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 <u>File>>New>>Project>>PCB</u> Project menu to create new PCB project.



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



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

Save the Project and Schematic file to your folder. Eile * Edit * View * Project * Place * Design * Tools * Reports * Window * Help *



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.

 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.

aining.SchDoc			Libraries 🔻 🖡
			Libraries Place 2N3904
	VCC Libraries Search [2] X		Component / Library Description Foo / 213904 Miscell. NPN General F T0-32 203906 Miscell. PNP General F T0-32 ADC-8 Miscell. Generic 8-8it / S OT 4(
	2N3904		Anterna Miscell Genetic Anter HINI Miscell Multicell Batter BAT-2 Bell Miscell Electrical Bell PIN2 Bridge1 Miscell Diode Bridge D-46 (195 components
	Options Search type Advanced	C2	Q? 2N3904
	Scope Path [®] Available libraries Path: [®] Libraries on path [®] Include Subdirectories [®] Reline last search [®] Include Subdirectories	20nF Q1 2N390	Model Name Model Type Source /
	Cancel		K TO-92A Footprint Miscellaneous I 🚯

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 **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.

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



 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

- 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

Component Proper	ties						<u>? ×</u>
Properties						Description for C1 - Con-	
Designator	C1	🗹 Visible 🔲 L	ocked	Visible	Name	A Value	Tupe
		E VG-31-			LatestRevisionDate	17-Jul-2002	STRING
Lomment					LatestRevisionNote	Re-released for DXP Platfo	rm. STRING
	$\langle \langle \rangle \rangle$	Part 1/1 🗌 L	ocked		PackageReference	RAD-0.3	STRING
Description	Capacitor				Published	8-Jun-2000	STRING 🔽
Unique Id					Publisher	Altium Limited	STRING
Onique la	DGKAYIXI	F	leset	✓	Value	20nF	STRING
Туре	Standard		•				
_ Library Link							
Design Item ID	Сар	С	hoose				
🗹 Library Name	Miscellaneous Devices.IntLib						
🗹 Table Name							
		V	alidate	L L L	Dama I	549 A44	
Sub-Design Links -				<u>A</u> dd	Hemo <u>v</u> e	Edit Add as <u>H</u> ule	
Sub-Project	None					Models for C1 - Cap	
Canfiguration	None			Name	Туре	V Description	
Configuration	INDIRE			CAP	Simulatio	on Capacitor	
Graphical				Lap 1/000.00	Signal In	tegrity Canacitor Coramio The	w Hole, Disped Radial: Rody 5.1 v
Location X	490	r 510		11 02 0.2	roopini	capacitor, coranio, mi	arroid, Dipped Fradial, Dody 3.1 x
Orientation	0 Degrees 💌	Mirrored					
Mode	Normal	🔽 Lock Pins					
	Show All Pins On Sheet (Even	if Hidden)					
	Local Colors					E N	
				<u>Add</u>	Hemove	E Oli	
Edit Pjns							OK Cancel

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.
-----	--------	-----	-------	------	----	-------	--------

Analyses Setup				<u>? ×</u>
Analyses/Options	Enabled	Transient/Fourier Analysis Setup		
General Setup		Parameter		Value
Operating Point Analysis	✓	Transient Start Time	0.000	
Transient/Fourier Analysis		Transient Stop Time	10.00m	
AC Small Signal Analysis		Transient Step Time	20.00u	
Noise Analysis	H	Transient Max Step Time	20.00u	
Pole-Zero Analysis		Use Initial Conditions	<u> </u>	
Transfer Function Analysis				
Temperature Sweep		Use Transient Defaults		\frown
Parameter Sweep		Default Cycles Displayed	5	
Monte Carlo Analysis Global Parameters		Default Points Per Cycle	50	
Advanced Options				
		Enable Fourier		
		Fourier Fundamental Frequency	1.000mea	
		Fourier Number of Harmonics	10	
			Set <u>D</u> efaults	
Preferences			ОК	Cancel

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

DXP Elle Edit Yjew Project Io	iols Chart	Blot Ways	Window Help	☆芥末:											C:\Use	ers\Administrato	r\Deskt	op\Traini 🔹 🧲
Projects	▼ 9 X	글 Training.S	chDoc 🔣 Training.:	sdf *														
Workspace1.Dsr/Wrk Vo	orkspace		15.00					\rightarrow									_	
Franing.PijPcb	Project	ε	10.00 7.500							_		-					1	
File View O Structure Editor			2.500									.						
Training PriPcb * Source Documents Training SchDoc	B		0.000 0.000m	1.000m	2.000m	3.000	m	4.000	Dm 5 T	5.000m fime (s)	6.00	Om	7.000	m	8.000m	9.000	Dm	10.00n
E Generated		1	15.00															
AdvancedSim Netlists Simulation Documents		S	10.00	1						-					\square			
🔤 Training.sdi ×	B	1 °	5.000															
			0.000 0.000m	1.000m	2.000m	3.000	m	4.000	Dan 5	5.000m	6.00	0m	7.000	m	8.000m	9.000)m	10.00n
		-	2.500							nine (s)								
		ε	2.500 2.500 5.000 7.500 10.00					_				_						
			0.000m	1.000m	2.000m	3.000	m	4.000	Dan S T	5.000m fime (s)	6.00	0m	7.000	m	8.000m	9.000	Dm	10.00m
		ε	2.500 0.000 2.500 5.000 7.500 10.00 12.50											1 1 1				
	- 8		0.000m	1.000m	2.000m	3.000	m	4.000	0m € ⊺	5.000m fime (s)	6.00	Orn	7.000	m	8.000m	9.000	Dm	10.00n
	1	<u>.</u>																
Files Projects Navigator Sim Dat	ia /	Operating Po	int)(Transient Analysis	/														Mask Levi
Messages																		
Class	Document		Source		Message											Time	D	ate
[Start Output]			Ouput Generator		Start Output	Generation At 5:	19:51 PM 0)n 27/9/	2007							5:19:51 PM	27	//9/2007
Ca [Output]			Ouput Generator		Name: Mixe	d Sim Type:Adv	/SimNetlist	From: P	roject (Training.)	PriPcb]						5:19:51 PM	27	//9/2007
(Hint)	Training.Schl	Joc	AdvSim		U1 · Model I	ound in: C:\PRO	GHAM FILE	:SVALTI	UM DESIGNER	1 6\Library\	Miscellaneou	s Devices.	IntLib			5:19:51 PM	27	79/2007
[[Lenerated File]			Ouput Generator		Fraining, nsx	and Connection 1	1540.F1 7	u o	10/0007							5:13:51 PM	21	79/2007
Le [ninished Uutput]			ouput Generator		r inished Uu	iput Generation A	x p: 19:51 H	m Un 2/	/3/2007							0:10:01 PM	2	75/2007

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**).

