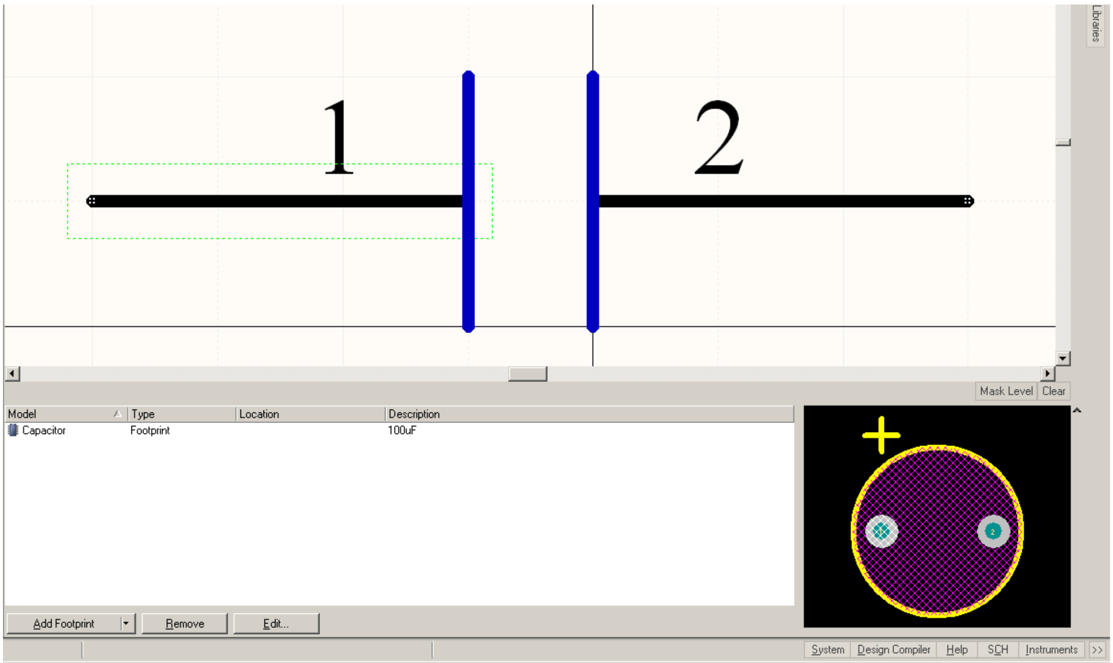
100. Click browse to add your footprint. **Choose** which **footprint** you want to use, press **OK** to Add.



