

Printed Circuit Board Design

Manual (Basics)

Tuan Huynh University of Tulsa Electrical Engineering 2004 -2006

TABLE OF CONTENTS

Getting Started2
Schematic
Placing the Components4
Wiring the Components5
Adding Power Ports6
Editing Component Properties7
Changing the Footprints7
Numbering the Components8
°CB14
Board Wizard14
Updating Footprints on the Board21
Room Definition24
Routing the Board25
Design Rule Check27
• View the Board in 3-D29

• To start Protel, go to Programs -> Tools -> DXP



• Next in Protel, to make a new PCB board, go to New -> PCB Project



• You should see your PCB Project file in the left hand explorer window.



- First you want to draw your schematic of your circuit.
- Go to File -> New -> Schematic
- Now in the left hand window, you should see **Sheet1.SchDoc*** in your project. If it is not in your project drag that file into your project.
- Next, to draw your circuit, you need to click on the **Sheet1.SchDoc** file and you should see the schematic sheet in the main widow.



In the toolbar up top, clicking on the Place Part symbol will open the Place Part window.

 Design Explorer DXP - Sheet 1.5chDoc *

Pesigir Enplore		icceriser					
D <u>X</u> P <u>File</u> <u>E</u>	dit <u>V</u> iew	Proje <u>c</u> t	Place	<u>D</u> esign	<u>T</u> ools	<u>R</u> eports	<u>W</u> indow
0 🖻 🖪 🚭) 🖪 🔎	ତ୍ର 🔍	<u>ta [d</u>	x7 %	Þ C	e []]	* +
🛯 🗳 😤 🗖	S 🔁	🔔 🗈	ı 🅸	<u> </u> ≈1	⊨ ⊾ ⊻	let) 🛨 💶	> 🖾 🛛
Projects	_	-	•	❷ × [Sheet1	l.SchDoc	Place Part
ProjectGroup1.Prj	Grp		Gro	up			
PCB Project1.PrjP	св		 Proj 	ect			
Discrete Schemer Discrete Disc	ect1.PrjPt atic Sheets	CB *	B				
🔚 She	et1.SchDoo	°	B				
Plac	ce Part				2	2 🗵	
						_	
	Lib Ref	Res1			•]	
	<u>D</u> esignator	R?]	
	<u>C</u> omment	Res1]	
	<u>F</u> ootprint	AXIAL-0.3			*]	
				ОК	Cance	el	

- Click on the **limit** to get to the **Browse Libraries** window.
- In this window, find the part that you need, click **OK** in the **Browse Libraries** window, then **OK** in the **Place Part** window.
- The component selected should be on the cursor.
- Continue this process for all the parts for the circuit.

Note: When you have selected a component to move or placing a component, you can press the **SPACEBAR** to rotate the component.

Browse Libraries			? 🛛
Libraries Miscellaneous Devi	ces.IntLib		Find
<u>M</u> ask			
Mask Component Name 2N3904 2N3904 2N3906 2N3906 2ADC-8 Antenna Battery Bell Bridge1 Bridge2 Bridge2 Buzzer Cap Cap Feed Cap Pol1 Cap Pol2 Cap Pol3 Cap Semi Cap Semi Cap Var Cap2 Cap2 Cacax D Schottky D Schottk		Q?	3
Model Name Type BCY-W3/D4.7 Footprin 2N3904 Signal I 2N3904 Simulat	Zener Diode		1
_		OK	Cancel

• To make connections between components, click on the **Wire** symbol , now you can draw wires.

When you see a **Red X** when you move your cursor over a component, it means it's an anchor point for the wire.

• When you complete a connection, pressing **ESC** or **Right-Click** will finish the line that you make, and you can continue draw other connections.





- To add power and ground points on your circuit, click on the Power Port symbol
- After placing a **Power Port**, you can double click on the component to get the **Power Port** window.
- In this window, you change the **Net** in the **Properties** section to represent power or ground (e.g. Vcc, +9V, +5V, GND)

🚾 Design Exp	olorer DXP -	Sheet1.SchDo)C *			
D <u>X</u> P <u>F</u> ile	<u>E</u> dit <u>V</u> iev	v Proje <u>c</u> t <u>F</u>	elace <u>D</u> esign	<u>T</u> ools	<u>R</u> eports	<u>W</u> indow
l 🗅 🖻 🔛	🖨 🖪 🖌	ð 🖸 🤤	b: 🔨 🛛	4 B B	8 []]	≫₀ ↔ ►
🛯 🕒 🖄 🐿	🛃 🖾 🖻	b 🚅 🗈 🛛	ج 🏖 🕯	7⊨ K ⊉	leti 🏣 🕫	≻ :□□: 🖸
Projects			▼ Ø ×	🔚 Sheeti	I.SchDoc × Place	e Power Port
ProjectGroup	1.PrjGrp		Group			
PCB Project1	.PrjPCB	•	Project			
E PCB	Project1.Prj hematic Shee Sheet1.SchD	PCB * ts loc *				
	Power Port				? 🗙	
	Color	NE	Sty Lo	le Bar cation X 440 Y 250		
	Properties Net VCC					

• Now to change the **Value** of a component, you can double click the **Value** next to a component or you can double click the component itself to bring up a **Component Properties** window to change the value of the component.