omponent Proper	ties		No. And Annual Structure	
Properties			Browse Libraries	
Designator	12/nos	Parameters for 12Vpos - VSRC		
Designation		Visible Name / Value lijpe	Libraries Miscellaneous Connectors.IntLib [Footprint View]	▼ …
Comment	VSRC Visible	LatestRevisionNote Reveleased for DXP Platform. STRING		
	<< > >> Pat 1/1 Locked	Note PCB Footprint - Not required STRING	- H - I	
Description	history Carrow	PackageReference Not Applicable STRING	Mask	
	vukage souce	Published 8-Jun-2000 STRING		
Unique Id	ATFYQMwX Reset	Publisher Altium Limited STRING	Name 🛆 Library Description 📥	
Туре	Standard 💌	Value +12 STRING	- CHAMP1.27-2H68 Miscellaneous Conni Connector:	2
Liboon Link			- CHAMP1 27-2H10f Miscellaneous Conni Connector:	
Davies New ID	http:		CHAMP1 27 2/00, Missellaneous Come Connectory	
nesiðunigu in	Choose		DOUDT OSS OLID ALS IN DOUD CONTRECTOR	
Library Name	Simulation Sources.IntLib		-DSUB1.385-2H9 Miscellaneous Conni Connector;	
🗹 Table Name			-DSUB1.385-2H15 Miscellaneous Conni Connector;	
	Validate		- DSUB1 385-2H254 Miscellaneous Conni Connector:	
	10000	Add Remoye Edt Add as <u>Rule</u>	HDP1Y2 Missellaneous Const Consectory	
Sub-Design Links -				
Sub-Project	None	Models for 12Vpas - VSRC	HDH1X2H Miscellaneous Lonni Lonnector;	
Configuration	None	Name Type T Description	HDR1X3 Miscellaneous Conni Connector;	
Granhinal		Vanc Vanc	HDR1X3H Miscellaneous Connector:	
Fichards	200		HDB1X4 Miscellaneous Comp Connector:	
Location X	Y 530		UDD1V/U Miscellaneous Comil Connector,	
Unentation	0 Degrees 💌 🗖 Minored		HUR IX4H Miscellaneous Lonni Lonnector;	
Mode	Normal 🗾 🔽 Lock Pins			
	Show All Pins On Sheet (Even if Hidden)	<u> </u>	181 items	
	Local Colors	Add. Remove. Edt.		
		Footprint		
care I		Simulation	UK	
Edit Miria		PCB3D UK Cano		

- 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

DXP Eile Edit View Project Place Design Io	ols <u>R</u> eports <u>W</u> indow <u>H</u> elp	DXP Eile	e <u>E</u> dit <u>V</u> iew Proje <u>c</u> t <u>P</u> lace <u>D</u> e	esign <u>T</u> ools <u>R</u> eports <u>W</u> indow <u>H</u> elp
🗈 🚅 🛃 🎒 💁 🔍 🧶 의 🗔 🕫 📲 🔞 🐚	🖺 🗐 🖂 🕂 🛪 📈 🔊 🕬	🗋 🗁 🔜	New •	🚽 Schematic 💦 😪 🖂 🕫 💵
Dy> Ele Edt Yrev Project Blace Design Io Project Warkspace1.DenWrik Warkspace1.DenWrik	Also Beports Window Help Image: Second	Vorksp Training File V	Edit View Project Place DV New Open Ctrl+P Import Copen Design Workspace Save Adl Save Adl Save Copy As Save Copy As Save Design Workspace As Page Setup Prink Preview Prink Preview Prink Preview Save Adl Save Design Workspace As Save Design Workspace As Page Setup Prink Preview Prink Preview Savet Documents Recent Projects Recent Design Workspaces	sign Tools Reports Window Help Schematic PCB U Verlog Document U Verlog Document C Header Document D C Source Document C Header Document D C Source Document D L D C C M D C C C C C C C C C C C C C C C
Local History Project Packager Project Qptions			Recent Design Workspaces	

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



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

PEB Board Wizard Choose Board Profiles Select a specific board type from the predefined standard choose custom.	profiles or
[Custom] A A0 A1 A2 A3 A4 AT long bus (13.3 x 4.2 inches) AT long bus (13.3 x 4.2 inches) AT long bus (13.3 x 4.5 inches) AT long bus with break-away tab (13.3 x 4.2 inches) AT long bus with break-away tab (13.3 x 4.2 inches) AT long bus with break-away tab (13.3 x 4.5 inches) AT short bus (7 x 4.2 inches) AT short bus (7 x 4.2 inches) AT short bus with break-away tab (7 x 4.2 inches) AT short bus with break-away tab (7 x 4.5 inches) AT short bus with break-away tab (7 x 4.8 inches) B B B	
	Cancel < Back Next > Finish

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.

PCB Board Wizard Choose Board Details Choose Board Details	×
Outline Shape: © Rectangular © Circular © Custom Board Size: <u>W</u> idth 1000 mil <u>H</u> eight 1000 mil	Dimension Layer Mechanical Layer 1 ▼ Boundary Irack Width 10 mil Dimension Line Width 10 mil Keep Out Distance 0 mil From Board Edge Corner Cutoff ✓ Itile Block and Scale Corner Cutoff ✓ Legend String Inner CutOff ✓ Dimension Lines
	<u>C</u> ancel < <u>B</u> ack <u>N</u> ext > ⊟inish

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.

			×
for your		$\langle \bigcirc$	
Cancel	< <u>B</u> ack	<u>N</u> ext >	Einish
	for your	tor your	tor your

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.

B Board Wizard	
Choose Via Style	
Choose the routing via style that is suitable for your design.	
Thrubale Vias andu	
C Blind and Buried Vias only	
	Cancel < Back Next > Einish

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

PCB Board Wizard	×
Choose Component and Routing Technologies	
Choose the component and routing style that you intend to us	e CSS
The board has mostly:	
 Surface-mount components. 	
C Through-hole components.	
Do you put components on both sides of the board?	
O Yes	
⊙ No	
	Cancel < Back Next > Einish

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

PCB Board Wizard				×
Choose Default Tra Choose the minimum track clearances to use on the	ck and Via siz (size, via size and new board.	z es d the copper to copper		
Minimum <u>T</u> rack Size	<u>8 mil</u>	↓		×
Minimum Via <u>W</u> idth	<u>40 mil</u>			
Minimum Via <u>H</u> oleSize	<u>25 mil</u>			
Minimum <u>C</u> learance	<u>8 mil</u>	Ţ		
		Cancel	< <u>B</u> ack <u>N</u> ext >	Einish

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

PCB Board Wizard	Altium Designer Board Wizard is complete You have successfully completed the Board Wizard. To close this wizard and create the board click Finish.	×
	Cancel < Back Next> Einish	

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).)

P DXP File Edit View Project Place Design Tools Auto Route Reports	DXP Eile Edit View Project Place Design Tools Auto Route Report:
	🗈 🐸 🛃 💩 🗶 🔍 🔍 🐼 🐼 🕹 🐁 🕾 🕄 🖂 🕂 📈
Projects V V Training.SchDoc V PCB1.Pcb	Projects 🗸 🗸 🔽 Training SchDoc 🔛 Training
Workspace1.DsnWrk Vorkspace	Workspace1.DsnWrk
Project	Training DriDab
🖸 File View 🔿 Structure Editor 🛛 😒 💷 💌	Project
Training, PriPcb *	File View C Structure Editor
Source Documents	🗆 🗾 Training. PriPcb
PCB1.PcbDoc	Source Documents
	Training.SchDoc
Gree Documents Source Documents	E Generated
🕎 PCB2.PcbDoc *	
5	
• • • • •	
(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 <u>D</u>esign>><u>Import Changes from qqq.PrjPcb (qqq is your project name.) (D, I)</u>

