PROTEL DXP

# Printed Circuit Board Design

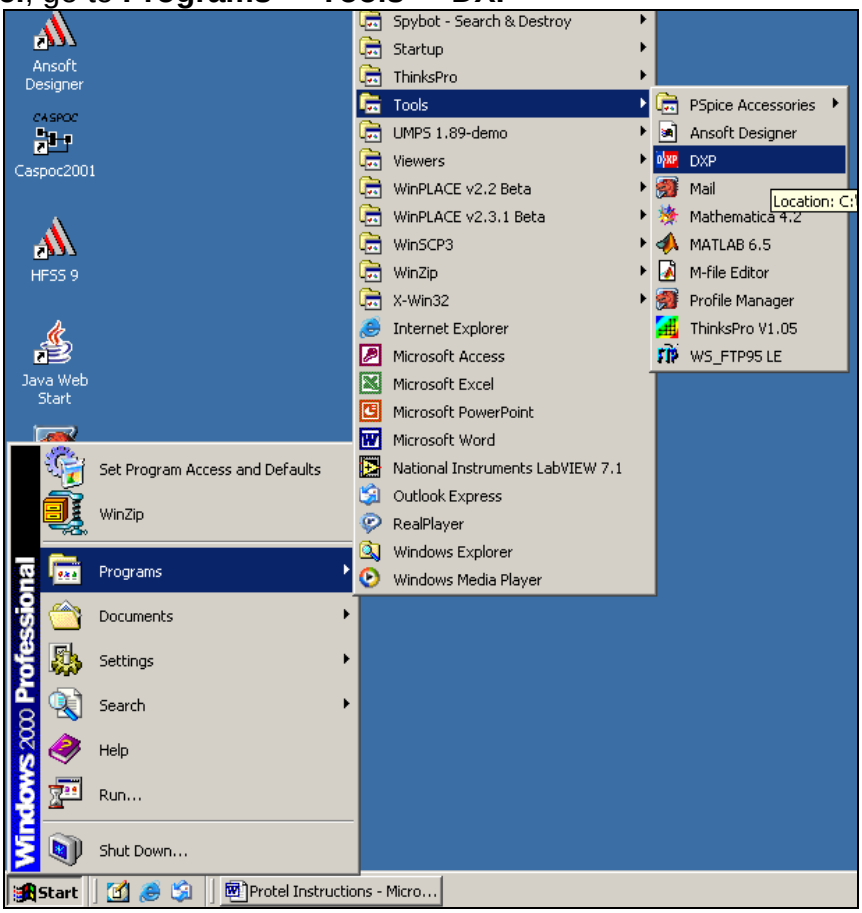
## Manual

(Basics)

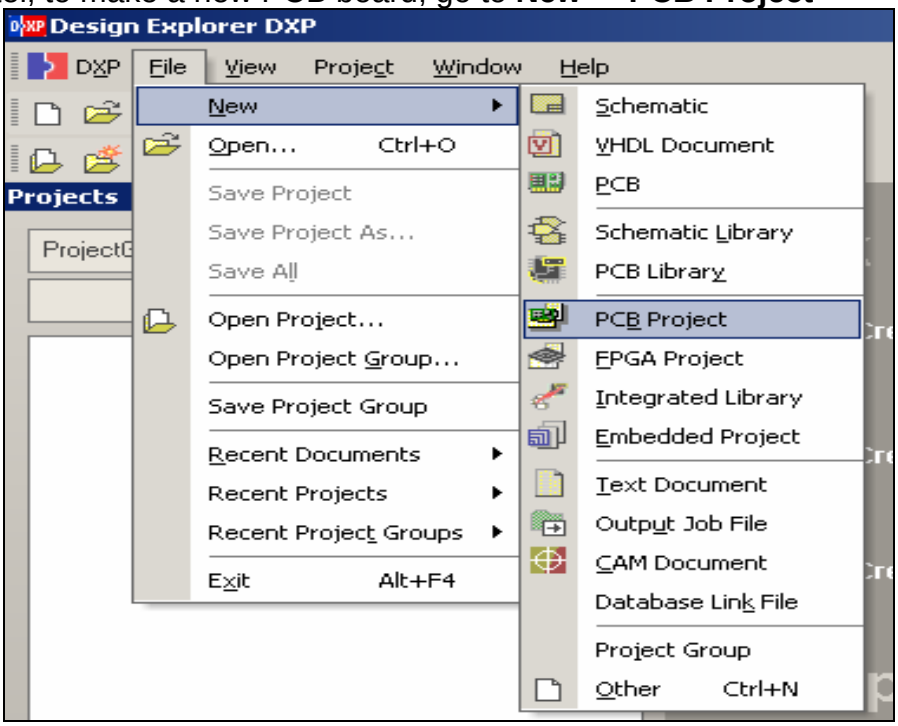
Tuan Huynh
*University of Tulsa*
*Electrical Engineering*
*2004 -2006*

# TABLE OF CONTENTS

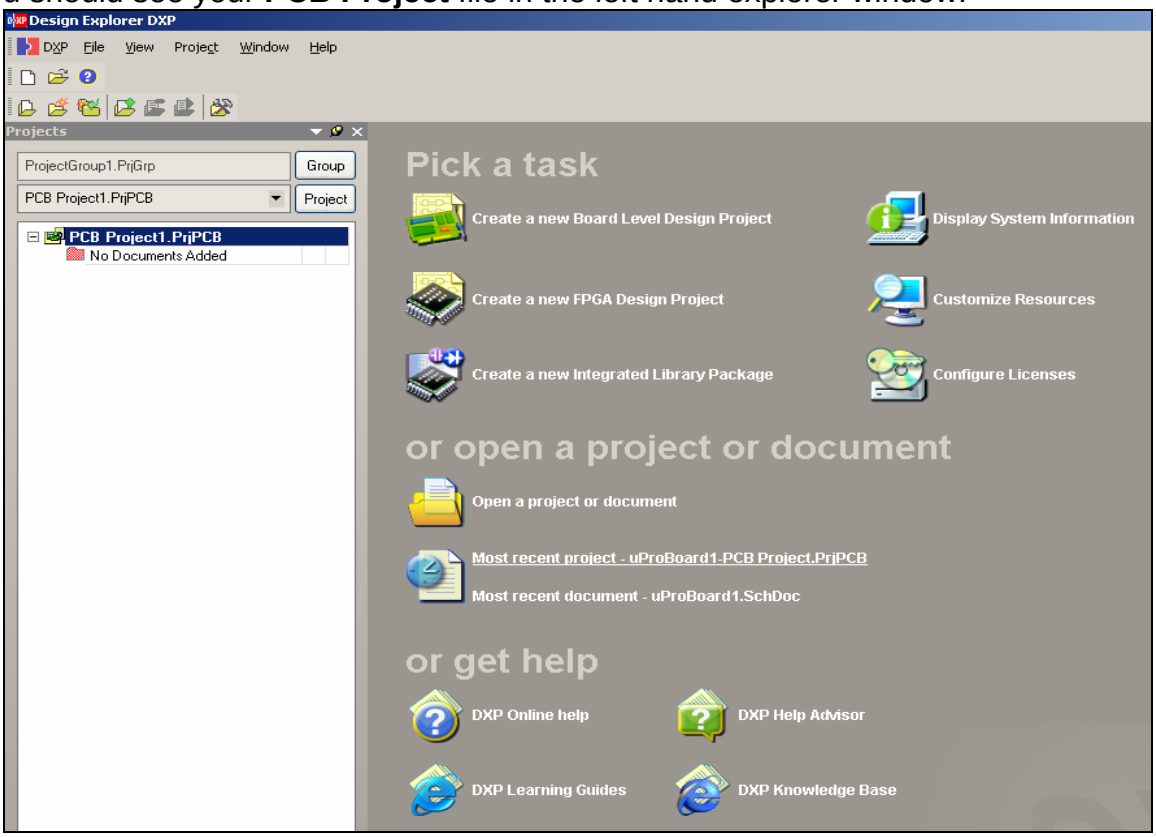
- To start **Protel**, go to **Programs** -> **Tools** -> **DXP**