Co	omponent l	Properties				? 🛛
ſ	Properties -				Devenuelous (es D.2. Dev.1	
	Designator	B2 Visible	Visible	Name /	Value	Tune
	Designator			Class I	Passive	STRING
	Comment	Res1 Visible		Class II	Resistor	STRING
		Don't Annotate Component		Manufacturer	Generic Components	STRING
		C C Part 1/1		Published	8-Jun-2000	STRING
				Publisher	Altium Limited	STRING
	Library Ref	Res1		Revision	July-2002: Re-released for DXF	STRING
	Library	Miscellaneous Devices.IntLib	_	Value	<u>10K</u>	STRING
	Description	Besistor				
	Description					
	Unique Id	HEKUULHP Reset				
	Sub-Design		<u> </u>			
	_		<u>A</u> dd	Hemove	Edit Add as <u>R</u> ule	J
	Туре	Standard			Models for B2 - Bes1	
	Graphical –		Name	Type	∇ Description	
	<u></u>		RESISTOR	Simulation	Resistor	
	$Location\ X$	200 Y 360	Res1	Signal Inte	grity	
	Orientation	90 Degrees 🔻 🗌 Mirrored	AXIAL-0.3	 Footprint 		
	Mode	Normal				
	Mode	Chan All Dire On Chank (Even & Hidden)				
		Show All Fins On Sheet (Even if Hidden)				
		Local Colors M Lock Pins				
					F 12	
l				Hemove		
C						
l	Edit Pin	S				OK Cancel

- The capacitors that were placed on the schematic do not have the correct footprint.
- Go to a capacitor on your schematic and double click on it to bring up the **Component Properties** window.
- Click to the left of the **Footprint** text in the bottom right of the window and then click **Edit**.
- This should bring up the **PCB Model** window.
- In this window you will see the footprint at the bottom. You will notice red pads on the model, which means this footprint is requiring a surface-mount component of which we are not using in this example.
- In the PCB Library window, select Any.
- Then click **Browse**.
- The Browse Libraries window should show up.
- Find **RAD-0.3** in the list.
- Click OK in the Browse Libraries window, then OK in the PCB Library window.

_ Footprint	Model							
Name	C32	25-1210			Brov	wse	<u>P</u> in Map	
Descripti	on							511
C PCB Libra	ary —							
💿 Any								
🔾 🔾 Libra	ary name							
🔘 🔘 Libra	ary path						<u>C</u> hoose	
🔾 🔾 Use	footprint from inte	egrated library	Miscellan	eous Devi	ces.IntLib			
C Selected	Footprint							
		1		2				
			-	_				
Found in	: C:\Program File	es\Altium\Libi	rary\Misce	llaneous D	evices.In	tLib		
						014		
						ОК	Car	ncel
se Libraries						ОК	Car	ncel
se Libraries	18 Devices.IntLib)				ОК	Car	
se Libraries mes Miscellaneou	ıs Devices.IntLib)				ОК	Car	
se Libraries aries Miscellaneou k	ıs Devices.IntLib)	T			ОК	Car	
se Libraries aries Miscellaneou k	IS Devices.IntLib		•			ОК	Car	ncel
se Libraries aries Miscellaneou k lel Name 1N2 1N-W2/E2.8	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I	o Connector Devices P(V			OK	Car	ncel
se Libraries aries Miscellaneou k lel Name A IN2 IN2 IN2 OLAR0.8	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Y			OK	Car	rcel
se Libraries aries Miscellaneou k	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(V			OK	Car	ncel
se Libraries aries Miscellaneou k	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(×			OK	Car	ncel
se Libraries aries Miscellaneou k	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Y			OK	Car	rcel
se Libraries aries Miscellaneou k	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(0	OK	Car	rcel
se Libraries aries Miscellaneou k lel Name A PIN2 PIN-W2/E2.8 POLAR0.8 P2012-0805 PAD-0.1 PAD-0.1 PAD-0.2 PAD-0.3 PAD-0.4 PB5-10.5	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(0	OK	Car	rcel
se Libraries aries Miscellaneou k VIN2 VIN	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(1	OK	Car	rcel
se Libraries aries Miscellaneou k	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Single In-Line v	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Mevices P(M			0	OK	Car	rcel
se Libraries aries Miscellaneou k lel Name A PIN2 PIN-W2/E2.8 POLAR0.8 R2012-0805 RAD-0.1 RAD-0.2 RAD-0.1 RAD-0.2 RAD-0.3 RAD-0.4 R5-10.5 RFM-F3/B1.5 FFM-F3/P1.5 FFM-F3/P2.3 FFM-F3/P2.3	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Single In-Line v Single In-Line v	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Mith Mounti With Mounti				OK	Car	ncel ▼
Se Libraries aries Miscellaneou k lel Name PIN2 PIN-W2/E2.8 POLAR0.8 POLAPOL8 POLAP	IS Devices.IntLib Source Library Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Miscellaneous I Single In-Line v Single In-Line v Vertical, Single	Connector Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Devices P(Mounti Noth Mounti Row, Flan				OK	Car	rcel