101. Now your schematic symbols have a footprint. **Click OK to apply**.



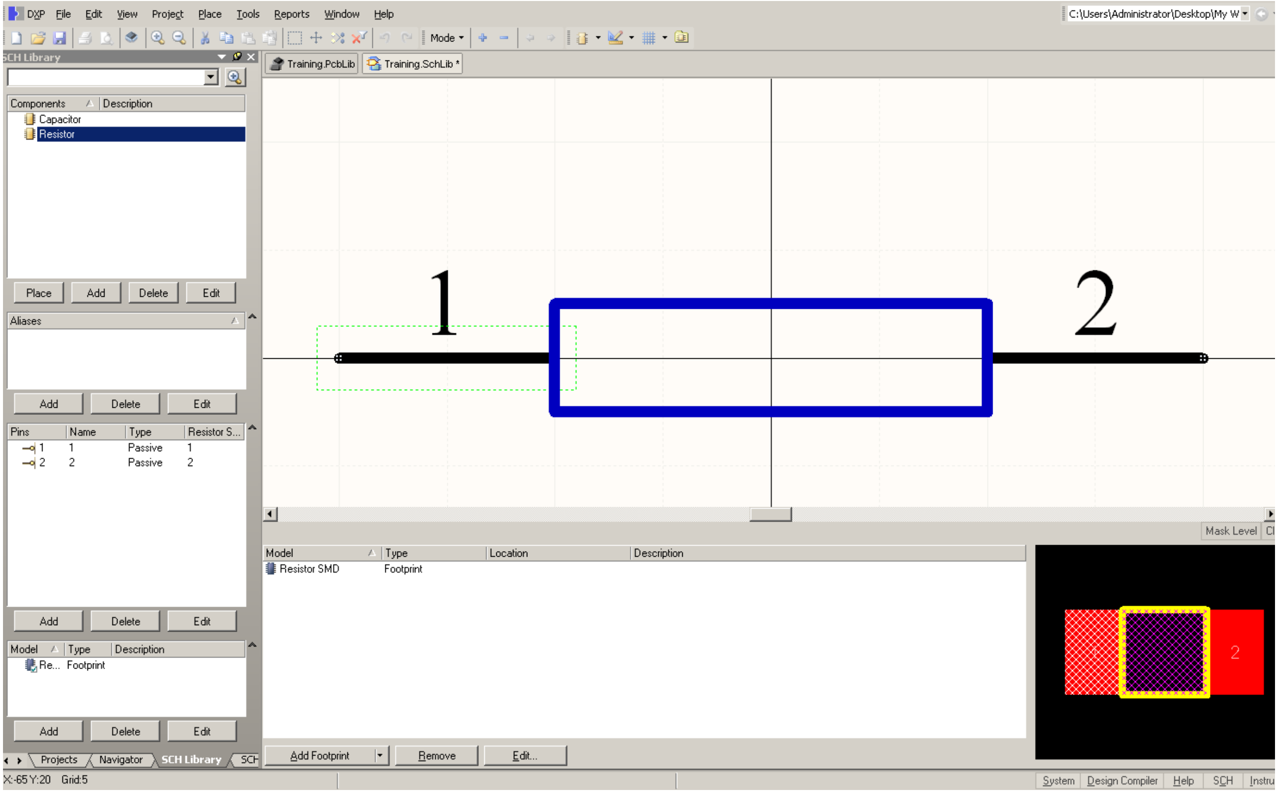
102. You can cross check for the connection between Schematic symbols and Footprint.