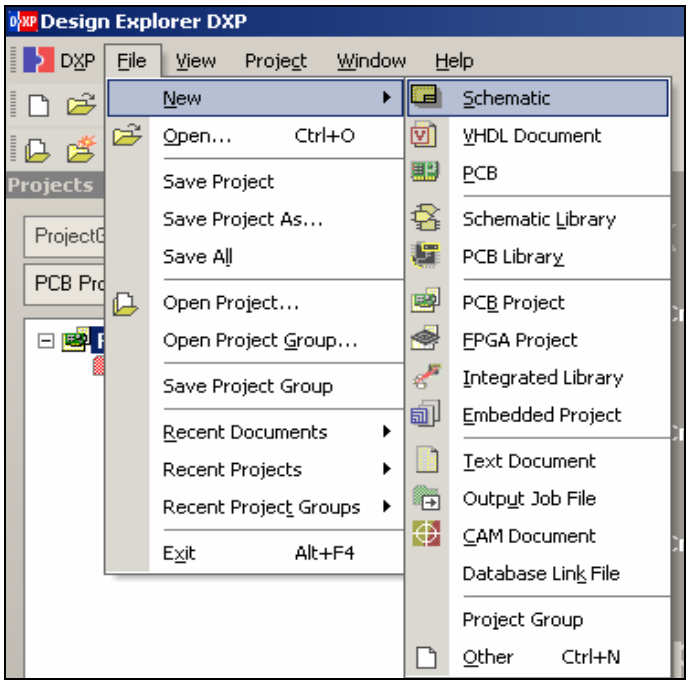

- Next in Protel, to make a new PCB board, go to **New** -> **PCB Project**



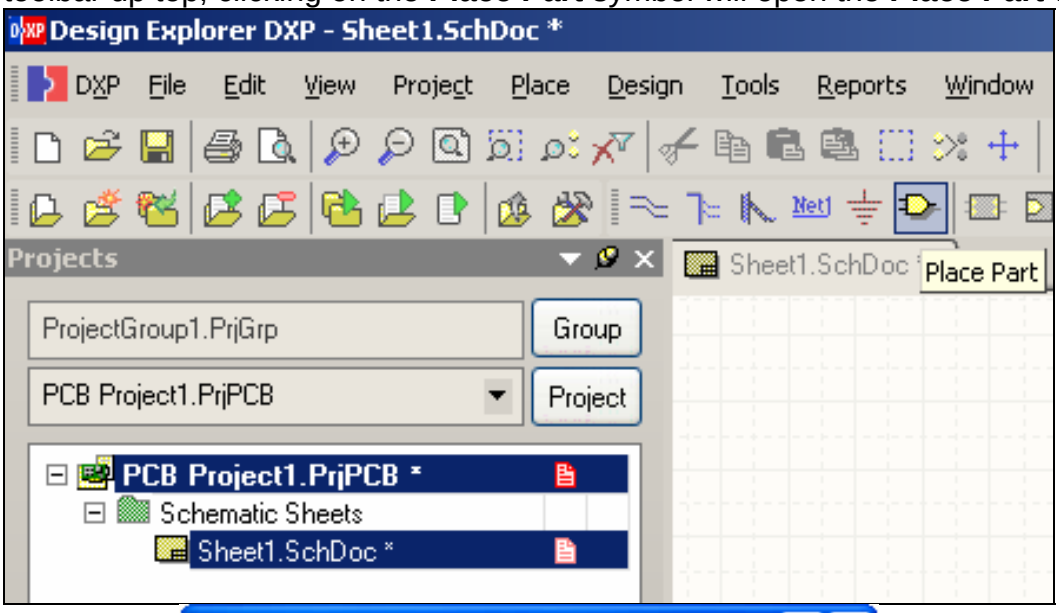
- You should see your **PCB Project** file in the left hand explorer window.

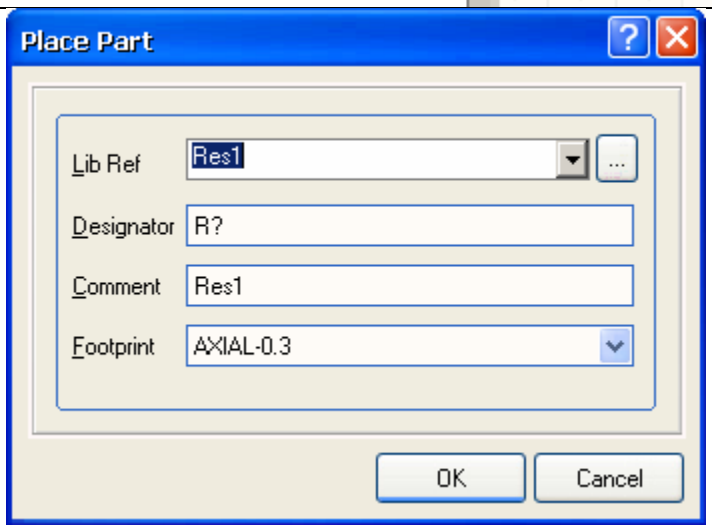



- First you want to draw your schematic of your circuit.
- Go to **File** -> **New** -> **Schematic**
- Now in the left hand window, you should see **Sheet1.SchDoc*** in your project. If it is not in your project drag that file into your project.
- Next, to draw your circuit, you need to click on the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



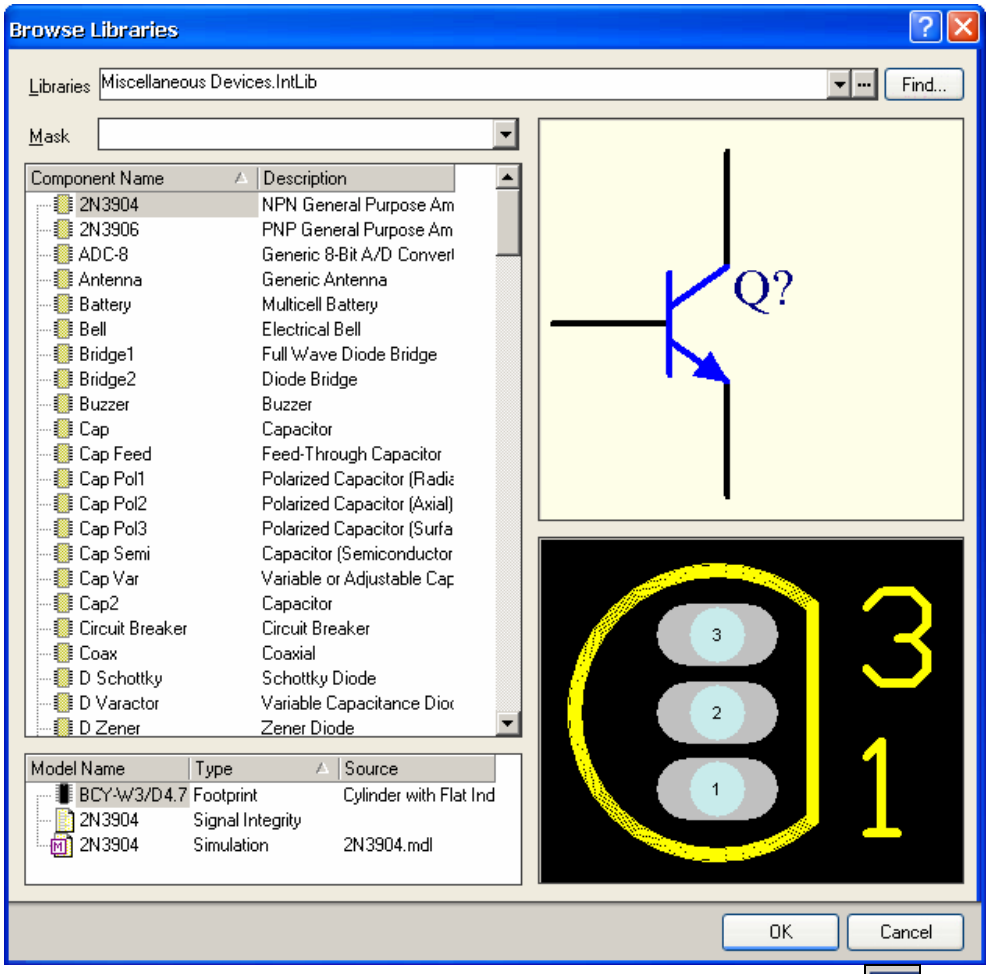
- In the toolbar up top, clicking on the **Place Part** symbol will open the **Place Part** window.




- Click on the  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.

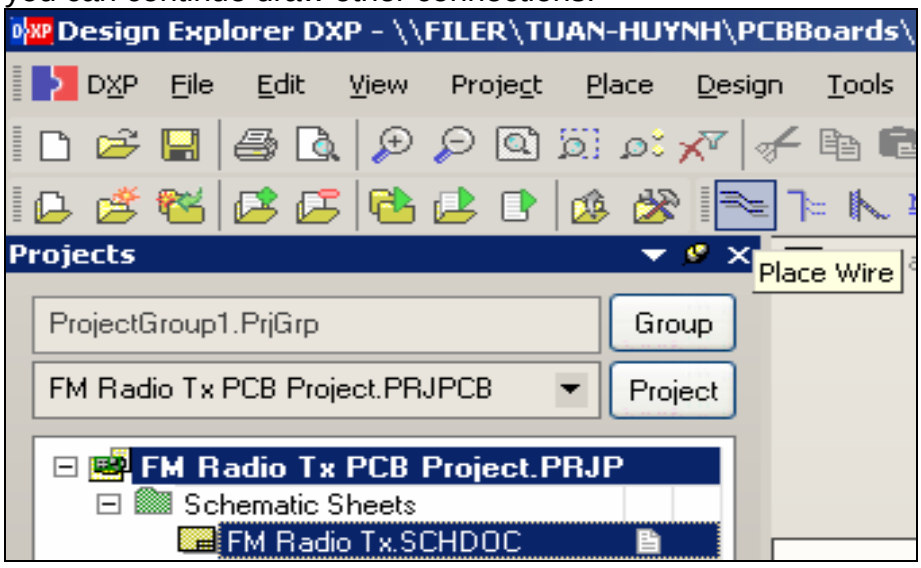
**Note:** When you have selected a component to move or placing a component, you can press the **SPACEBAR** to rotate the component.