PCB Model		? 🛛
Footprint Model]
Name	RAD-0.3	Pin Map
Description		
- PCB Library		
 Any 		
🔘 Library name	Miscellaneous Devices.IntLib	
🔘 Library path		<u>C</u> hoose
🔘 Use footprint fro	om integrated library Miscellaneous Devices.IntLib	
Selected Footprint		
	1 2	
Found in: C:\Progr	am Files/Altium/Library/Miscellaneous Devices.IntLib	
	ОК	Cancel

- If you look in the left hand window, the files in there are colored **Red**, which means the files are not yet saved.
- To save go to File -> Save All.
- By going through this method of **Save All** instead of just **Save As** or **Save**, this allows you to save everything including your **Schematic**, **Project File**, and **Project Group**.
- It will bring up **Save As** window(s) for you to name and save in a folder in which you will be able to locate your files.

D	🚧 Design Explorer DXP - Sheet1.SchDoc *											
	Σ	D <u>X</u> P	Eile	<u>E</u> dit	<u>V</u> iew	Pro	je <u>c</u> t	<u>P</u> la	асе	<u>D</u> esi	gn	<u>T</u> ools
Ī	Ď	B		<i>a</i>	Ð	P	Q.	o l	<u>ئم</u>	X	Æ	ēð 🕻
≣	ß	É	😤	🖻 🖪		⊵		ø	2		= 7	= 1
P	roje	ects							•	& ×		Shee
	Pr	oject0	âroup1	.PrjGrp					Gro	up		
	PC	CB Pro	oject1.	PrjPCB				•	Proj	ect		
		- 6	PCB F Scł	Project1 nematic S Sheet1.S	.PrjP iheets ichDo	CB * c *			E) E)			





- After you have saved all your files.
- The next step is to number your components (e.g. R1, R2, R3,...)
- If you notice, the components are designated as **R?**, **C?**,... By Annotating the schematic, this will change all the "?" into numbers (in order)
- To do this, go to **Tools** -> **Annotate**
- An **Annotate** window should show up.
- In the top left of the window, you can select how the program will number the components placed on the schematic.
- Click on the **Update Changes List** button at the bottom right, it should tell you the number of changes it has detected. Click **OK**.
- Then, click on the Accept Changes [Create ECO] button at the bottom right.
- An Engineering Change Order window should show up.
- Click on the Validate Changes button at the bottom left.
- Then, click on the **Execute Changes** button at the bottom left.
- After you have clicked these two buttons, you should see all green check marks for all the components.

🐙 Design Explorer DXP - \\FILER\TUAN-HUYNH\PCB	Board	ls\FM Radio Tx.SCHDOC
DXP File Edit View Project Place Design	<u>I</u> oo	ls <u>R</u> eports <u>W</u> indow <u>H</u> elp
-> 🛪 ٹم لگ 🕰 🔍 🔍 🔊 🚭 🔲 ڬ	-	Find Component
L 🛎 📽 🕞 🕞 🔂 L 🗈 🕼 🖧 🔍 🔍 –	₽ ₽	Up/Down <u>H</u> ierarchy
Projects $\checkmark & \times & \blacksquare$		Pa <u>r</u> ameter Manager
ProjectGroup1 PriGro		Update From Libraries
		Update From <u>D</u> atabase
FM Radio Tx PCB Project.PRJPCB		<u>A</u> nnotate
E 🖼 FM Radio Tx PCB Project.PRJP		Back Annotate
E Schematic Sheets	\sim	Signal Integrity
		Convert_
	1	⊆ross Probe
		Select PCB Components
		Schematic Preferences

Annotate							?	
Schematic Annotation Configuration			Proposed Cl	hange Li	st			
	Match By Para	Match By Parameters			Proposed		Location of Part	
O 1 Up then across	Class I	Class I		∧ Sub	Designator	Sub	Schematic Sheet	
	🔨 - 🗌 Class II				C?		FM Radio Tx.SCHDOC	
Q 2 Down then across	Comment		C?		C?		FM Radio Tx.SCHDOC	
	Component	Kind	C?		C?		FM Radio Tx.SCHDOC	
	🚈 - 🗌 Datasheet		C?		C?		FM Radio Tx.SCHDOC	
Across then down	Description		C?		C?		FM Radio Tx.SCHDOC	
	Footprint	-	E?		E?		FM Radio Tx.SCHDOC	
		12	L?		L?		FM Radio Tx.SCHDOC	
Schematic Sheets To Annotate	Designator Index Control	Add Suffix	MK?		MK?		FM Radio Tx.SCHDOC	
Schematic Sheet	Start Index	Suffix	Q?		Q?		FM Radio Tx.SCHDOC	
📕 🗹 🛛 FM Radio Tx.SCHDOC	1		Q?		Q?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
			R?		B?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
			B?		B?		FM Radio Tx.SCHDOC	
	Back	< Annotate	Update Chan	iges List	Reset Designato	irs i	Accept Changes (Create ECO	
							Close	