	D <u>X</u> P	Eile	<u>E</u> dit	<u>V</u> iew	Proje <u>c</u> t	<u>P</u> lace	Desi	ign	<u>T</u> ools	<u>A</u> uto Route	<u>R</u> eports	Win	dow	Help	1 🔛	•
	1 🖻		a	۲	🔉 🔍	्रः 🟹		Upd	ate Sche	ematics in Trair	ning.PrjPcb		2	1	11	I
Pr	ojects			1		-		Imp	ort Char	nges From Trail	ning.PrjPcb		oc			ľ
Γ	Worksp	ace1.	DsnWrk		-	Worksp		<u>R</u> ule	s							
ľ	Trainin	n PriPr	•h			Projer		Rule	e <u>W</u> izard							
	T File 1	gan na c	C Shuo	huro Ed	itor			Boa	rd <u>S</u> hape	е		•				
	o File V	1ew	o suuc	luie Eu	itor			<u>N</u> eti	ist			•	-			
		l raini	ng.PrjP	cb				Laye	er Stac <u>k</u>	Manager						
		IIIII Ti	raining.Po	cbDoc		B		Boa	rd La <u>v</u> er	rs & Colors	L					
		TI	raining.S	chDoc		Ĩ		Man	iage Lay	er Se <u>t</u> s		•				
	+	Lien	erated					Roo	<u>m</u> s			•				
								⊆las	ses							
							識	Brow	vse Com	ponents						
								Add	/Remov	e <u>L</u> ibrary						
								Mak	e <u>P</u> CB Li	brary						
								M <u>a</u> k	e Integr	ated Library						
								Boa	rd <u>O</u> ptio	ns						
										<u> </u>			<u>ا</u>			
										_•						
									. (\square						
										\supset						
																ſ

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

DXP File Edit View Project Place	Des	ign <u>T</u> ools <u>R</u> eports <u>W</u> indow <u>H</u> elp	C:\Users\Administra	tor\Desktop
🗈 💕 🖬 🍠 🛕 🗶 🔍 🔍 🔍		Update PCB Document Training,PcbDoc	🗐 🖓 🎉 🗢	: 7= K
Projects 🗸 🗸	13	Browse Library		
Workspace1.DsnWrk	11	Add/Remove Library		
Training PriPch Project		Make Schematic Library		
		Make Integrated Library		
		Template		
E B Source Desuments		Netlist For Project		
IIII Training.PcbDoc		Netlist For Document	1 R2	
Training.SchDoc 🗎		Simulate	00K 🗧	
		Create Sheet From Symbol	1K	
		Create HDL File From Symbol		
		Create Sheet S⊻mbol From Sheet Or HDL		
		Create Component From Sheet	P1	
		Rename Child Sheet		
		Synchronize Sheet Entries and <u>P</u> orts		
		Document Options		
		VSRC		

42. Engineering Change Order screen will appear..

Engineering Cl	hange Order						<u>? ×</u>
Modifications					Status		
Enable 🗸	Action	Affected Object		Affected Document	Check	Done	Message
✓	Add		In	🕮 Training.PobDoc			
✓	Add		In	🕮 Training.PobDoc			
✓	Add		In	🗱 Training.PobDoc			
✓	Add	- R1-2 to VCC	In	🗱 Training.PobDoc			
✓	Add		In	🗱 Training.PobDoc			
✓	Add	- R2-2 to VCC	In	🕮 Training.PobDoc			
✓	Add		In	IIII Training.PobDoc			
	Add	- R3-2 to VCC	In	🗱 Training.PobDoc			
	Add		In	时 Training.PobDoc			
	Add	- R4-2 to VCC	In	时 Training.PobDoc			
-	Add Component Class Members(9)						
 Image: A start of the start of	Add	📑 12Vpos to Training	In	III Training.PobDoc			
 ✓ 	Add	💷 C1 to Training	In	MBP Training.PobDoc			
 ✓ 	Add	💷 C2 to Training	In	III Training.PcbDoc			
✓	Add	💷 Q1 to Training	In	🗱 Training.PcbDoc			
	Add	🔒 Q2 to Training	In	🗱 Training.PcbDoc			
 Image: A start of the start of	Add	📵 R1 to Training	In	🗱 Training.PobDoc			
 Image: A start of the start of	Add	📑 R2 to Training	In	III Training.PobDoc			
 ✓ 	Add	💷 R3 to Training	In	MBP Training.PobDoc			
✓	Add	💷 R4 to Training	In	III Training.PcbDoc			
=	Add Rooms(1)						
	Add	Scope=InComponent	tí To	🖼 Training.PcbDoc			
Validate Chan	ges Execute Changes Eeport C	Changes					Close

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

Enginee	ring Ch	ange Order						? ×
Modifical	tions					Status		
Enat	ble 🗸 🗸	Action	Affected Object		Affected Document	Check	Done	Message
-		Add Components(8)						
	✓	Add	🔒 C1	To	🕮 Training.PcbDoc	3	9	
	◄	Add	🚺 C2	To	III Training.PcbDoc	2	2	
	✓	Add	🕘 Q1	To	III Training.PcbDoc	2	<i>.</i>	
	✓	Add	🔒 Q2	To	III Training.PcbDoc	2	<i>.</i>	
	◄	Add	间 R1	To	🕮 Training.PcbDoc	2	9	
	◄	Add	归 R2	To	🕮 Training.PcbDoc	9	9	
	✓	Add	归 R3	To	🕮 Training.PcbDoc	9	<i>.</i>	
	✓	Add	间 R4	To	🕮 Training.PcbDoc	2	a	
-		Add Nets(6)						
	✓	Add	🔁 GND	To	🕮 Training.PcbDoc	9	9	
	✓	Add	~ P1	To	III Training.PcbDoc	2	<i>-</i>	
	◄	Add	🔁 P2	To	🕮 Training.PcbDoc	2	9	
	◄	Add	~ P3	To	🕮 Training.PcbDoc	9	9	
	✓	Add	~ P4	To	🕮 Training.PcbDoc	9	9	
	✓	Add	🔁 VCC	To	🕮 Training.PcbDoc	9	ø –	
-		Add Component Classes(1)						
	✓	Add	🗀 Training	To	III Training.PcbDoc	2	9	
-		Add Rooms(1)						
	✓	Add	🌛 Room Training (Scope=InComponent)	CTo	🕮 Training.PcbDoc	9	9	
· · · · · ·		E E	1					
Validat	e Chang	ges Execute Changes <u>R</u> eport C	hanges 🔲 Only Show Errors					Close
-								

- 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



© PCB GRAPHTECH Pte Ltd. Singapore BW

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 <u>Place>>Polygon Pour (P, G)</u> 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 <u>T</u>ools>>Polygon Pours>>Shelve polygon. And after edit the track, you



can restore back your polygon by go to <u>T</u>ools>>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 <u>File>>Fabrication outputs>>Gerber Files</u> to create the Gerber file and Gerber Setup dialog will appear.