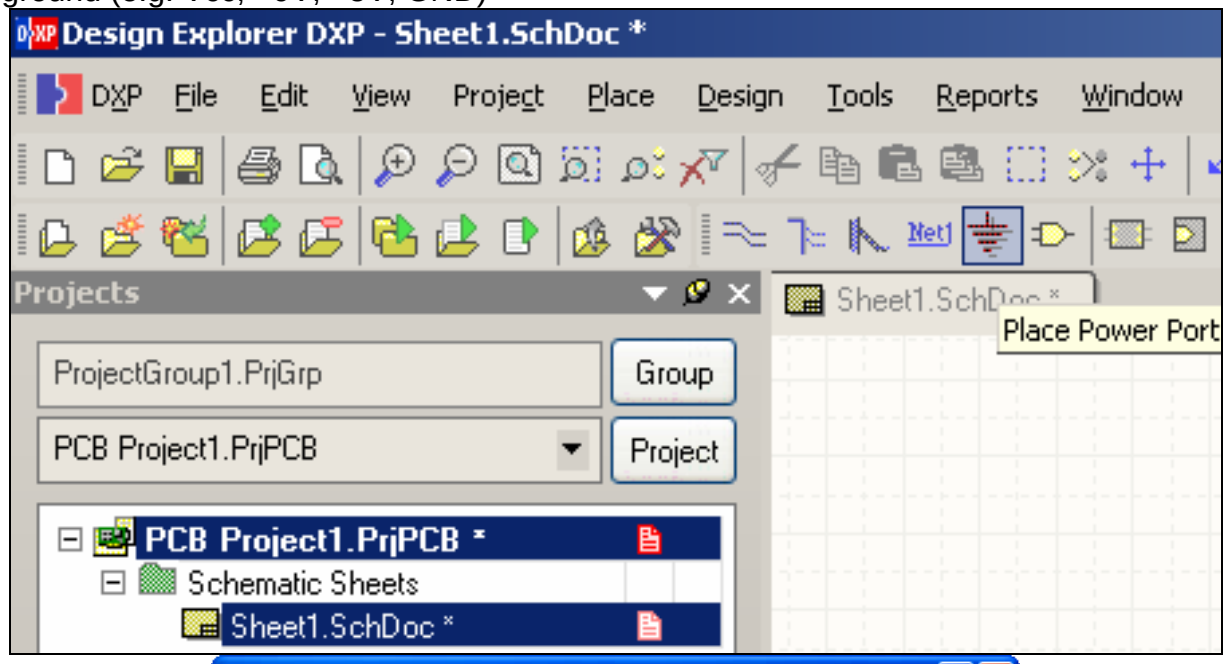

- To make connections between components, click on the **Wire** symbol  , now you can draw wires.

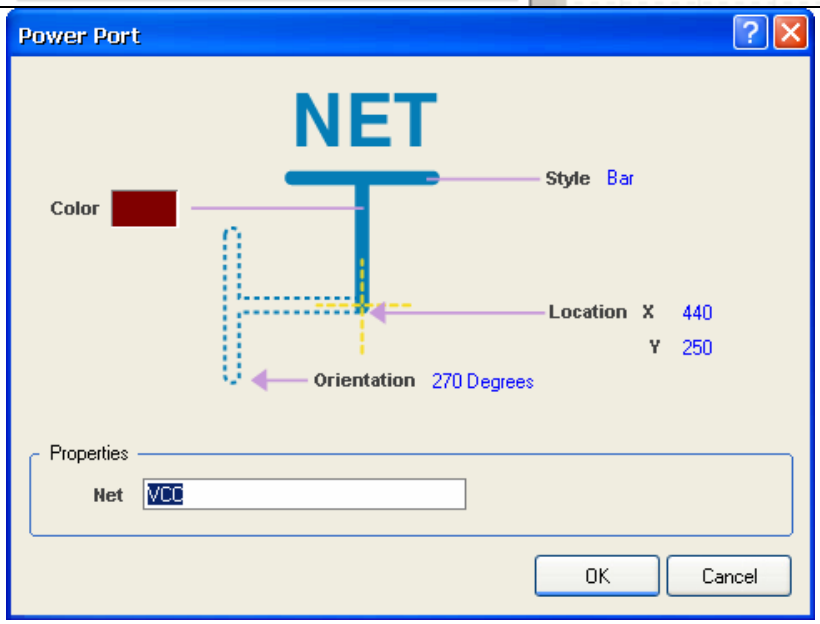
When you see a **Red X** when you move your cursor over a component, it means it's an anchor point for the wire.

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

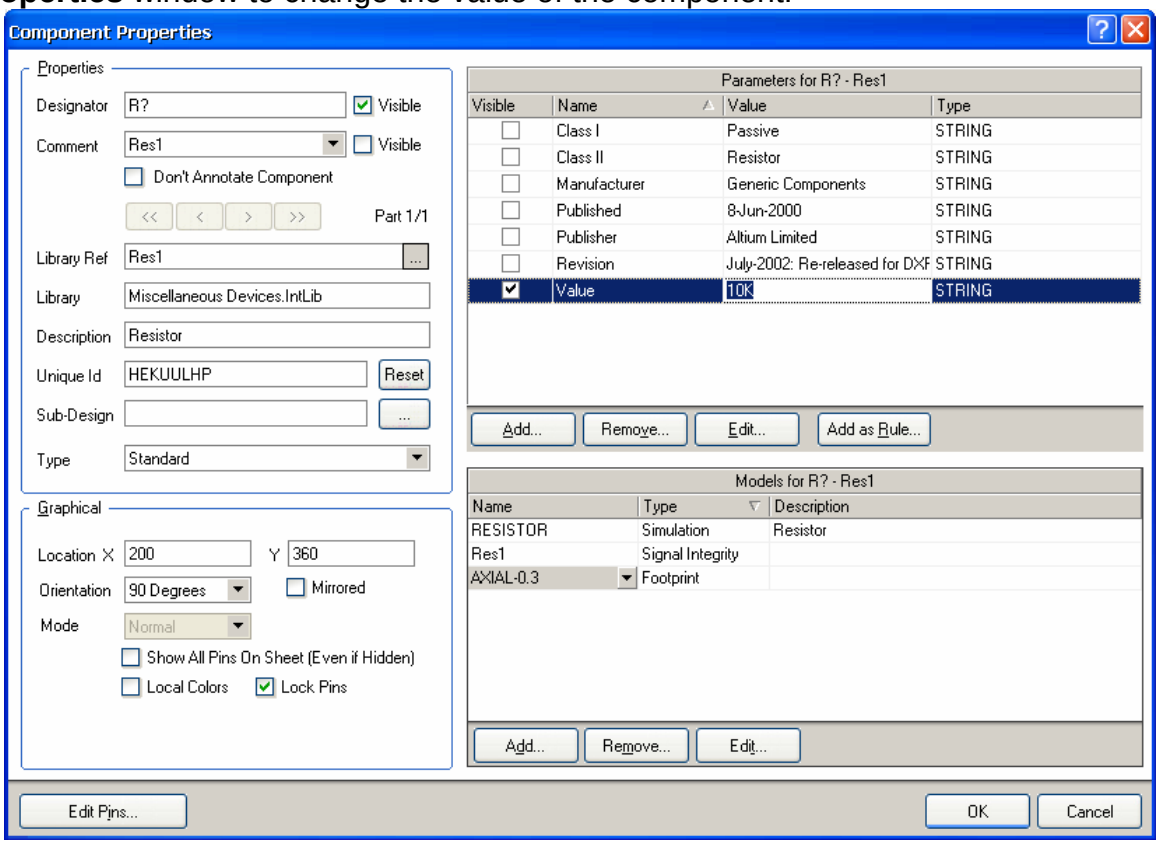


- To add power and ground points on your circuit, click on the **Power Port** symbol 
- After placing a **Power Port**, you can double click on the component to get the **Power Port** window.
- In this window, you change the **Net** in the **Properties** section to represent power or ground (e.g. Vcc,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