Engine	eering	Change Order					? 🗙
			Modificatio	ns		S	tatus
E	nable	Action	Affected Object		Affected Document	Check	Done
-		Annotate Component(17)					
	✓	Modify	₽ C? → C1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	₽ C? → C2	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	₽ C? → C3	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	₽ C? → C4	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	₽ C? → C5	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 E? → E1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 L? → L1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 MK? -> MK1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 Q? -> Q1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 Q? -> Q2	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R? -> R1	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R? → R2	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R?-> R3	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R? -> R4	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R? -> R5	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	🔁 R? -> R6	In	🔚 FM Radio Tx.SCHDOC		
	✓	Modify	₽ R? → R7	In	🔚 FM Radio Tx.SCHDOC		
Valid	ate Char		Beport Changes				Close
	ate undr		Tepor changes				CIUSE

Engin	eering	Change Order						? 🔀
			Modificati	ons			S	itatus
E	Enable	Action	Affected Object			Affected Document	Check	Done
🗆 🏙		Annotate Component(17)						
	~	Modify	€ C? → C1	In	[🖬 FM Radio Tx.SCHDOC		۵
	✓	Modify	€ C? → C2	In	[🖬 FM Radio Tx.SCHDOC		
	✓	Modify	€ C? → C3	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ C? → C4	In	[🖬 FM Radio Tx.SCHDOC		O
	✓	Modify	€ C? → C5	In	[🖬 FM Radio Tx.SCHDOC		
	✓	Modify	€ E? -> E1	In	[🖬 FM Radio Tx.SCHDOC		
	✓	Modify	€ L? → L1	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€⊃ MK? → MK1	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	Ð Q?→Q1	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	Ð Q?→Q2	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R1	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R2	In	[📕 FM Radio Tx.SCHDOC		8
	✓	Modify	€ R? → R3	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R4	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R5	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R6	In	[📕 FM Radio Tx.SCHDOC		
	✓	Modify	€ R? → R7	In	[📕 FM Radio Tx.SCHDOC		
Valid	late Chan	ges Execute Changes	Report Changes					Close

After you have Annotated the components, we want to create the PCB board

- Click on the **Files** tab at the bottom left of the program.
- Go to New from template and click on PCB Board Wizard.

New from existing file 🛛 🚿	
🕮 Choose Document	
🕮 Choose Project	
New from template	•
🔜 PCB Templates	
Schematic Templates	
PCB Projects	
🚔 FPGA Projects	
Embedded Projects	
🕮 PCB Board Wizard	
	l
	_
Files Projects Navigator	EM Radio Tx/
X:20 Y:160 Grid:10	

It should bring up the PCB Board Wizard window.

• Click Next.

<u>.</u>		
PCB Board Wizard		? 🛛
	Protel DXP New Board Wizard This wizard will help you create and set up a new printed circuit board. It will take you through some simple steps to define the board layout, manufacturing parameters and layer information.	
	<u>C</u> ancel <u>N</u> ext >	Einish

• Keep it on Imperial, Click Next.

PCB Board Wizard	? 🗙
Choose Board Units Choose the type of measurement units for the board being created.	
If you use mils, click Imperial. If you use millimetres, click Metric.	
C Metric	
Cancel < Back Next >	<u>F</u> inish

• Keep it on Custom, Click Next.

PCB Board Wizard	? 🛛
Choose Board Profiles Select a specific board type from	n the predefined standard profiles or choose custom.
[Custom] A A0 A1 A2 A3 A4 AT long bus (13.3 x 4.2 inches) AT long bus (13.3 x 4.5 inches) AT long bus (13.3 x 4.8 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 long bus with break-away tab (13.3 x 4.5 inches) AT long bus with break-away tab (13.3 x 4.8 inches) AT short bus (7 x 4.2 inches) AT short bus (7 x 4.5 inches) AT short bus (7 x 4.8 inches) AT short bus with break-away tab (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 4.2 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) AT short bus with break-away tab (7 x 4.8 inches) AT short bus with break-away tab (7 x 4.8 inches) AT short bus with break-away tab (7 x 4.8 inches) AT short bus with break-away tab (7 x 4.8 inches) B	No Preview
	Cancel < Back Next > Einish

- On the **Board Size**, change the **Width** to 3000mil and the **Height** to 3000mil (Note: 1000mil = 1 inch).
- Then click Next.

PCB Board Wizard	? 🛛
PCB Board Wizard Choose Board Choose Board C Outline Shape:	Ind Details Details Dimension Layer Boundary Irack Width 10 mil Dimension Line Width 10 mil Keep Out Distance 50 mil From Board Edge I Title Block and Scale Cgmer Cutoff ✓ Legend String Inner CytOff ✓ Dimension Lines
	<u>C</u> ancel < <u>B</u> ack <u>N</u> ext > <u>F</u> inish

- Put the value of "0" for the number of **Power Planes**.
- Click **Next**.