Gerber Setup		? ×
General Layers Drill Drawing	Apertures Advanced	
Specify the units and format to be u This controls the units (inches or mil decimal point.	sed in the output files. limeters), and the number of digits before and	l after the
Units	Format	
Inches	• 2: <u>3</u>	
C Millimeters	C 2: <u>4</u>	
	C 2: <u>5</u>	
The number format should be set to The 2:3 format has a 1 mil resolutior	suit the requirements of your Project. h, 2:4 has a 0.1 mil resolution, and 2:5 has a	0.01 mil
resolution. If you are using one of the higher re	solutions you should check that the PCB ma	nufacturer
supports that format. The 2:4 and 2:5 formats only need t	to be chosen if there are objects on a grid fin	er than 1
mil.		
		OK Cancel

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

Extension GTO GTO GTP GTS	Layer Name Top Overlay Top Paste	Plot Mir	Layer Name Mechanical 1	Plot
GTO GTP GTS	Top Overlay Top Paste		Mechanical 1	
GTP GTS	Top Paste		14 1 1 10	
GTS			Mechanical 2	
	Top Solder		Mechanical 3	
GTL	Top Layer		Mechanical 4	
GBL	Bottom Layer		Mechanical 5	
GBS	Bottom Solder		Mechanical 6	
GBP	Bottom Paste		Mechanical 7	
GBO	Bottom Overlay		Mechanical 8	
GKO	Keep-Out Layer		Mechanical 9	
GM1	Mechanical 1		Mechanical 10	
GM2	Mechanical 2		Mechanical 11	
GM3	Mechanical 3		Mechanical 12	
GM4	Mechanical 4		Mechanical 13	
GM5	Mechanical 5		Mechanical 14	
GM6	Mechanical 6		Mechanical 15	
GM7	Mechanical 7		Mechanical 16	
GM8	Mechanical 8			
GM9	Mechanical 9			
GM10	Mechanical 10			
GM11	Mechanical 11			

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.

		Deill Description Combinels
Plot all used layer pairs	I <u>M</u> irror plots	Drill Drawing Symbols
Top Layer-Bottom Layer		Graphic symbols
		O Size of hole string
		C <u>C</u> haracters
		Symbol size
		50mil
Pot all used layer pairs	I <u>M</u> irror plots	_
T Op Layer-Bottom Layer		_

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

Embedded apertures (RS274×) 🔽	Aperture	s List				
If the Embedded apertures option is enabled apertures will extomatically be created from the PCB each time you generate the output files using this GAM setup. If this option is not enabled the aperture list on the right is used. Use the buttons to create or load a suitable aperture list.	DCode	Shape	Usage	X Size	Y Sixe	Hole Size
 ✓ Generate relief shapes ✓ Elash pad shapes ✓ Flash all fills 	▲ <u>N</u> ew. <u>C</u> rea	E <u>d</u> it. te List From PC	<u>H</u> ename B Lgad !	. <u>Cle</u> ar Sa <u>v</u> e	De	lete

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

Gerber Setup	? ×
General Layers Drill Drawing Apertures (Advanced)	
Film Size Leading/Trailing Zeroes X (horizontal) 20000mil Y (vertical) 16000mil Border size 1000mil Plus 0.005mil Migus 0.005mil	
Batch Mode C ganate file per layer C ganetize layers Description D	
Other © G54 on aperture change © Generate DRC Rules export file (.RUL) © Use software arcs © Generate DRC Rules export file (.RUL)	
ОК	Cancel

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



File>>Fabrication 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.

C Drill Format –		
pecify the units	and format to be used in th	ne NC Drill output files.
his controls the lecimal point.	units (inches or millimeters) Units Inches	, and the number of digits before and after the Format • 2:3
	C <u>M</u> illimeters	C 2: <u>4</u> C 2: <u>5</u>
he number form as a 1 mil resolu :4 has a 0.1 mil igher resolutions hould check tha	at should be set to suit the tion, resolution, and 2:5 has a 0 you t the PCB manufacturer su	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the
he number form as a 1 mil resolu (4 has a 0.1 mil igher resolutions hould check tha nly need to be c there are holes eading/Trailing)	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions
he number form las a 1 mil resolu 24 has a 0.1 mil ligher resolutions hould check that nily need to be c there are holes eading/Trailing 2 C Keep leading	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes and trailing zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions C Reference to <u>a</u> bsolute origin
he number form as a 1 mil resolu igher resolutions hould check tha nly need to be c there are holes eading/Trailing <u>Keep leading</u> Suppress lea	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes and trailing zeroes ding zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions C Reference to gbsolute origin C Reference to relative origin
he number form as a 1 mil resolutions nould check tha hy need to be of there are holes sading/Trailing: <u>Keep leading</u> Suppress lea Suppress trai	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes and trailing zeroes ding zeroes ling zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions C Reference to <u>absolute origin</u> C Reference to relative origin
he number form as a 1 mil resolu gher resolutions nould check tha nly need to be a there are holes aading/Trailing Suppress lea Suppress lea Suppress trai	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes and trailing zeroes ding zeroes ling zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions Coordinate Positions Reference to absolute origin C Reference to relative origin
he number form as a 1 mil resolu 4 has a 0.1 mil giber resolutions nould check tha nould check that nould check that nould check that nould check that eading/Trailing : Keep leading Suppress lea Suppress lea Suppress trai ther	at should be set to suit the tion, you the PCB manufacturer su hosen on a grid finer than 1 mil. Zeroes and trailing zeroes ding zeroes ling zeroes	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions C Reference to <u>absolute</u> origin C Reference to relative origin
he number form as a 1 mil resolu 4 has a 0.1 mil igher resolutions nould check tha niy need to be o there are holes eading/Trailing i Suppress Jea Suppress Jea Suppress Jea Unter Definice che Generate se	at should be set to suit the tion, escolution, and 2:5 has a 0 you the PCB manufacturer su hosen on a gif finer than 1 mil. Zeroes and trailing zeroes ding zeroes ling zeroes ange location commands parate NC Drill files for plat	requirements of your design. The 2:3 format .01 mil resolution. If you are using one of the pports that format. The 2:4 and 2:5 formats Coordinate Positions Coordinate Positions Reference to absolute origin Coordinate Positions Coordinate Positions

Import Drill Data	? ×
Settings Start Units: 2.3 Leading Abs Inch	
Shape/Default Hole Size	
0.0320:0.0320 <u>I</u> ool Ta	ble
ОК С	ancel

- 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.

Name 🔺	- Date	emodified	[+]	Size +			0
						Ð	Ð
History	Project Outputs for Training	Aper.lib	Aper.lst	cam.ipc	CAMtastic1.Cam	CAMtastic2.Cam	doc092807.gbl
Ì			0				
job.drc	New Text Document.txt	PCB1.PcbDoc	PCB1.PcbDoc.htm	Training.PcbDoc	Training.PcbDoc.htm	Training,PrjPcb	Training.PrjPcbStructure
	<u>\</u>	A		A	A. A	-	
Training.SchDoc	Training.sdf	Training PCB ECO 2-10-2007 11-18	Training PCB ECO 2-10-2007 11-22	Training PCB ECO 2-10-2007 11-23	Training PCB ECO 2-10-2007 11-24	Training SCH ECO 27-9-2007 1-32	

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 <u>File>>Smart PDF (F, M)</u> and smart pdf setup dialog will display.