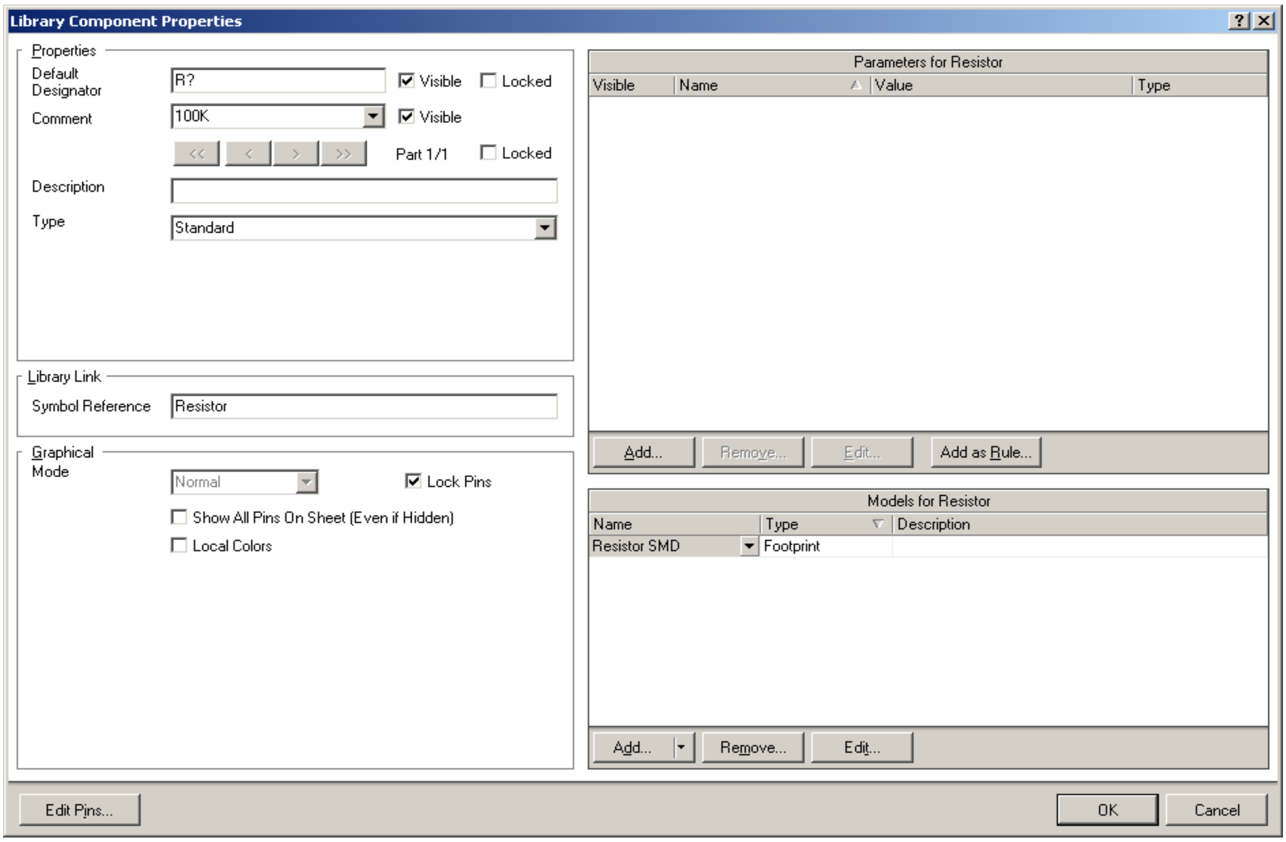


Note: Schematic and footprint is linked based on Pin number on schematic symbol and Pad number on footprint. One schematic symbol can have more than 1 footprint