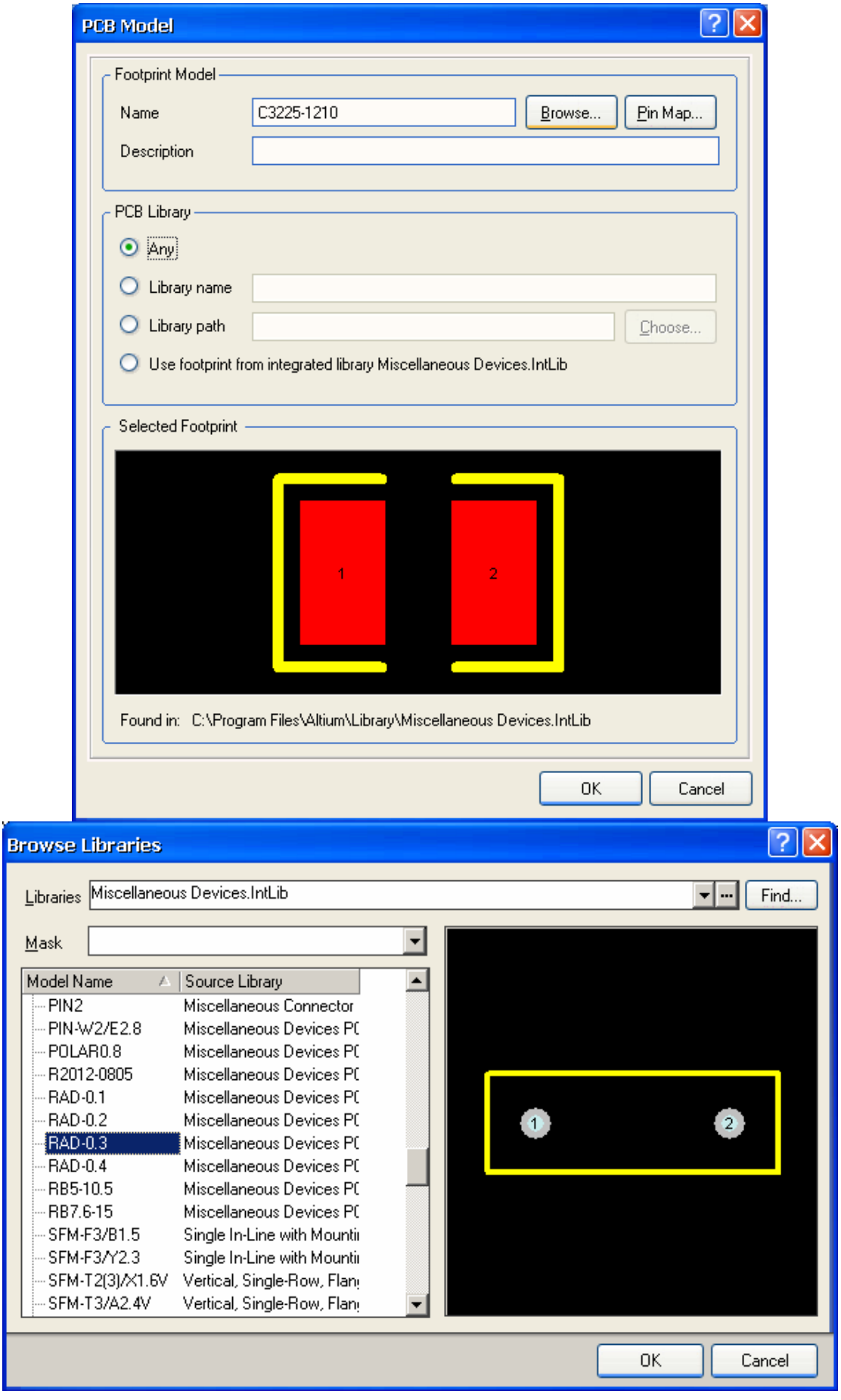


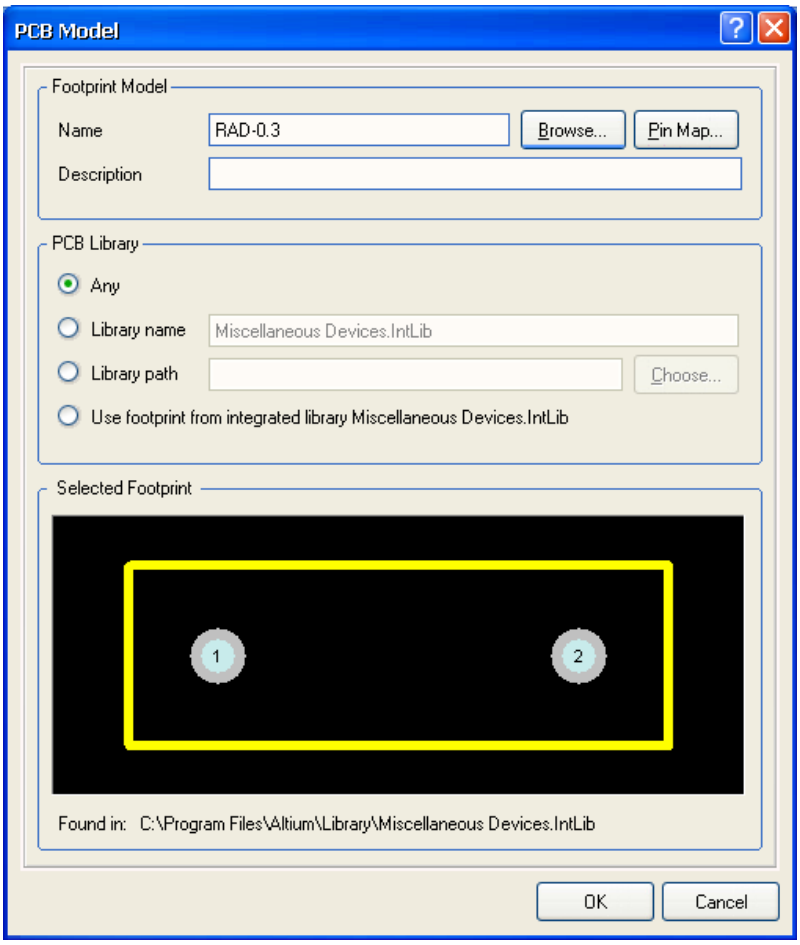



- Now to change the **Value** of a component, you can double click the **Value** next to a component or you can double click the component itself to bring up a **Component Properties** window to change the value of the component.

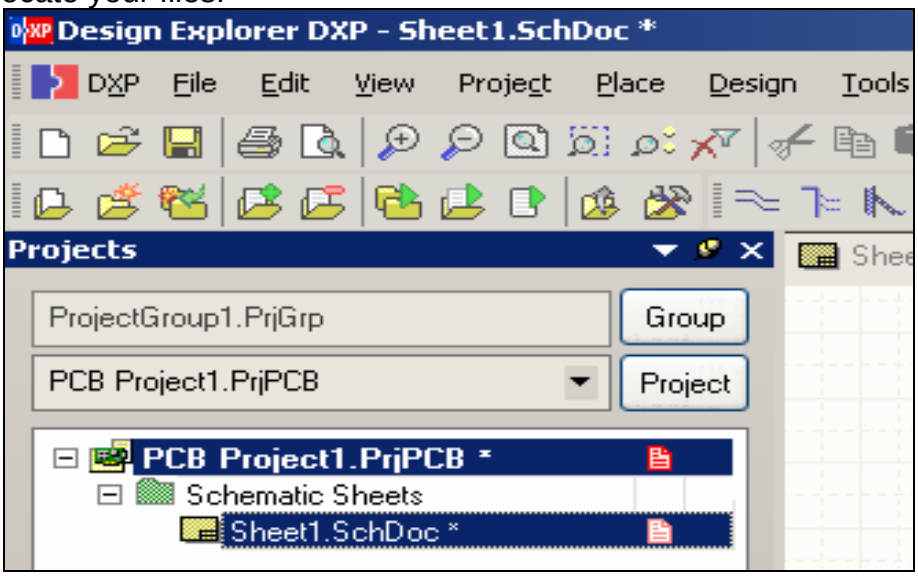


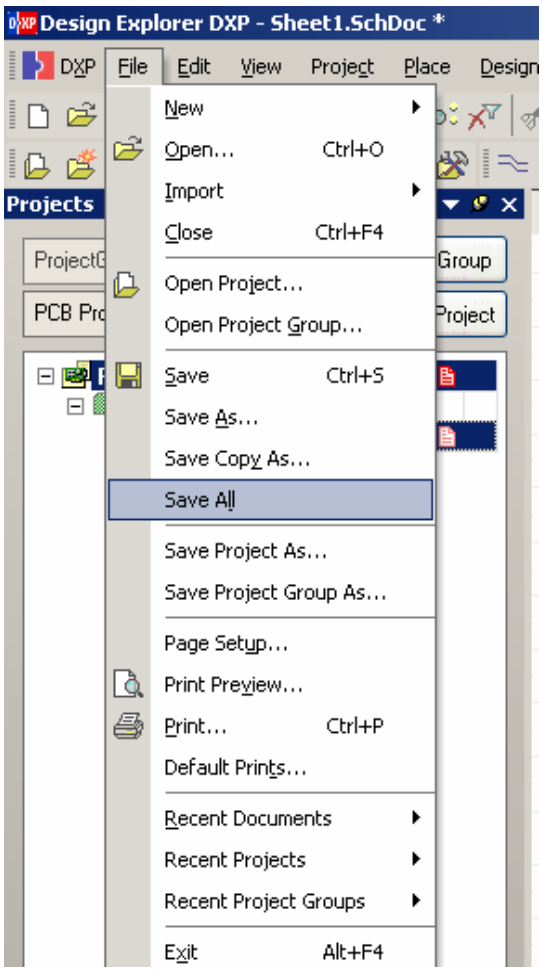

- The capacitors that were placed on the schematic do not have the correct footprint.
- Go to a capacitor on your schematic and double click on it to bring up the **Component Properties** window.
- Click to the left of the **Footprint** text in the bottom right of the window and then click **Edit**.
- 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.

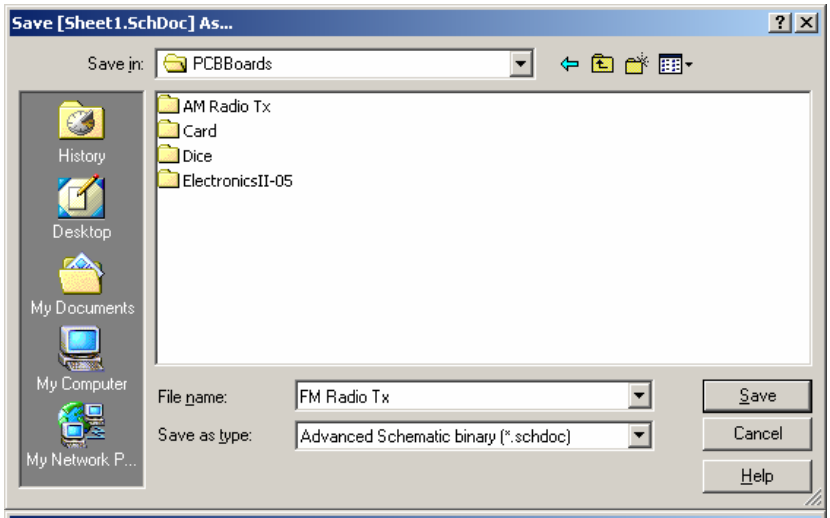

PCB Model
Footprint Model
Name
C3225-1210
Browse...
Pin Map...
Description
PCB Library
Any
Library name
Library path
Choose...
Use footprint from integrated library Miscellaneous Devices.IntLib
Selected Footprint
1
2
Found in: C:\Program Files\Altium\Library\Miscellaneous Devices.IntLib
OK
Cancel
Browse Libraries
Libraries
Miscellaneous Devices.IntLib
Find...
Mask
Model Name
Source Library
PIN2
Miscellaneous Connector
PIN-W2/E2.8
Miscellaneous Devices PC
POLAR0.8
Miscellaneous Devices PC
R2012-0805
Miscellaneous Devices PC
RAD-0.1
Miscellaneous Devices PC
RAD-0.2
Miscellaneous Devices PC
RAD-0.3
Miscellaneous Devices PC
RAD-0.4
Miscellaneous Devices PC
RB5-10.5
Miscellaneous Devices PC
RB7.6-15
Miscellaneous Devices PC
SFM-F3/B1.5
Single In-Line with Mountii
SFM-F3/Y2.3
Single In-Line with Mountii
SFM-T2(3)/X1.6V
Vertical, Single-Row, Flan
SFM-T3/A2.4V
Vertical, Single-Row, Flan
1
2
OK
Cancel