PCB Board Wizard	? 🛛
Choose Board Layers Set the number of signal layers and power planes suitable for your desig	ın.
Signal Layers:	
Power Planes:	
<u>C</u> ancel < <u>B</u> ack <u>N</u> ext>	Einish

- Keep it as Thruhole Vias only.
- Click Next.

PCB Board Wizard	? 🔀
Choose Via Style Choose the routing via style tha	t is suitable for your design.
 	
	<u>Cancel < Back</u> <u>N</u> ext > <u>Finish</u>

- For the "The board has mostly:" select Through-hole components.
- For the "Number of tracks between adjacent pads," select One Track.
- Click Next.



No changes needed on this window.

• Click Next.

PCB Board Wizard					? 🔀
	Choose Default T Choose the minimum new board.	rack and Via sizes track size, via size and the	e copper to cop	per clearance	s to use on the
Minimum <u>T</u> rack Size	<u>12 mil</u>	_ ↓			
Minimum Via <u>W</u> idth	<u>62 mil</u>				
Minimum Via <u>H</u> oleSize	<u>32 mil</u>				
Minimum <u>C</u> learance	<u>13 mil</u>				
		<u>C</u> ancel	< <u>B</u> ack	<u>N</u> ext >	Einish

• Finally click Finish.



Now you should get this window to show up.

• Go to the bottom left of the window, and select the **Projects** tab.

This allows you to select the new PCB file you created, so that you can save it.

• Right-click on the file -> Save As.



	0 <mark>XP</mark> D	esigr	ı Ехр	lorer D	KP - PC	B1.PcbD	oc *	
		DXP	<u>F</u> ile	<u>E</u> dit	⊻iew	Proje <u>c</u> t	<u>P</u> lace	<u>D</u> esigi
		õ		50	. 🔎	P 🔍	0 0	S X
	I 🟳	É	1	22	; 🔁	🛃 🕑	128 👌	8 0
	Proje	ects					-	• 🖉 🗙
	Pr	ojectO	iroup	1.PrjGrp			G	roup
	FN	/ Rad	io Tx	PCB Pro	ject.PR	JPCB	▼ Pr	oject
		-	M R	adio Tx hematic :	PCB	Project.I	PRJP 🖺	
Files Projects PCB Navigator				FM Rad	io Tx.SI	CHDOC *	e	1
X:940mil Y:3640mil			PC 🔤	Bs PCB1.Pc	cbDoc	×	<u></u>	



There are two ways to put the footprints on you PCB board.

- 1) If you are on the **PCB** file
 - Go to Design -> Import Changes From [....]
- 2) If you are on the Schematic file



The Engineering Change Order window should pop up.

- Click on the Validate Changes button at the bottom left.
- Then, click on the **Execute Changes** button at the bottom left.

After you have clicked these two buttons, you should see all green check marks for all the components.