72. Smart pdf setup dialog will display. Click next to go to the next setup. © PCB GRAPHTECH Pte Ltd. Singapore BW

Smart PDF				×
Choose Export Target				1-11-
Smart PDF can export the currently viewed document or the entire project.				C ARE
Smart PDF can export the currently viewed document or documents in the current proje	ot.			
Current Project (Training.PrjPcb)				
C Current Document (Training.PcbDoc)				
Output File Name:				
C:\Documents and Settings\Administrator\Desktop\Training NYP\Training.pdf			6	
		_		
	<u>C</u> ancel	< <u>B</u> ack	<u>N</u> ext >	Einish

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

Smart PDF			×
Choose Project Files			1-11-
Select the files in the project to export from the list			
From the list below, select the files to export. Multiple files can be selected by Ctrl+Click	or Shift+Click.		
C:\Documents and Settings\Administrator\Desktop\Training NYP\Training.Po	bDoc		
C:\Documents and Settings\Administrator\Desktop\Training NYP\Training.Sc	hDoc		
	<u>C</u> ancel	< <u>B</u> ack	Next > <u>F</u> inish

74. Choose your project file then click next.

	gured here.					
Printouts & Layers	Printout Options					
lame	Тор	Bottom	Double Sided	Holes	Mirror	TT Fonts
— Top Loveray — Top Lover — Bottom Layer — Mechanical 1 — Keep-Out Layer — Multi-Layer						
C Specific Area Lower Left Corner Upper Right Corner	: : Omil : : Omil		Y: Omil Y: Omil		<u></u> c	ifine

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

Dristente V I suere		Include Comp	mente		Drinker & Orthony			
Printouts & Layers	Top	Bottom	Double Sided	Holes	Printout Uptions Holes Mirror TT Fonts			
Multilayer Composite Print								
Keep-Dut Layer Multi-Layer Area to Print Entire Sheet Specific Area Lower Left Corner X: Or Upper Right Corner X: Or	าปี าปี	Create Powe Create Mask Create Drill I Create Asse Create Comp Move Up Move Up Move Down Insert Printo Insert Layer Delete	r-Plane Set .Set Drawings mbly Drawings posite Drill Guide	_		fine		
references		Preferences						

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

Set		×					
o create a complet I remove all the cu	te final artw Irrent print s	ork set ettings					
/es No							
			•				
s							
B files can be configur	red here.						1
							17 1990 A
x Layers		Include Comp	onents		Printout Optic	ins	
	Тор	Bottom	Double Sided	Holes	Mirror	TT Fonts	
	✓		>				
	✓	◄	•				
rlay	✓	✓	⊻				•
					_	. 1	
werLeftCorner X:Ju	Jmil		Y: Jumi			efine	
per Right Corner X : [20100		Y: John				
			Cancel	(Bac	L Novi	<u></u>	Finish
	-Set o create a complei Il remove all the cu (es No S S S B files can be configur à Layers atlay wer Left Corner X : [oper Right Corner X : [-Set o create a complete final artw I remove all the current print s /es No S S S S S S S S S S S S S	Set x o create a complete final artwork set I remove all the current print settings /es No s S S S files can be configured here. A Layers Include Comp Top Bottom ✓ ✓ A stay wer Left Corner X: Omit oper Right Corner X: Omit	Set o create a complete final artwork set l'es No S S B files can be configured here. A Layers Include Components Top Bottom Double Sided V V V etlay V V V V V etlay V V V V V etlay V V V V V V etlay V V V V V V V etlay V V V V V V V V V V V V V V V V V V V	Set o create a complete final artwork set I remove all the current print settings /es No S B B B B B B B Cancel Cancel	S Cancel	Set X o create a complete final artwork set I remove all the current print settings (es No S B Cancel Cancel

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

Smart PDF			×
Additional PDF Settings Select the additional settings for the PDF.			
Zoom Use the slider to control the zoom level in Acrobat when jumping to components or nets. Far Additional Bookmark Image: Close Image: Close	Schematics Include: P No-ERC Markers P Parameter Sets PCB PCB Color C Greyscale C Monochrome	Color mode: © Color © Greyscale © Monochrome	
	Cancel	< Back Next >	Einish

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

Smart PDF		×
Structure Settings		
Select the structure to use for the P	DF.	1 VIII PR
		and the state of t
Structure		
If checked, physical designators wi physical sheets. A choice can also designators, net labels, ports, sheet	I be used in the exported PCB and schematic sheets will be expanded from logical sheets to be made on which variant to use and whether to display the expanded physical names of entries, sheet number and document number parameters.	
Use Physical Structure		
Variant	[None]	
Designators		
Net Labels		
Ports and Sheet Entries		
Sheet Number Parameter		
Document Number Parameter		
	<u>C</u> ancel < <u>B</u> ack <u>N</u> ext >	<u> </u>

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

Smart PDF				×
Final Steps Select the structure to use for the PDF.				
Once the export is complete, the default Acrobat Reader can be opened to view the	PDF.		and the second	
☑ <u>0</u> pen PDF file after export				
	Consert	- Paul	Process I Process	_
		< <u>B</u> ack		



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



© PCB GRAPHTECH Pte Ltd. Singapore BW

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 <u>File>>New>>Project>>Integrated Library</u>.



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

D <u>x</u> P	Eile	<u>V</u> iew Proje <u>c</u> t <u>W</u> indow <u>H</u> e	p		_		DXP File	⊻ie	ew Proje <u>c</u> t <u>W</u> indow <u>H</u> elp			
0		New •		Schematic			🗋 💕 🗶					
Projects	2	Open Ctrl+O	11	<u>P</u> CB			Projects		▼			
Works	3	Open Project	Ø	VHDL Document			Workspace1.	Dsn₩	/rk Vorkspace			
Integra		Open Design Workspace	Ø	Verilog Document			Integrated_Lib	orary1	LibPkg Project			
(De)		Save Project	C	⊆ Source Document			File View	O St	tructure Editor 💌 💽			
		Save Project As	ы 	C Header Document			🗆 🗃 Integr	凰	Compile Integrated Library Integrated_Library1.LibPkg	1		
		Save Design Workspace		Task Designed			NOL		Recompile Integrated Library Integrated_Library1.LibPkg			
		Save Design Workspace As		CAM Document					Add New to Project		<u>O</u> ther	Ctrl+N
		Save Ali		Output Job File				<u></u>	Add Existing to Project	2	Schemat	ic Library
	E	Smart PDF	đ	Database Link File					Save Project	2	PCB Libr	ary
		Import Wizard		Project >					Make Active Project		VHUL LIE	rary
		Recent Documents		Library 🕨	8	Schematic Library			Open Project Documents	₽ ₽	Taut Day	
		Recent Projects		Scrigt Files	2	PCB Library			Close Project		Text Do	umenc
		Recent Workspaces		Mixed-Signal Simulation $ ightarrow $	1	VHDL Library			Explore			
		Exit Alt+F4		Other +	5	PCB3D Library		63	Show Differences			
			į,	Design <u>W</u> orkspace	•	Database Library			Version Control			
			Т		1	SVN Database Library		1	Pariat Parlage			
									Project Packager SVN Database Library Maker			
									Project Options			

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>Place>>Pad</u> (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.