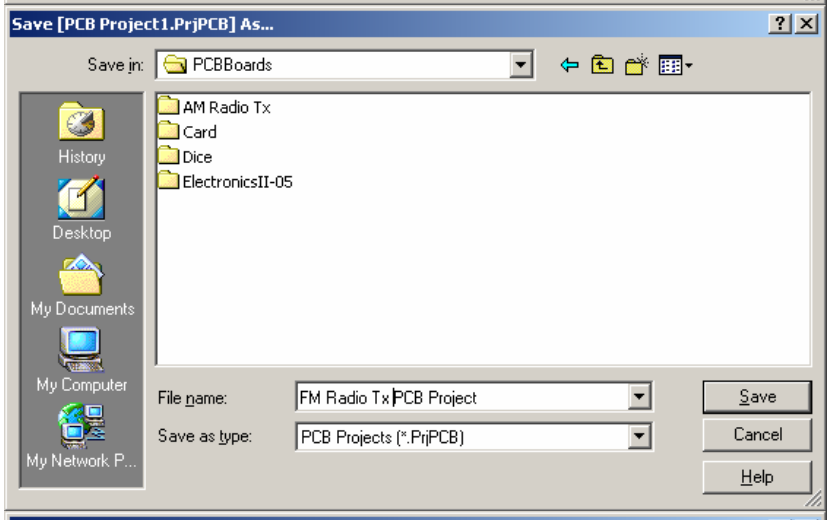


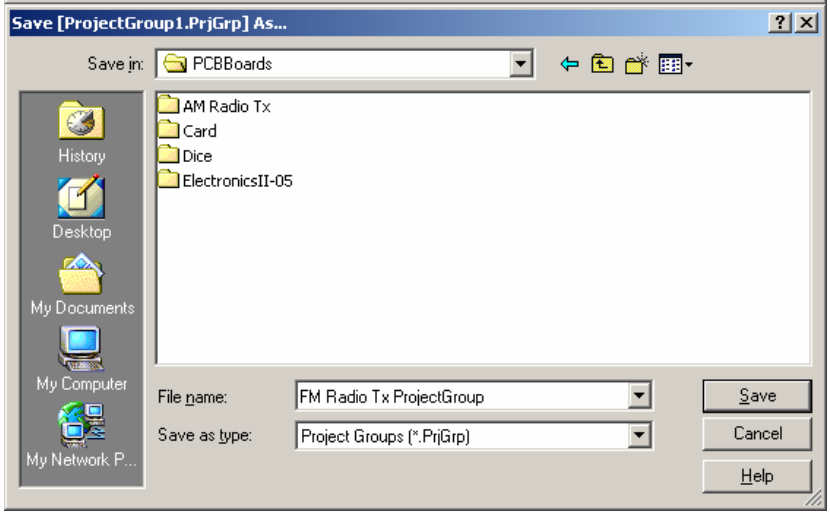

- If you look in the left hand window, the files in there are colored **Red**, which means the files are not yet saved.
- To save go to **File** -> **Save All**.
- By going through this method of **Save All** instead of just **Save As** or **Save**, this allows you to save everything including your **Schematic**, **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.



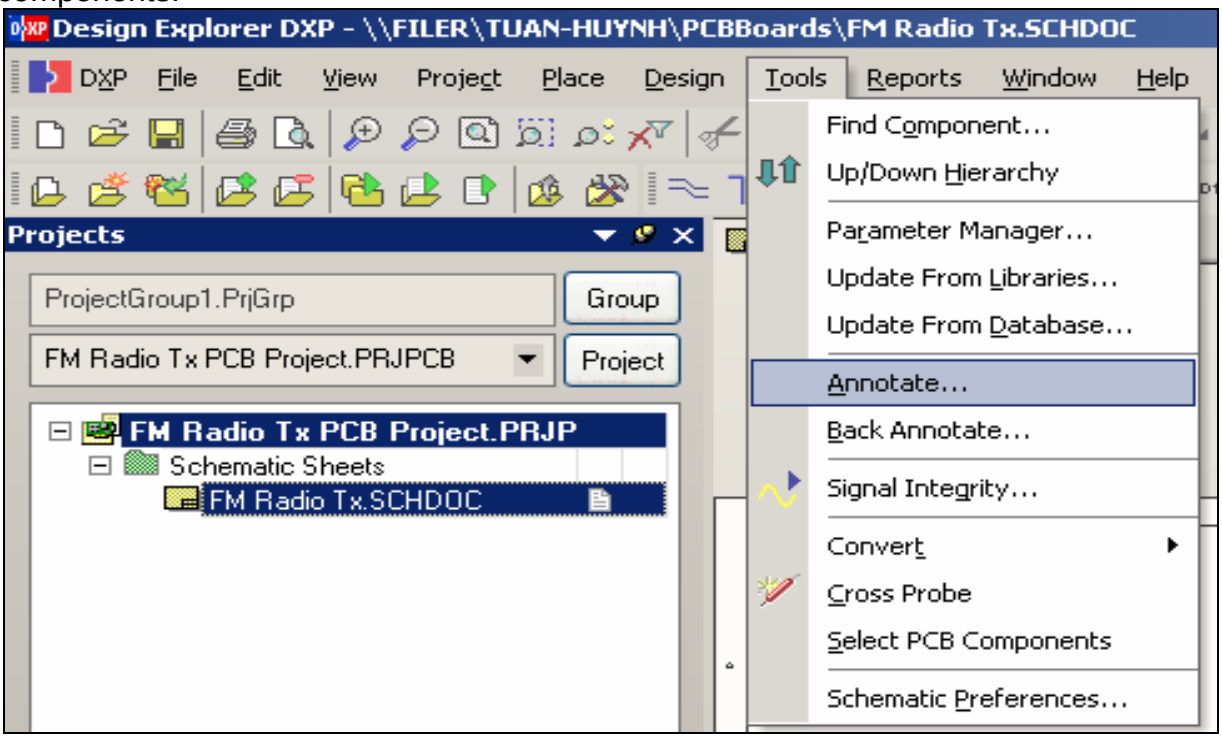

Design Explorer DXP - Sheet1.SchDoc *
DXP
File
Edit
View
Project
Place
Design
Projects
New
Open... Ctrl+O
Import
Close Ctrl+F4
Open Project...
Open Project Group...
Save Ctrl+S
Save As...
Save Copy As...
Save All
Save Project As...
Save Project Group As...
Page Setup...
Print Preview...
Print... Ctrl+P
Default Prints...
Recent Documents
Recent Projects
Recent Project Groups
Exit Alt+F4

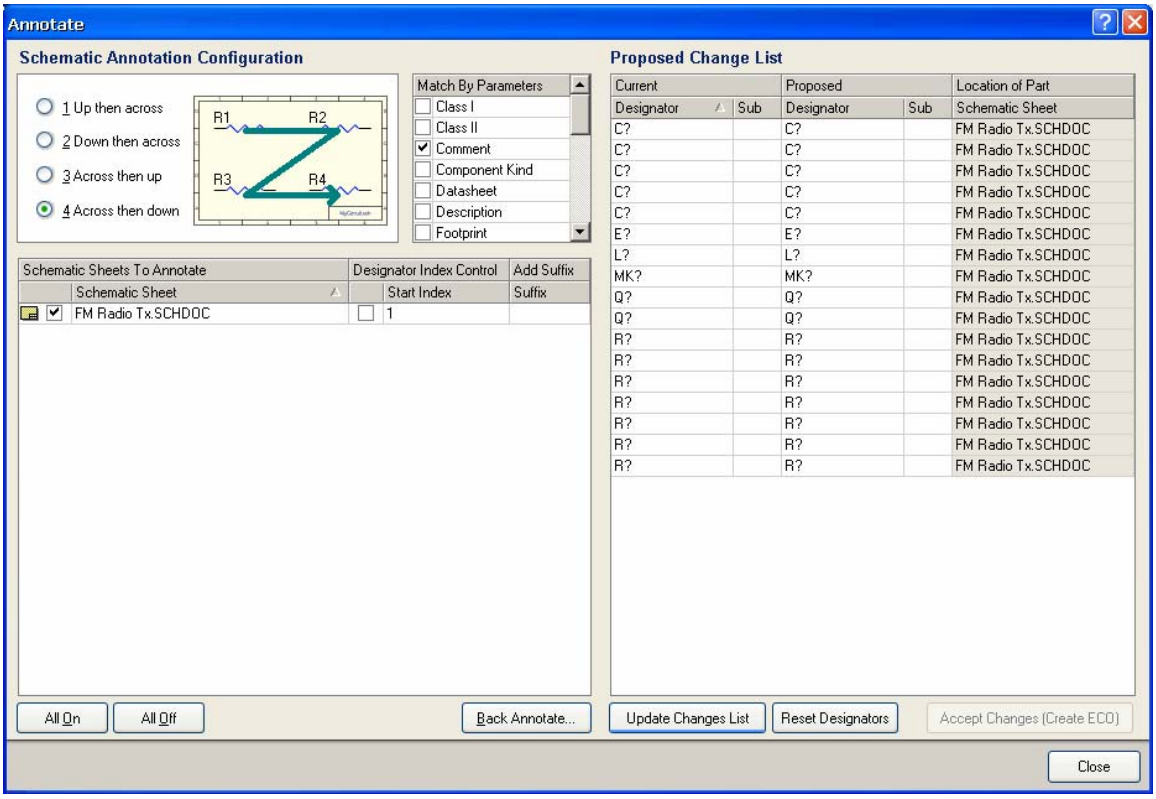

Save [Sheet1.SchDoc] As...
Save in: PCBBoards
AM Radio Tx
Card
Dice
ElectronicsII-05
History
Desktop
My Documents
My Computer
My Network P...
File name: FM Radio Tx
Save as type: Advanced Schematic binary (*.schdoc)
Save
Cancel
Help