Engineering	j Change Order					?	X
		Modifications				Status	
Enable	Action	Affected Object		Affected Document	Check	Done	
🗆 🏙	Add Components(17)						
✓	Add	🚺 C1	То	📖 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 C2	To	🏨 FM Radio Tx PCB.PCBDOC			
✓	Add	📳 C3	То	🏨 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 C4	To	🏨 FM Radio Tx PCB.PCBDOC			
✓	Add	🎒 C5	То	🎫 FM Radio Tx PCB.PCBDOC			
	Add	📒 E1	To	🏨 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 L1	To	🎫 FM Radio Tx PCB.PCBDOC			
✓	Add	🎒 МК1	То	🎫 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 Q1	То	📖 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 Q2	То	📖 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 R1	To	🎫 FM Radio Tx PCB.PCBDOC			
✓	Add	📒 R2	To	🎫 FM Radio Tx PCB.PCBDOC			
✓	Add	📵 R3	To	📖 FM Radio Tx PCB.PCBDOC			
 Image: A start of the start of	Add	🚺 R4	To	📖 FM Radio Tx PCB.PCBDOC			
 Image: A start of the start of	Add	📳 R5	То	📖 FM Radio Tx PCB.PCBDOC			
 Image: A start of the start of	Add	🚺 R6	То	🎫 FM Radio Tx PCB.PCBDOC			
✓	Add	🚺 R7	To	📰 FM Radio Tx PCB.PCBDOC			
E 🏙	Add Nets(10)						
Image: A start of the start	Add	🔨 GND	To	📖 FM Radio Tx PCB.PCBDOC			
	Add	∼ NetC1_1	То	📰 FM Radio Tx PCB.PCBDOC			.
Validate Cha	nges Execute Changes	<u>R</u> eport Changes				Close	
Engineering	j Change Urder	Mar Keastana				Charlana -	
Enable	Action	Affected Object	1	Affected Decument	Chook	Done	
	Add	Allected Object	To	RM Badio Ty PCB PCBDOC		Done	
	Add	0 112 1 B3	To		×.	×	
	Add	E RA	To		×.	×	
	Add	0 114 0 85	To		×.	×	
	Add	0 B6	To		×.	×	
 ▼ 	Add	E 87	To	EM Badio Ty PCB PCBDOC	×.	×	
	Add Nets(10)	····	10				
	Add	∼ GND	To	EM Badio Ty PCB PCBDOC		(),	
•	hbd	∼ NetC1_1	To	EM Badio Ty PCB PCBDOC	×.	ő	
×	Add	∼ NetC2 1	To	EM Badio Ty PCB PCBDOC		×.	
×	Add	~ NetC3 1	To	EM Badio Ty PCB PCBDOC		×.	
· · · · · ·			10		12.00	W	
	Add	∼ NetC4_1	To	EM Badio Te PCB PCBDOC	l.		
 ✓ ✓ 	Add	∼ NetC4_1 ∼ NetC5_1	To To	FM Radio Tx PCB.PCBDOC FM RADIO FM	Š		
▼ ▼	Add Add	∼ NetC4_1 ∼ NetC5_1 ∽ NetC5_2	To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC FM Radio Tx PCB PCBDOC	Ö	0	
> > >	Add Add Add	∼ NetC4_1 ∼ NetC5_1 ∼ NetC5_2 ∼ NetC5_1	To To To To	FM Radio Tx PCB.PCBDOC	ě e e	0	
> > > >	Add Add Add Add	 ∼ NetC4_1 ∼ NetC5_1 ∼ NetC5_2 ∼ NetQ2_1 ∼ NetP3_1 	To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC	00000	0	
> > > > >	Add Add Add Add Add Add	 NetC4_1 NetC5_1 NetC5_2 NetQ2_1 NetR3_1 VCC 	To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			
Y Y Y Y Y	Add Add Add Add Add Add Add	NetC4_1 NetC5_1 NetC5_2 NetQ2_1 NetR3_1 VCC	To To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			
▼ ▼ ▼ ▼ ▼	Add Add Add Add Add Add Add Component Classes(1 Add	 NetC4_1 NetC5_1 NetC5_2 NetR3_1 VCC 	To To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			
Y Y Y Y Y Y Y	Add Add Add Add Add Add Add Component Classes[Add	 NetC4_1 NetC5_1 NetC5_2 NetR3_1 VCC FM Radio Tx 	To To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			
Y Y Y Y Y Y Y Y	Add Add Add Add Add Add Add Component Classes(1 Add Add Rooms(1)	 NetC4_1 NetC5_1 NetC5_2 NetQ2_1 NetR3_1 VCC FM Radio Tx 	To To To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			
▼ ▼ ▼ ▼ ▼ ▼ ▼ ▼	Add Add Add Add Add Add Add Component Classes(Add Add Rooms(1) Add	 NetC4_1 NetC5_1 NetC5_2 NetR3_1 VCC FM Radio Tx @ Room FM Radio Tx (Scope=I) 	To To To To To To	FM Radio Tx PCB.PCBDOC FM Radio Tx PCB.PCBDOC			Ţ

After you've verified that there are green check marks,

Click Close

Your components should show up next to the black board in a purplish enclosure (Room).

СРКЈР		_						
:								
DOC *	B					_		
						R6 R7 Mk1 R5 02 (5 01 0	4 C3 R2 R3 R4 R1 C2 C1	
							M Radio Tx	
						LEGEND		10
tor /) om Laver (Mechanical 1 (N	/echanical 2 /Mechanical	3 / Mechanical 4 / Mech	nanical 5 (Mechanical 6 (Mechanical 7	/ /Mechanical 8 / Mechanical 9 /	Ma Real S⊳⊽ Ma
		<u></u>	,,				,,	

- Next, **Right-click** in the enclosure.
- Go to **Design** -> **Rules**.



The PCB Rules and Constraints Editor window should show up.

• Find "Room Definition" under the Type column and Uncheck it.

We did this because, if it was checked, the enclosure (room) is associated with the components. For example, if you try to move the enclosure, the components will move along with it, which leads to parts not being on the board.