 Use line to draw component silkscreen, first step is change your current layer to Top Overlay, go to <u>Place>>Line (P, L)</u> or

click on this icon . To Draw ARC, Go to <u>Place>>ARC</u> (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>Edit>>Set Reference>>Pin 1 (E, F, P)</u>

3	<u>A</u> rc (Center)
0	Arc (Edg <u>e</u>)
$\boldsymbol{\sigma}$	Arc (Any Angle)
0	F <u>u</u> ll Circle
	Eil
2	Solid <u>R</u> egion
۲	Place Component Body (Mechanical Layers Only)
1	Line
Α	String
0	Pad
P	<u>V</u> ia
2	Polygon Pour Cutout
	Keepout

	P 6H I	Edit ⊻e ≦ To. 4	w Projeg	t Blace Ioo	ls Beports Window	w ttelp + v≴ ≠n ⇔ ≡ •	i∕ o ◊• A	× 0000	- 14		
PCB Libr	ary			- Ø ×	Peh iht Peh ih				- 454		
Mask				•	- reconnect						
	nniu 🗸 🗸	Clear	🔾 Macoliu	1							
E Mai	4 🔽 Sala	et 🔽 700	vo 🔽 Class	j Evistina							
- <u>-</u>	ni je golo		ante gou	Canady							
Name	nents	/ Pa	de	Primitives							
PCBCO	IMPONEN	T_1 2		6							
	•										
	T										
						<u> </u>					
				_							
Do	uble cli	ck to cl	hange ni	ame							
<u> </u>				_					<u> </u>		
Campo	onet Dúnit	ú						2			
Type	Name /	X-Size	Y-Size	Laver							
Arc		10mil		TopOverlay							
Track		10mil		TopOverlay							
Track		10mil		TopOverlay							
Track		10mi	00.1	TopUverlay							
Pad	1	60mil	60mil	MultiLayer MultiLayer					/		
Fau	2	CONTRACT	ounn	MOREGYO							
	_										
	1	+									
		7									
		1_	<u></u>								
		•	•								
		1									
	Desinche	/ Alucian		1111111			Masharinal 1	Quarter (Kass Oct.)	and I Makil and	1	
4 . 2007	higetts	A manife	410-1	N PU		Layer A = contom Layer A	reconcel 1	ovenay All Keepourt	ayer A Multi-Layer	/	

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

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



Resistor



© PCB GRAPHTECH Pte Ltd. Singapore BW

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

			1				
,							
Top Layer/							
Location			Size and	Shape			
x	Omil] 💽 Simp	le O Top	p-Middle-Bot	om 🔹 🔿 Full Stat	ck
Y	Omil]	X-Size	Y-Size	Shane	Corner Badius (%)
Rotation	0.000			80mil	120mil	Rectangular	▼ 50%
Hole Information -							
Hole Information - Hole Size	Omil		1				n.e.v. [
Hole Information - Hole Size I <u>R</u> ound	Omil		1			Edit Full Pad Layer	Definition
Hole Information - Hole Size © <u>R</u> ound © Square	Omil					Edit Full Pad Layer	Definition
Hole Information - Hole Size © Bound © Square © Slot	Omil		Paste Ma	ask Expansio	on —	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Round © Square © Slot	Omil		Paste Ma © Exp	ask Expansio bansion valu	on	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Bound © Square © Slot	Omil		Paste Ma © Exp	ask Expansio pansion valu ecify expans	on e from rules	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Bound © Square © Slot Properties Designator			Paste Ma © Exp © Spiller M	ask Expansio bansion valu ecify expans ask Exnansi	on ue from rules sion value	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Bound © Square © Slot Properties Designator Layer	Omil		Paste Ma © Exp © Spi Solder M	ask Expansio pansion valu ecify expansi ask Expansion valu	on ue from rules sion value ions ue from rules	Edit Full Pad Layer Omil	Definition
Hole Information - Hole Size © Bound © Square © Stot Properties Designator Layer Net	Omil 1 Top Layer No Net	×	Paste Ma © Exy Solder M © Sp	ask Expansio pansion valu ecify expans ask Expansio pansion valu	on ue from rules sion value ions ue from rules	Edit Full Red Layer	Definition
Hole Information - Hole Size © Bound © Square © Slot Properties Designator Layer Net Electrical Typ	Dmil 1 Top Layer No Net Ie Load	×	Paste Ma © Exp Solder M © Exi © Sp	ask Expansii oansion valu ecify expans ask Expansi oansion valu ecify expans	on ue from rules sion value ions ue from rules sion value	Edit Full Pad Layer Omil 4mil	Definition
Hole Information Hole Size Square Slot Properties Designator Layer Net Electrical Typ Testpoint	[Omil] [1] Top Layer No Net Ie Load Top	V V V V	Paste Ma © Exp Solder M © Exp © Sp © Sp © Fo	ask Expansio pansion valu ecify expans ask Expansi pansion valu ecify expans rce compl	on ue from rules sion value ions ue from rules sion value ete tenting	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Round © Square © Slot Properties Designator Layer Net Electrical Typ Testpoint Plated	Dmil Top Layer No Net Load Top	▼ ▼ ▼ Bottom	Paste Ma © Exp © Sp Solder M © Exp © Sp □ Fo □ Fo	ask Expansion valu ecify expansi ask Expansi carsion valu ecify expansi rce compl rce compl	on le from rules sion value le from rules sion value ete tenting ete tenting	Edit Full Pad Layer	Definition
Hole Information - Hole Size © Eound © Square © Slot Properties Designator Layer Net Electrical Typ Testpoint Plated Locked	Drnil Top Layer No Net Load Top	¥ ¥ ▼ Bottom	Paste Ma © Exp © Sp Solder M © Exp © Sp □ Fo □ Fo	ask Expansion valu ecify expansi ask Expansi opansion valu ecify expans rce compl rce compl	on ue from rules sion value ue from rules sion value ete tenting ete tenting	Omil 4mil on top on bottom	Definition
Hole Information - Hole Size © Bound © Square © Slot Properties Designator Layer Net Electrical Typ Testpoint Plated Locked	Ornil Top Layer No Net Load ▼ Top	▼ ▼ ▼ ■ Bottom	Paste Ma © Exi © Spi Solder M © Exi © Spi Fo Fo	ask Expansion valu ecify expansi ask Expansi oansion valu ecify expans rce compl rce compl	on ue from rules sion value ue from rules sion value ete tenting ete tenting	Omil Amil on top	Definition

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 <u>T</u>ools>><u>I</u>PC Footprint Wizard (T, I) and Wizard dialog will display. Click Next to continue your setting. Choose SOP and click next

otprint Wizard			
elect Compon	rent Type		· August
You can chose he	ere the family of components that you wish to create.		
Component Type:	8		
Name	Description	Included Packages	 The selected component is SOP.
BGA	Ball Grid Array	BGA, CGA	This will allow you to generate SUP, SUP Exposed
BQFP	Bumpered Quad Flat Pack	BQFP	Fau paukages.
CFP	Ceramic Dual Flat Pack - Trimmed and formed Gullwing Leads	CFP	
CHIP	Chip Components, 2-Pins	Capacitor, Inductor, Resistor	
CQFP	Ceramic Quad Flat Pack - Trimmed and formed Gullwing Leads	CQFP	
DPAK	Transistor Outline	DPAK	
LCC	Leadless Chip Carrier	LCC	
MELF	MELF Components, 2-Pins	Diode, Resistor	
MOLDED	Molded Components, 2-Pins	Capacitor, Inductor, Diode	
PLCC	Plastic Leaded Chip Carrier, Square - J Leads	PLCC	
PQFP	Plastic Quad Flat Pack	PQFP, PQFP Exposed Pad	· ·
QFN	Quad Flat Pack No-Lead	QFN, LLP	
QFN-2ROW	Quad Flat Pack No-Lead, 2 Rows, Square	Double Row QFN	
SOIC	Small Outline Integrated Package, 1.27mm Pitch - Gullwing Leads	SOIC, SOIC Exposed Pad	
SOJ	Small Outline Package - J Leads	SOJ	
SOP	Small Outline Package - Gullwing Leads	SOP, SOP Exposed Pad	
0.00 M 4 4 0 10 4 0	Small Outline Transistor	SOT143, SOT343	
501143/343		0.0 7.000	