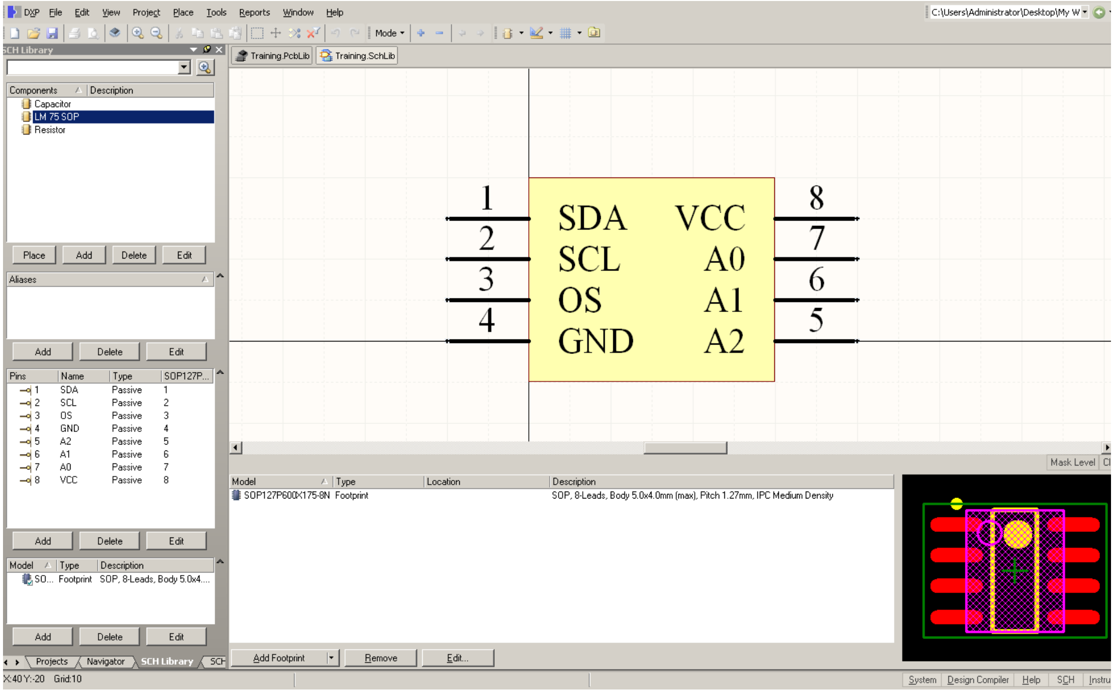
Exercise

103. Do the same step for Resistor, Draw your Schematic symbol, add footprint and edit the component properties

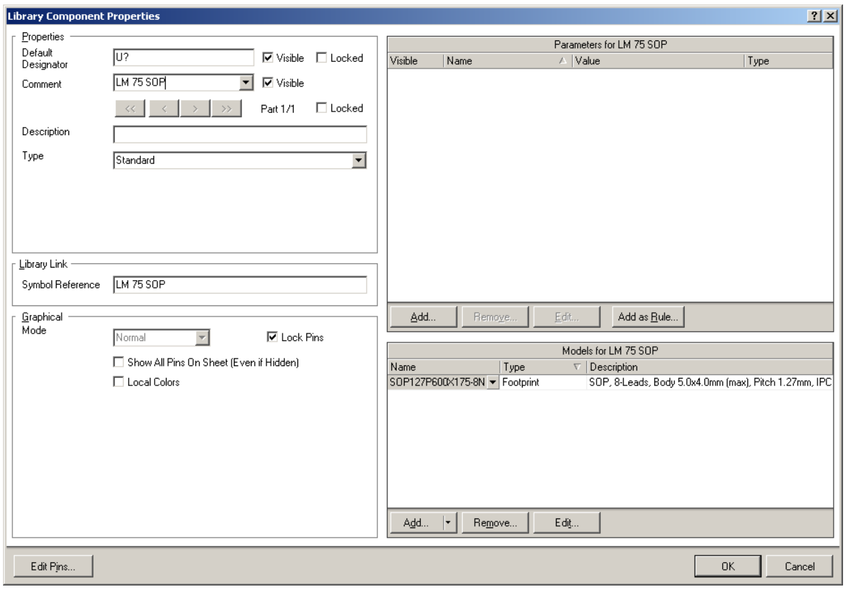

DXP File Edit View Project Place Tools Reports Window Help
SCH Library
Training.PcbLib
Training.SchLib *
Components
Description
Capacitor
Resistor
Place Add Delete Edit
Aliases
Add Delete Edit
Pins Name Type Resistor S...
1 1 Passive 1
2 2 Passive 2
Add Delete Edit
Model Type Description
Re... Footprint
Add Delete Edit
1
2
Model Type Location Description
Resistor SMD Footprint
Add Footprint Remove Edit...
Mask Level
Projects Navigator SCH Library SCH
X:-65 Y:20 Grid:5
System Design Compiler Help SCH


Library Component Properties
Properties
Default Designator
R?
Visible
Locked
Comment
100K
Visible
Part 1/1
Locked
Description
Type
Standard
Library Link
Symbol Reference
Resistor
Graphical
Mode
Normal
Lock Pins
Show All Pins On Sheet (Even if Hidden)
Local Colors
Parameters for Resistor
Visible Name Value Type
Add... Remove... Edit... Add as Rule...
Models for Resistor
Name Type Description
Resistor SMD Footprint
Add... Remove... Edit...
Edit Pins...
OK
Cancel

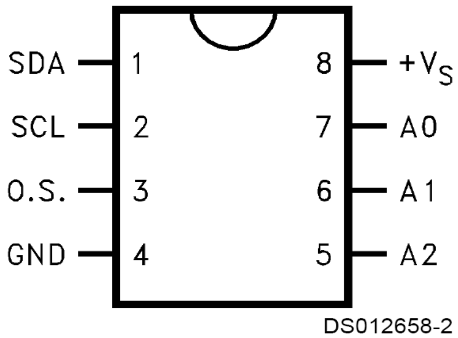
104. Draw your Schematic symbol same as shown below,