🕢 Design Rules	Name 🛆	Pri	En	Туре	Category	Scope	Attributes
Electrical	Clearance	1	-	Clearance	Electrical	All - All	Clearance = 13mil
🛛 🎝 Routing	ComponentClearance	1	•	Component Clearance	Placement	All - All	Clearance = 10mil
SMT	🖧 Fanout_BGA	1	~	Fanout Control	Routing	IsBGA	Style - Auto Direc
- Mask	🖧 Fanout_Default	5	✓	Fanout Control	Routing	All	Style - Auto Direc
Plane	S Fanout_LCC	2	•	Fanout Control	Routing	IsLCC	Style - Auto Direc
I estpoint	S Fanout_Small	4	-	Fanout Control	Routing	(CompPinCount < 5)	Style - Auto Direc
Manufacturing	S Fanout_SOIC	3	•	Fanout Control	Routing	IsSOIC	Style - Auto Direc
High Speed	FM Radio Tx	1		Room Definition	Placement	InComponentClass('FM	Region (BR) = (700
Signal Integrity	Height	1	•	Height	Placement	All	Pref Height = 500m
1-Illing piðuar u keðurða	The HoleSize	1	•	Hole Size	Manufacturing	All	Min = 1 mil Max =
	LayerPairs	1	•	Layer Pairs	Manufacturing	All	Layer Pairs - Enford
	PasteMaskExpansio	1		Paste Mask Expansion	Mask	All	Expansion = Omil
	PlaneClearance	1		Power Plane Clearance	Plane	All	Clearance = 20mil
	PlaneConnect	1	•	Power Plane Connect S	Plane	All	Style - Relief Conne
	PolygonConnect	1	-	Polygon Connect Style	Plane	All	Style - Relief Conne
	a RoutingCorners	1	-	Routing Corners	Routing	All	Style - 45 Degree
	🖧 RoutingLayers	1	v	Routing Layers	Routing	All	TopLayer - Horizon
	a RoutingPriority	1	~	Routing Priority	Routing	All	Priority = 0
	🖧 RoutingTopology	1	~	Routing Topology	Routing	All	Topology - Shortes
	🖧 RoutingVias	1	-	Routing Via Style	Routing	All	Pref Size = 62mil
	ShortCircuit	1	v	Short-Circuit	Electrical	All - All	Short Circuit - Not A
	SolderMaskExpansi	1	v	Solder Mask Expansion	Mask	All	Expansion = 4mil
	🟏 Testpoint	1	~	Testpoint Style	Testpoint	All	Under Comp - Allov
	🟏 TestPointUsage	1	~	Testpoint Usage	Testpoint	All	Testpoint - Require
	TunRoutedNet	1	~	Un-Routed Net	Electrical	All	(No Attributes)
	So Width	1	-	Width	Routing	All	Pref Width = 12mil

Place your components wherever you like within the black grid.

Note: When you have selected a component to move, you can press the SPACEBAR to rotate the component.



After you have placed your components, save your file.

• Next, go to Auto Route -> All



The Situs Routing Strategies window should show up.

- Keep it like it is "Default 2 Layer Board".
- Click Route All.

The software should start routing all your connections.

A Messages window should pop up at the same time.

When everything is finished, you should get "Routing finished with 0 contention(s). Failed to complete 0 connection(s)" in the Message window.

Situs Routing Strategies	? 🗙
Strategy No Warnings	1
Available Routing Strategies	
Name 🛆 🛛	Description
Cleanup	Default cleanup strategy
Default 2 Layer Board D	Default strategy for routing two-layer boards
Default 2 Layer With Edge Connectors D	Default strategy for two-layer boards with edge connectors
Default Multi Layer Board D	efault strategy for routing multilayer boards
Via Miser S	Strategy for routing multilayer boards with aggressive via minimization
Add <u>R</u> emove <u>E</u> dit	Duplicate Lock All Pre-routes
Routing Rules	Route All Cancel

Date No.	Time	Document Source Message
4/5/2005 11	11:48:56 AM	FM Radio Tx PCB Situs 20 of 23 connections routed (86.96%) in 0 Seconds
4/5/2005 12	11:48:56 AM	FM Radio Tx PCB Situs Completed Main in 0 Seconds
4/5/2005 13	11:48:56 AM	FM Radio Tx PCB Situs Starting Completion
4/5/2005 14	11:48:56 AM	FM Radio Tx PCB Situs Completed Completion in 0 Seconds
4/5/2005 15	11:48:56 AM	FM Radio Tx PCB Situs Starting Straighten
4/5/2005 16	11:48:56 AM	FM Radio Tx PCB Situs Completed Straighten in 0 Seconds
4/5/2005 17	11:48:56 AM	FM Radio Tx PCB Situs 23 of 23 connections routed (100.00%) in 0 Seconds
4/5/2005 18	11:48:56 AM	FM Radio Tx PCB Situs Routing finished with 0 contentions(s). Failed to complete 0 connection(s) in 0 Seconds
4/5/20	11:48:56 AM	FM Radio Tx PCB Situs Routing finished with 0 contentions(s). Failed to complete 0 connection(s) in 0 Seconds

If you look at your board, all the white connections between the components have changed to red and blue lines.

The **Red** lines represent **Horizontal** connections that are on the **Front** of the PCB board. The **Blue** lines represent **Vertical** connections that are on the **Back** of the PCB board.



The next step is to run a Design Rule Check on the board to make sure it meets the requirements of the software to be a good board.

• Go to **Tools** -> **Design Rule Check**.



A Design Rule Checker window should pop up. Keep everything like it is and

• Click on Run Design Rule Check.

Design Rule Checker	? 🛛
Image: Second system Particula Image: Second system Image: Second system Image: Second system <td>It Options Report <u>File</u> Violations et Details al Plane Warnings Shorting Copper 500 violations found e process of rule checking enable only the rules that are required for the task med. Note: Options are only enabled when corresponding rules have been C tests for design rule violations as you work. Include a Design Rule in the is dialog to be able to test for a particular rule type.</td>	It Options Report <u>File</u> Violations et Details al Plane Warnings Shorting Copper 500 violations found e process of rule checking enable only the rules that are required for the task med. Note: Options are only enabled when corresponding rules have been C tests for design rule violations as you work. Include a Design Rule in the is dialog to be able to test for a particular rule type.
<u>B</u> un Design Rule Check	Close

A Messages window will pop up, you can close that window.