Physical Dimensions inches (millimeters) unless otherwise noted



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

OP Package Dimension ter the required package valu	ns es.			1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1
Dverall Dimensions				Preview
Width Range (H)	Minimum Maximum	5.791mm 6.198mm	Top View	
Maximum Height (A) Minimum standoff heigh	it (A1)	1.753mm 0.254mm	e e t	
Maximum body Width (B	E)	3.988mm		
Maximum body length (I	D)	5.004mm	End View ← _E →	
in Information				
Number of pins		8		
Lead Width Range (B)	Minimum Maximum	0.356mm 0.508mm		
Lead Length Range (L)	Minimum Maximum	0.406mm 1.27mm		
Pitch (e)		1.27mm		
				Cancel (Back Next) Finis

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

IPC Footprint Wizard		×
SOP Package Heel Spacing Enter the heel spacing values.		- And - And -
The minimum heel spacing is calculated by subt	racting twice the Maximum Lead Length Range from the Minimum Body Width Range.	Preview
The maximum heel spacing is calculated by add leads to the minimum heel spacing.	ting the tolerance on the inner distance between the heels of the opposing rows of	
✓ Use calculated values	End View	
S Minimum 3.251mm		
S Mærinum 4.54mm		
		Cancel < Back Next> Einish

- 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.

IPC Footprint Wizard	×
SOP Solder Fillets Enter the required fillet values.	in the second
Solder fillet refers to the shape of the solder joint between the component lead and the PCB pad. Adequate fillet is required to ensure both the strength and reliability of the solder joint. A solder joint may be described by three fillet: the head and How PCE. based on industry empirical knowledge and reliability testing. These values are displayed below, however they may be adjusted to sult specific conditions. Use default values Board density Level Level B - Medium density Head Fillet (H Min) 0.31mm Head Fillet (H Min) 0.31mm Side Fillet (S Min) 0.02mm Head Fillet (J Min) 0.30mm Head Fillet (J Min) 0.30mm Head Fillet (J Min) 0.31mm Side Fillet (S Min) 0.02mm	Preview
Cancel	< <u>B</u> ack <u>N</u> ext> <u>F</u> inish

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

PC Footprint Wizard		×
SOP Component Tolerances Enter the required component tolerance values.		1000
Component manufacturers usually specify the minimum and maximum value for each package d ranget are derived by subtracting the minimum value from the maximum. These ranges may be suppliers.	fimension. Component tolerance adjusted based upon experience from	Preview
☑ Use calculated component tolerances		
Tolerance on the overall width of the component, including leads	0.407mm	
Tolerance on the inner distance between the heels of the opposing rows of leads	1.288mm	
Tolerance on the width of the component leads	0.152mm	
	<u>_</u>	ancel <back next=""> Einish</back>

Footprint Wizard	
SOP IPC Tolerances citer the required tolerance values.	- And
IPC specifies certain tolerances for a number of standardized surface-mount package types. These tolerances are assumed by 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 IP footprints.	this wizard in order to Preview
☑ Use Default Values	
Fabrication Tolerance Assumption	
This allowance may be adjusted according to the accuracy of the PCB fabricator to reproduce the PCB tooppint dimensions on the printed board.	
This allowance may be adjusted according to the accuracy of the assembler to center the component on the PCB footprint.	
Courtyard Excess	
The Courtyard of a PCB footpint defines the area required for electrical and mechanical clearance of both the comparent and its tootpint. The dimensions of the courtyard boundary are calculated by the addition of a courtyard excess to the maximum dimensions of the contrand comparent and tootpint. The value of the courtyard excess differs accounding to the density level of the printed circuit board.	m
	Cancel (Rack Next) Finish

ootprint Wizard		
OP Footprint Dimensions re footprint dimensions can now be inferred from the package dimensions. ou can review and modify them here.		- And
The footprint has 8 pads and a pitch (P) of 1.27mm. You can modify here the calc	ulated dimensions of the footprint.	Preview
Use calculated footprint values		
Pad Dimensions X 0.6mm Y 2.2mm Pad Spacing C 4.8mm	Top View ↓ P + + ↓ + ↓ + ↓ + ↓ + ↓ + ↓ + ↓ + ↓ ↓	
Pad Shape © Rounded © Rectangular		
		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.	in the second
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	Proview
Top View Top View Top View Top View Top View Top View Top View Top View Top View	
<u>_</u> Cancel	< <u>B</u> ack <u>Next></u> Einish

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.		- And - And -
Choose here whether to add Courtyard and Assembly information to the component drawing. For calculated dimensions values, or erfer the values manually. You can also choose the mechanics and the used line thickness. Finally you can decide whether or not to add a component body, wi corresponding to the package dimensions.	reach of these, you can use either the IPC al layer on which the drawing will be added, hich contains the volumetric information	Preview
✓ Use calculated values V1 7.5mm V2 5.5mm ✓ Add Assembly Information ✓ Use calculated values A 3.99mm B 5mm ✓ Add Component Body Information ✓ Use calculated values ✓ Add Component Body Information ✓ Use calculated values Vath 0.99mm Length 5mm	Top View \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow	
	Cancel	< <u>B</u> ack Next> Einish

IPC Footprint Wiza	rd	×
SOP Footprin The footprint valu You can review a	t Description es can now be inferred from the package dimensions. nd modify them here.	in the second
🔽 Use sugges	red values	
Name	SOP127P600X175-8N]
Description	SDP, 8-Leads, Body 5.0x4.0mm (max), Pitch 1.27mm, IPC Medium Density	
	Cancel	< <u>B</u> ack <u>N</u> ext> <u>F</u> inish

IPC Footprint Wizard		×
Footprint Destination Select where to store the	n fnished footprint.	and the second
C Existing PobLib File C New PobLib File C Current PobLib File	C-Wsers/Administrator/Desktop/My Work/My Training Manual/Example For Training Manual/Library Example for training	menuel/Training PobLib
	Cancel	< <u>B</u> ack <u>N</u> ext> <u>F</u> inish



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

rary •			Home 2 Training Pot	~ ~ Ⅲ • / ● 36.6•]		N ⊘ ⊒ ki		[C (ose	
upply isk 🔽 S	elect 🔽 Zoom 🖓 (Inity Dear Existing					_		
anents		1			- Balancessan	Representation (Second second			
100	/ Pads	Primitives							
TOR SH	AD 2	7			1000				
7P60 0 -	(175-8N 8	26							
					-				
						8			
ert Pr	initives								
lame /	X-Size Y-Size	Layer							
	0.2mm 0.2mm	TopOverlay TopOverlay							
	0.2mm	Top0 verlay							
	0.2mm	TopOverlay,				8			
	0.6mm 2.2mm	TopLayer TopLayer							
	0.6mm 2.2mm	TopLayer			1000 ACC				
	0.6mm 2.2mm	TopLayer							
	0.6mm 2.2mm	TopLayer							
	Utenn 2.2mm	TopLayer							
	0.0000 2.2000	TupLayer							

Schematic Library

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

Display Name	1 Visible	
Designator	1 Visible	1
Electrical Type	Passive 💌	
Description		
Hide	Connect To	
Part Number	0	
Symbols		Graphical
Inside	No Symbol	Location X -5 Y 10
Inside Edge	No Symbol 💌	Length 30
Outside Edge	No Symbol 💌	Orientation 180 Degrees
Outside	No Symbol 💌	Color
VHDL Parameters		1
Default Value		
Formal Type		
Unique Id	IAKBMOYD Beset	