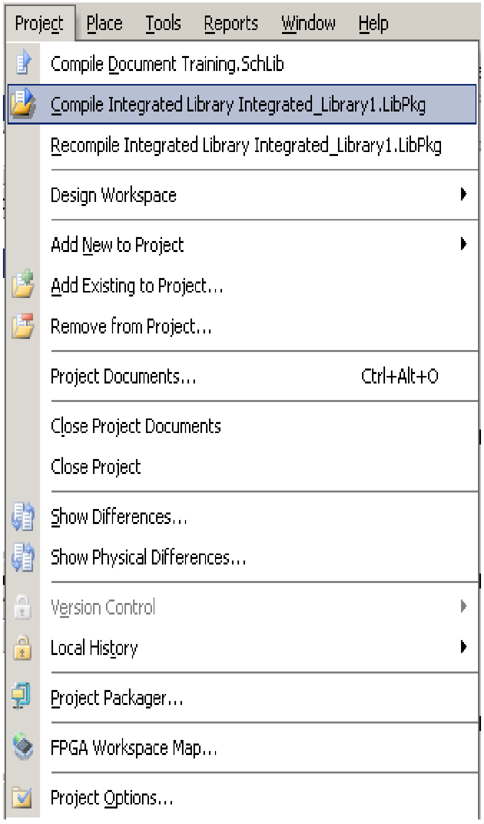
Note: To draw LM75 body, use rectangle. You can get this from **Place>>Rectangle (P, R)**

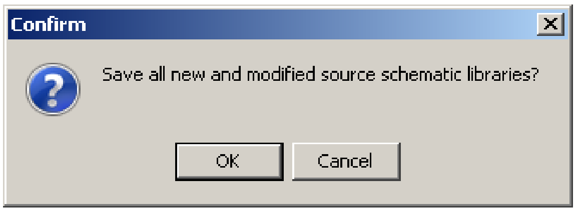


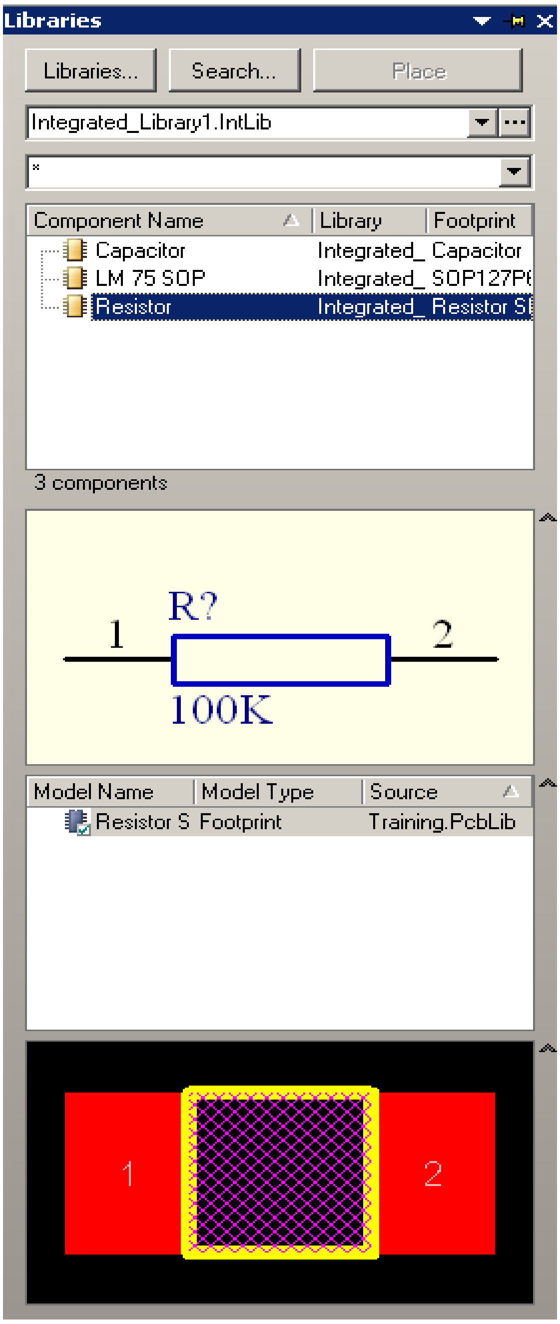
SOP-8 and Mini SOP-8



105. After build all libraries, now it's time to build your library becomes integrated library. Go to **Project>>Compile Integrated Library QQQ.LibPkg (QQQ is your integrated library name) (P, C)** and confirmation window will appear, click **OK** to continue.







106. After you compile, your new integrated library **will be automatically installed on your library and ready to use.**