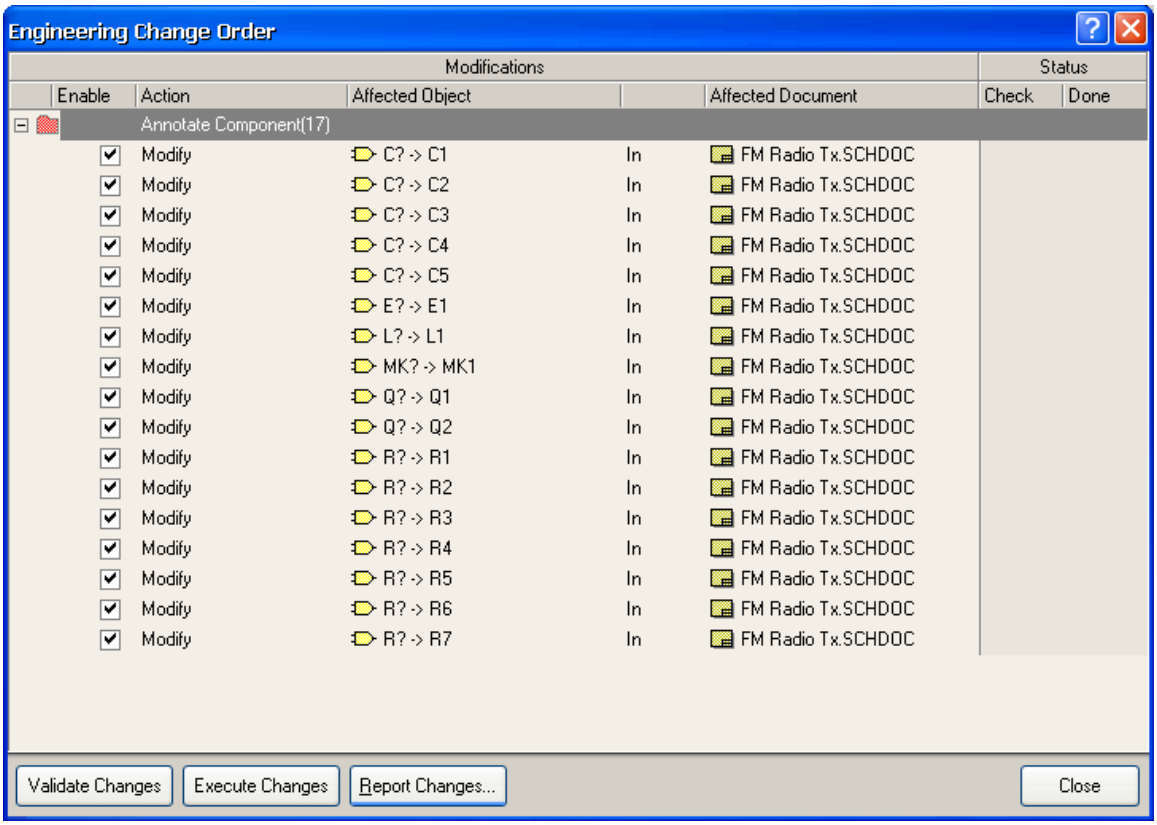

Save [PCB Project1.PrjPCB] As...
Save in: PCBBoards
AM Radio Tx
Card
Dice
ElectronicsII-05
History
Desktop
My Documents
My Computer
My Network P...
File name: FM Radio Tx PCB Project
Save as type: PCB Projects (*.PrjPCB)
Save
Cancel
Help

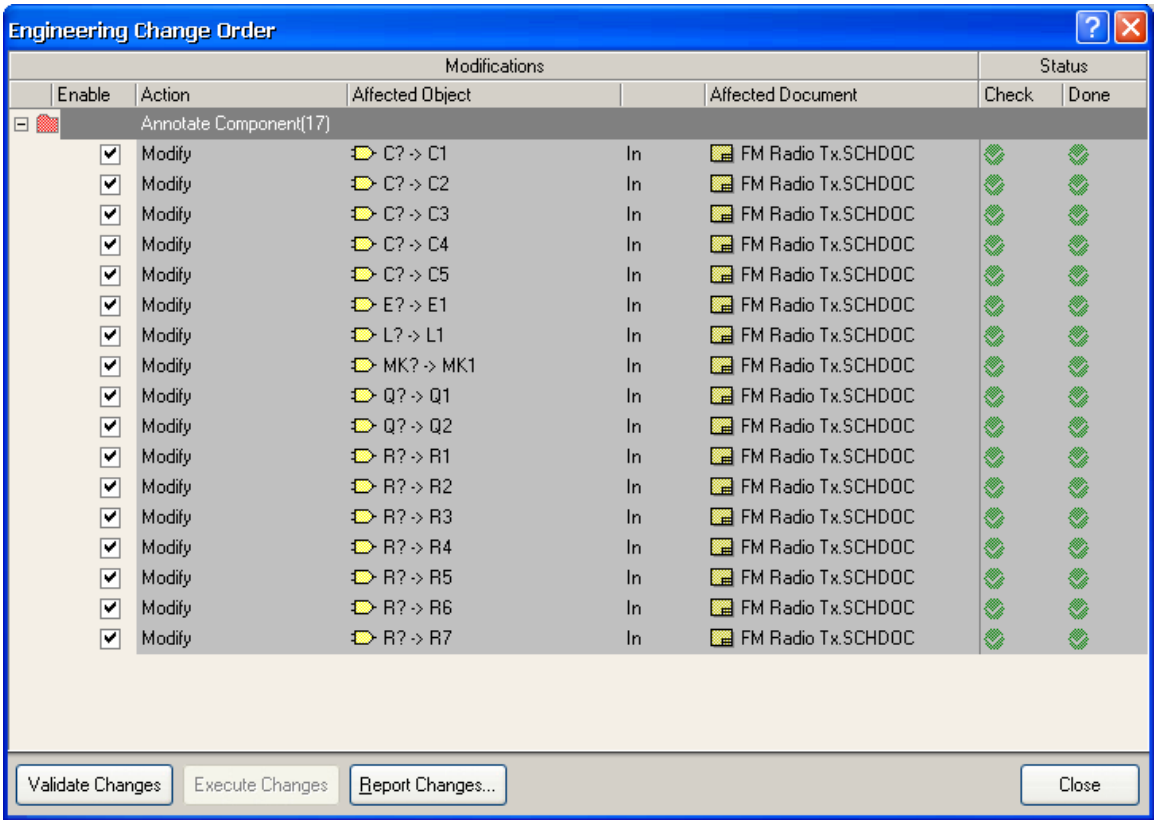

Save [ProjectGroup1.PrjGrp] As...
Save in: PCBBoards
AM Radio Tx
Card
Dice
ElectronicsII-05
History
Desktop
My Documents
My Computer
My Network P...
File name: FM Radio Tx ProjectGroup
Save as type: Project Groups (*.PrjGrp)
Save
Cancel
Help

- 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.



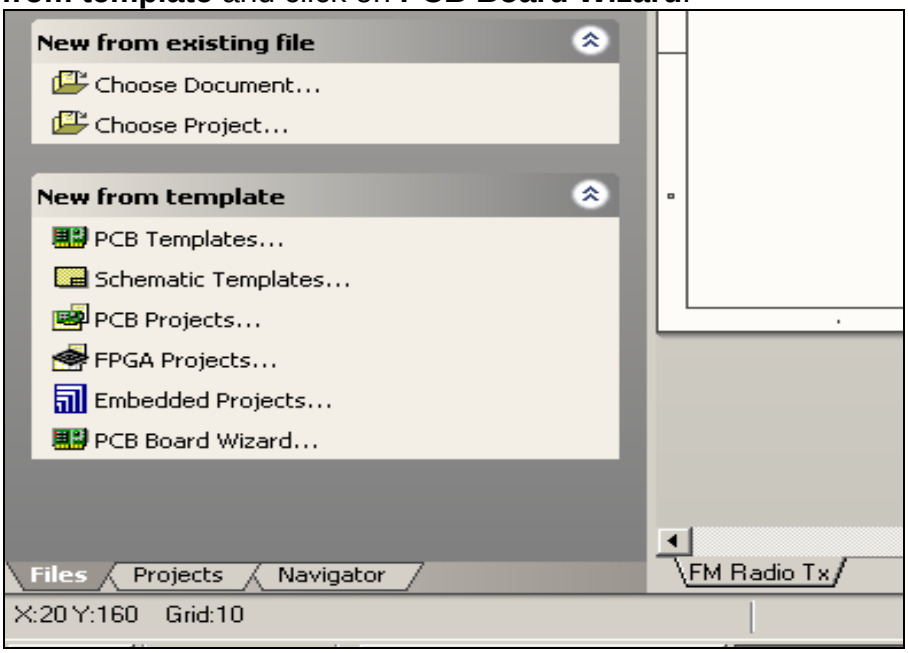

Annotate
Schematic Annotation Configuration
1 Up then across
2 Down then across
3 Across then up
4 Across then down
R1
R2
R3
R4
Match By Parameters
Class I
Class II
Comment
Component Kind
Datasheet
Description
Footprint
Schematic Sheets To Annotate
Schematic Sheet
FM Radio Tx.SCHDOC
Designator Index Control
Start Index
1
Add Suffix
Suffix
Proposed Change List
Current
Designator
Sub
Proposed
Designator
Sub
Location of Part
Schematic Sheet
C? C? FM Radio Tx.SCHDOC
E? E? FM Radio Tx.SCHDOC
L? L? FM Radio Tx.SCHDOC
MK? MK? FM Radio Tx.SCHDOC
Q? Q? FM Radio Tx.SCHDOC
R? R? FM Radio Tx.SCHDOC
All On
All Off
Back Annotate...
Update Changes List
Reset Designators
Accept Changes (Create ECO)
Close