DXP Elle Edit View Project Place Tools	Reports Window	r <u>H</u> elp V lan ov I Mode ≠	اعدامها	a . M . III . G		C:\Users\Administrator\Desktop\My W 🕶 😋
SCH Library	Home 2 Tra	ining.PobLib	ng.SchLib			
Components A Description						
				1	2	
Place Add Delete Edit						
Palayes						
Add Delete Edit Pins Name Type 0 1 1 Passive 0 2 2 Passive						
	•					Mask Level 0
Add Delete Edit	Model	∕∆ Туре	Location	Description		There is no consistent so calcula
Add Delete Edit						i nero is no preview dvaliaue
Projects / Navigator / SCH Library / SCH X:10 Y:0 Grid:10	Add Footprint	* <u>R</u> emove	<u>E</u> dit			 System Design Compiler Help SCH Instr

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)

Properties Parameters for Capacitor Delayat C? IF Visible Locked Visible Name / Value Type Comment Capacitor 100µF IF IF Visible Name / Value Type	
Default Designator C? IF Visible Locked Visible Name / Value Type Comment Capacitor 100 JF IF Visible Name / Value I Type	
Comment Capacitor 100uF 🔽 🗹 Visible	
< > >>> Part 1/1 □ Locked	
Description	
Type Standard	
Library Link	
Symbol Reference Capacito	
Braphical Edit Add as Bule	
Node Normal V Lock Pins Models for Capacitor	
Show All Pins On Sheet (Even if Hidden) Name Tune Description	
Local Colors	
	_
Ağd Regove Edi	
Edit Pins	el



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

Model		3
	Model Name Dis Mar	1
Name		4
Description	Footprint not found	
PCB Library		
Any		
C Library name		
C Library path	Choose	1
C Use footprint fr	om component library *	_
Selected Footprint		
М	odel Name not found in project libraries or installed libraries	
Found in:		

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

Browse Libraries		? ×
Libraries Training.PcbLib		Find
<u>M</u> ask	•	
Name 🛆 Libra	ry Description	
Capacitor Train Resistor SMD Train	ing.PcbLib 100uF ing.PcbLib	
SOP127P600X175 Train	ing.PcbLib SOP, 8-Leads,	
2 itoma		
o items		
		OK Cancel

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

Model			?
Footprint Model			
Name		Browse	<u>P</u> in Map
Description	100uF		
PCB Library			
 Any 			
C Library name			
🔿 Library path			Choose
C Use footprint fro	m component library *		
	+		
	•••		
Found in: CALVI	brary Example for training manual\Training.Po	:bLib	

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 Elle Edit Yiew Project Place Tools	Beports Window Help Call □ + X W ⊃ ⊂ I Mode • ♀ = > > 3 • W • Ⅲ • (a)	C:\Users\Administrator\Desktop\My W 🚽 📀
SCH Library ▼ Ø X	Training Poblub	
Components A Description Components A Description		
Place Add Delete Edit	1	2
Add Delete Edit Pins Name Type Resistor S 		
	<u>ــــــــــــــــــــــــــــــــــــ</u>	Mask Level C
Add Delete Edit Model / Type Description Re Footprint	Model ∧ Type Location Description ∰ Resistor SMD Foolprint	2
Projects / Navigator / SCH Library / SCH	Add Footprint + Remove Edit	
X:-65 Y:20 Grid:5		<u>System</u> <u>Design</u> Compiler <u>H</u> elp S <u>C</u> H Instru

Library Component	Properties							? ×
_ <u>P</u> roperties					Daramators for P	Pesister		
Default Designator	R? Visible	Locked	Visible	Name	Parameters for P △ Value	resistor	Туре	
Comment	100K 💌 Visible							
	<< < >> >>> Part 1/1	Locked						
Description								
Туре	Standard	•						
_ Library Link								
Symbol Reference	Resistor							
<u>G</u> raphical			<u>A</u> dd	Remo <u>v</u> e	Edit Ac	ld as <u>R</u> ule		
Mode	Normal 🗾 🗹 Lock Pin	s			Models for Be	veistor		
	🔲 Show All Pins On Sheet (Even if Hidden)		Name	Туре	∇ Descripti	on		
	Local Colors		Resistor SM	4D 🔻 Footprin	nt .			
			A <u>d</u> d	▼ Remove	Ediţ			
Edit Pins						_ F	ОК	Cancel
Edici jiio						L		Cancer

DXP Elle Edit View Project Place Iools	Reports Window Help					C:\Users\Administrator\Desktop\My W 🔹 📀
🗈 😂 🔙 🌧 这 🗶 🔍 😞 🔺 🖻	🖄 🗀 🕂 💥 🗙 🖌 🖓 🗠 🕴 Mode •	4 - > > B -	👱 • 🏢 • 🔯			
SCH Library VX	Training.PobLib					
Components / Description						
Pisce Add Delete Edit Alare: A Add Delete Edit Pirc Name Type SOP127P. -4 1 SDA Pasive 1 -4 2 SCL Pasive 2		$ \frac{1}{2} \overline{3} \overline{4} $	SDA SCL OS GND	VCC A0 A1 A2	8 7 6 5	
-4 GND Passive 4						
	1					
			10 IV			Mask Level C
Add Delete Edit Add Delete Edit Add Delete Edit Add Delete Edit	Model / Lype	Location	UPECIPION SOP: 8-Leads, Body 5.0x4	. Omm (max). Pitch 1.27mm, IPC	: Međum Densty	
×40Y:20 Grid:10						System Design Compiler Help SCH Instru

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)

Library Component	Properties								<u>1 A</u>	
Properties						Parameters for I	H 75 COD			
Default Designator	U?	Visible	Locked	Visible	Name	Parameters for L	.m 75 50P	Туре		
Comment	LM 75 SOP	Visible								
	$\langle \langle \rangle \rangle$	Part 1/1	Locked							
Description										
Туре	Standard		•							
Library Link										
Symbol Reference	LM 75 SOP									
<u>G</u> raphical				Add	Remoye	Edt	Add as <u>R</u> ule			
Mode	Normal V Lock Pins									
				Name	Name Tune V Description					
	Local Colors			SOP127F	600×175-8N 💌 Footpri	nt SOP, 8-	Leads, Body 5.0x4.0	0mm (max), Pitch	1.27mm, IPC	
				A <u>d</u> d	Hemove	Ediţ				
E dit Pjns								OK	Cancel	

SOP-8 and Mini SOP-8



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

Proj	e <u>c</u> t	Place	<u>T</u> ools	<u>R</u> eports	<u>W</u> indow	<u>H</u> elp				
1	Compile Document Training.SchLib									
	Compile Integrated Library Integrated_Library1.LibPkg									
1	Recompile Integrated Library Integrated_Library1.LibPkg									
	Design Workspace									
	Add New to Project									
Ľ	Add Existing to Project									
5	Remove from Project									
	Pro	ject Docu	uments			Ctrl+Alt+O				
	Close Project Documents Close Project									
	Show Differences									
(1)	Show Physical Differences									
Â	V <u>e</u> r	sion Con	trol)	•			
	Loc	al His <u>t</u> ory	/			,	•			
Ţ	Pro	ject Pack	ager							
۲	FPG	5A Works	pace Ma	p						
	Pro	ject <u>O</u> pti	ons							

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