In the Main window, the results of the Design Rule Check should be displayed. 6 Constraint Rules are listed.

- Short-Circuit Constraint whether or not you have short-circuits on your board
- Broken-Net Constraint any broken connections on your board
- Height Constraint do you fit the minimum and maximum height constraints
- Hole Size Constraint do you fit the minimum and maximum hole constraints
- Width Constraint do you fit the minimum and maximum width constraints
- Clearance Constraint whether your components are space right from each other

Me Design Explorer DXP - \\FILER\TUAN-HUYNH\PCBBoa	irds\FM Radio Tx PCB.DRC	<u>- 8 ×</u>
DXP File Edit View Project Tools Window	Help	
🗅 😅 🔛 🎒 🗲 🖻 🛍 🗠 🗠 🗡 😧		
Search		
Search for	Protel Design System Design Rule Check	- brani
Search text	PCB File : \\FILER\TUAN-HUTNH\PCBBoards\FM Radio Tx PCB.PCBDOC	
Search text	Date : 4/8/2005	
	Time : 10:35:22 AM	
Search Liear 🌾	Processing Rule : Short-Circuit Constraint (Allowed=No) (All),(All)	
Search Results	Rule Violations :0	
	Processing Rule - Broken-Net Constraint ((\$11))	
	Rule Violations :0	
	Processing Rule : Height Constraint (Min=Omil) (Max=1000mil) (Prefered=500mil) (All) Dule Violations :0	
	Processing Rule : Hole Size Constraint (Min=1mil) (Max=100mil) (All)	
	Rule Violations :U	
	Processing Rule : Width Constraint (Min=12mil) (Max=12mil) (Preferred=12mil) (All)	
	Rule Violations :0	
	Processing Rule : Clearance Constraint (Gan=13mil) (All).(All)	
Goto	Violation between Pad Q2-2 (3360mil, 4980mil) Multi-Layer and	
	Pad Q2-1(3410mil,4980mil) Multi-Layer	
Search Options	Violation between Pad Q2-3(3310m1,4950m1) Multi-Layer and Pad Q2-2(3360m1,4980m1) Multi-Layer	
	Violation between Pad Q1-2 (3780mil, 5000mil) Multi-Layer and	
Lase sensitive	Pad Q1-1(3830mil, 5000mil) Multi-Layer	
Begular Expression	Violation between Pad Q1-3(3730mil,5000mil) Multi-Layer and Pad O1-2(3780mil,5000mil) Wulti-Layer	
All Text Docs	Rule Violations :4	
		_
	Processing Rule : Room FM Radio IX (Bounding Region = (7140mil, 4960mil, 10400mil, 5560mil) (Disabled)(InComponent Bule Violations :0	C
	Violations Detected : 4	
		-
Files / Projects / Navigator) Search /	A	<u> </u>
21:39	Inset Search Files Libraries Maximate Projects Browner Pane	ls Help
		- nop
📰 Scarc 📋 💹 😸 🖓 👹 👘 🔤 Protei Instructions -	Million Pille university of Tuba 2 PillergyRadio.PPI - Millions Waldesign Explorer DXP	:56 AM

After you have verified that your board meets the Design Rules, your board is complete. There is an extra feature that allows you to view your board in 3D.

Note: The software guesses what your board may look like with components on it.

- Go to View -> Board in 3D.
- A Design Explorer Information window will pop up.
- Click OK. • 🚾 Design Explorer DXP - \\Filer\tuan-huynh\PCBBoards\FM F DXP File Edit View Project Place <u>D</u>esign Tools 🖨 🖪 🚇 **/** 🖻 Fit Document Ctrl+PgDn 🗋 🖻 🔛 Fit Sheet 🕒 🖄 🚰 🖾 Ŷ ۲ Fit Board FM Rad Area ProjectGroup1.PrjGrp Around Point FM Radio Tx PCB Pro 👧 🗧 Selected Objects O Filtered Objects 🖃 📑 FM Radio T> Ð 🖃 📖 Schematic Zoom In PgUp 🔙 FM Rad 🔎 Zoom <u>O</u>ut PgDn 🖃 🎒 PCBs Zoom Last 🔡 FM Rad Home Pan <u>R</u>efresh End 24 Board in <u>3</u>D Toolbars ۲ Workspace Panels ۲ Design Explorer Information × ~ Status Bar Command Status The 3D Viewer uses OpenGL to render the PCB. For i best results ensure that your video card supports Connections ۶ OpenGL and has the latest available drivers. <u>G</u>rids ۲ ΟK Toggle <u>U</u>nits Q

Now you can see your board in 3D in the main window.

In the left hand side, you see a black window with a smaller representation of your board. If you click and hold and move the mouse in any direction, the board will rotate for a 360 view of your board.