Engineering Change Order
Modifications
Status
Enable
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
Validate Changes
Execute Changes
Report Changes...
Close


Engineering Change Order
Modifications
Status
Enable
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
Validate Changes
Execute Changes
Report Changes...
Close

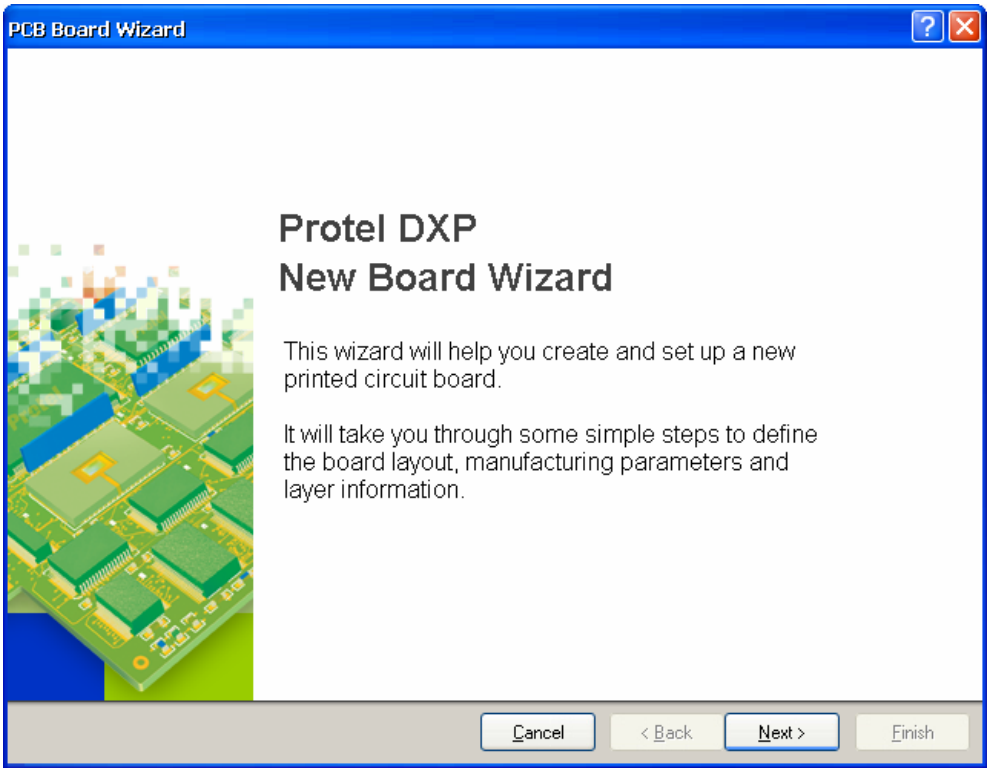
After you have Annotated the components, we want to create the PCB board

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

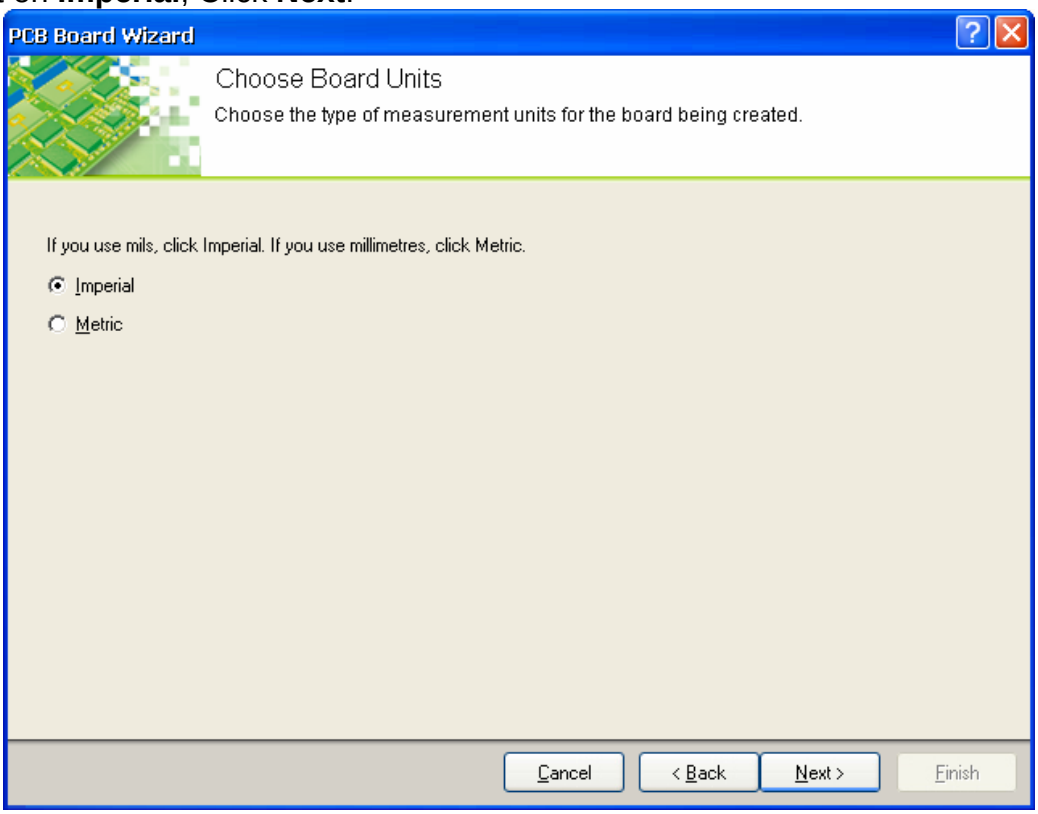



It should bring up the PCB Board Wizard window.

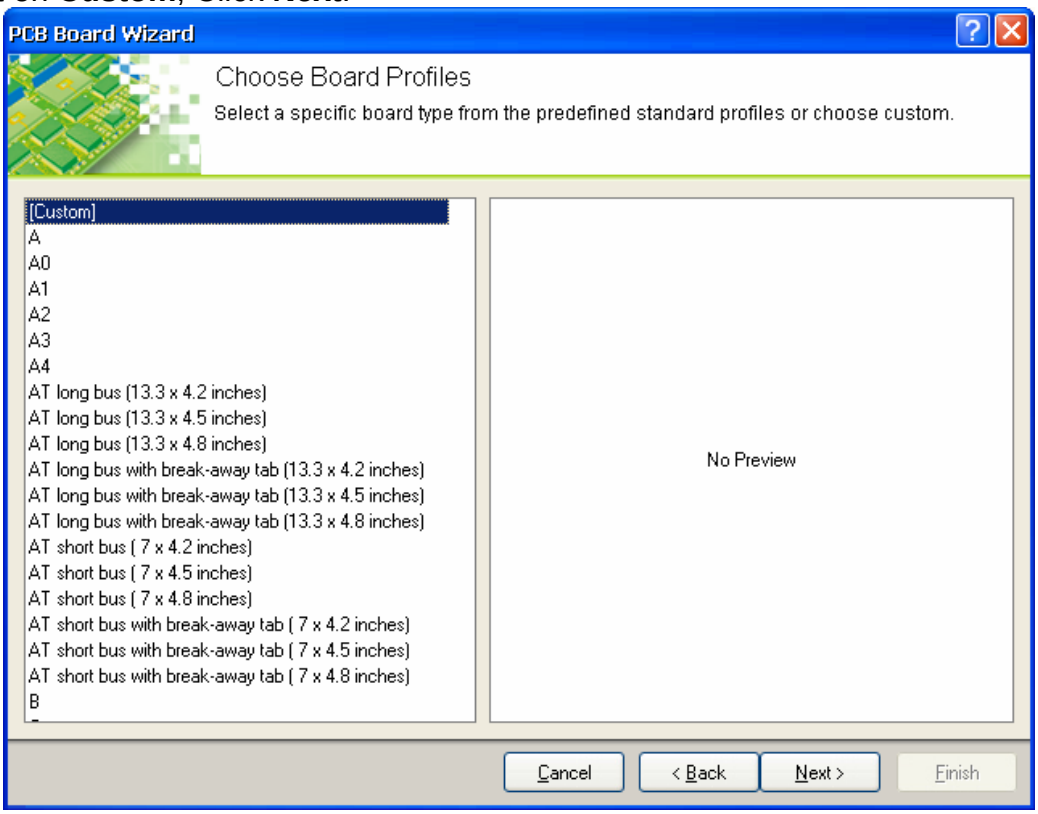
- Click **Next**.



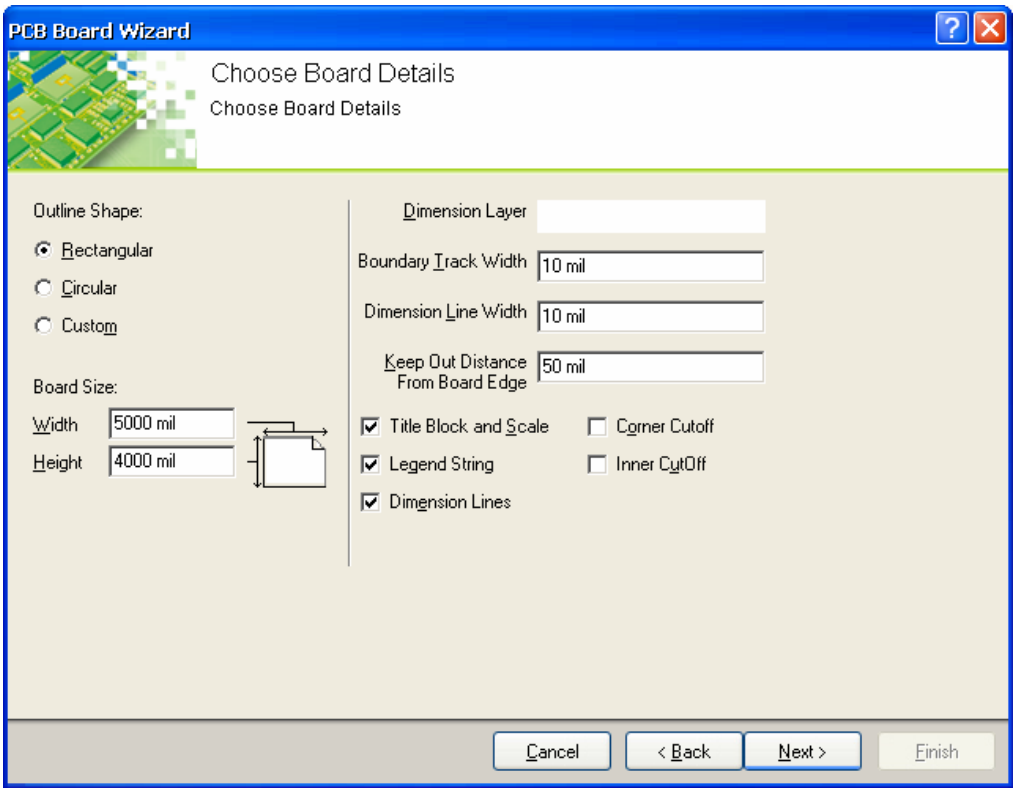
- Keep it on **Imperial**, Click **Next**.



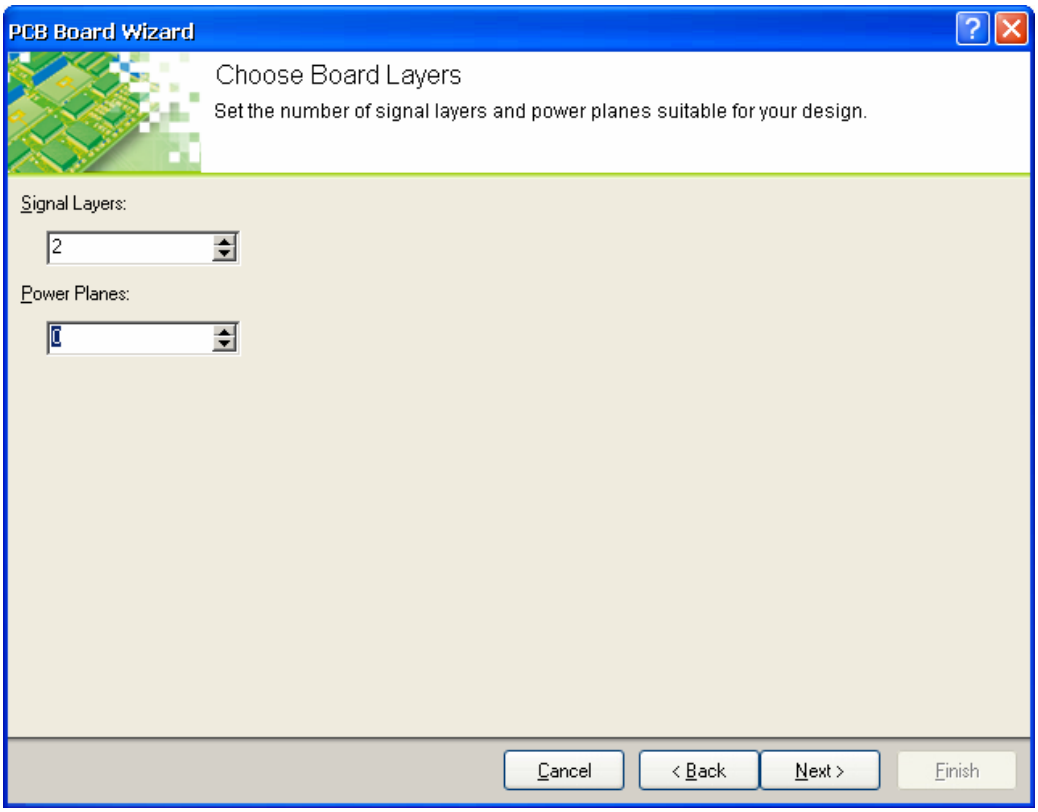

- Keep it on **Custom**, Click **Next**.



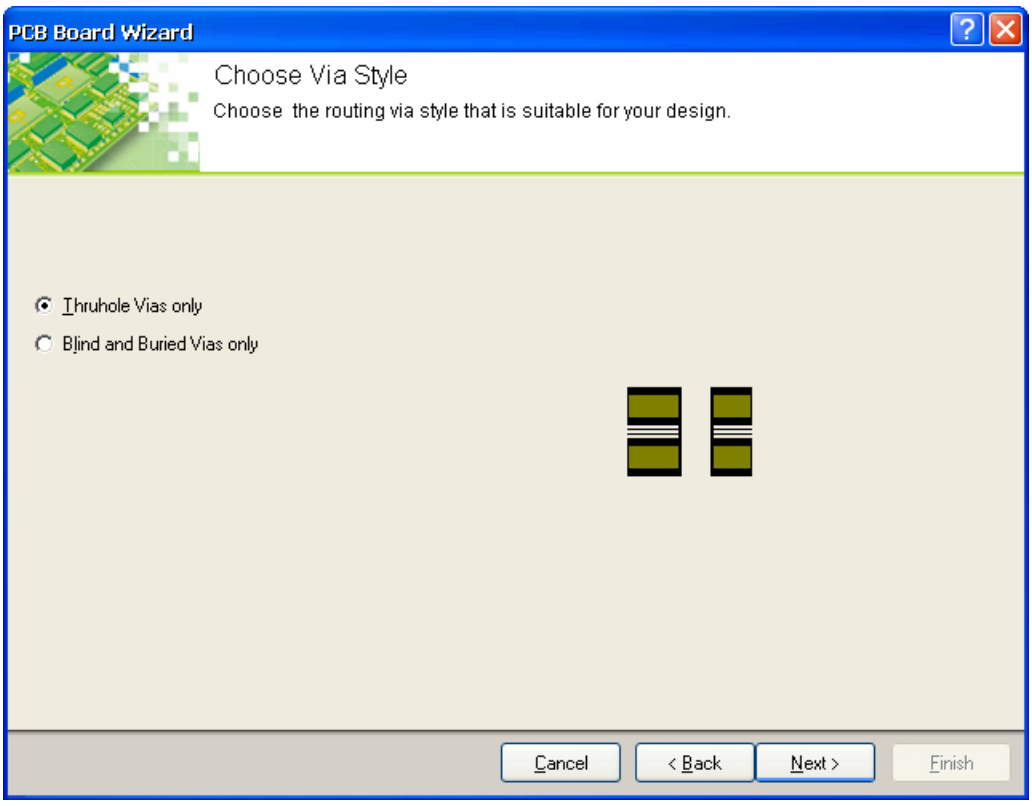
- On the **Board Size**, change the **Width** to 3000mil and the **Height** to 3000mil (Note: 1000mil = 1 inch).
- Then click **Next**.



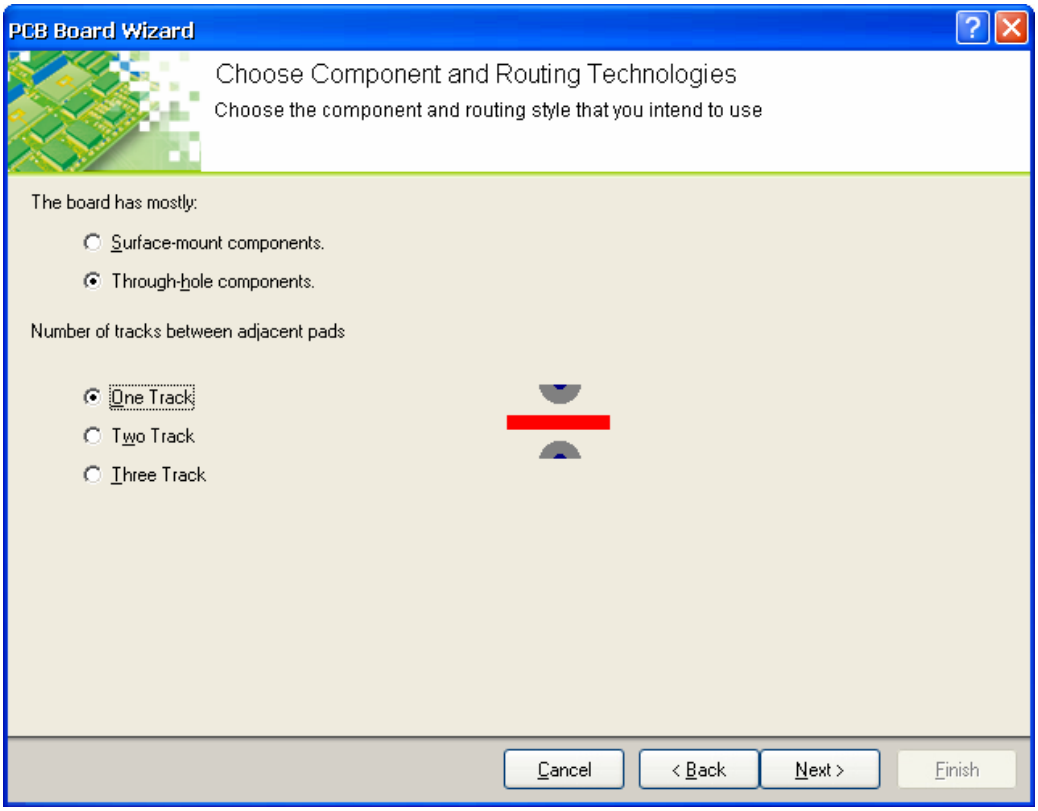

- Put the value of “0” for the number of **Power Planes**.
- Click **Next**.



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

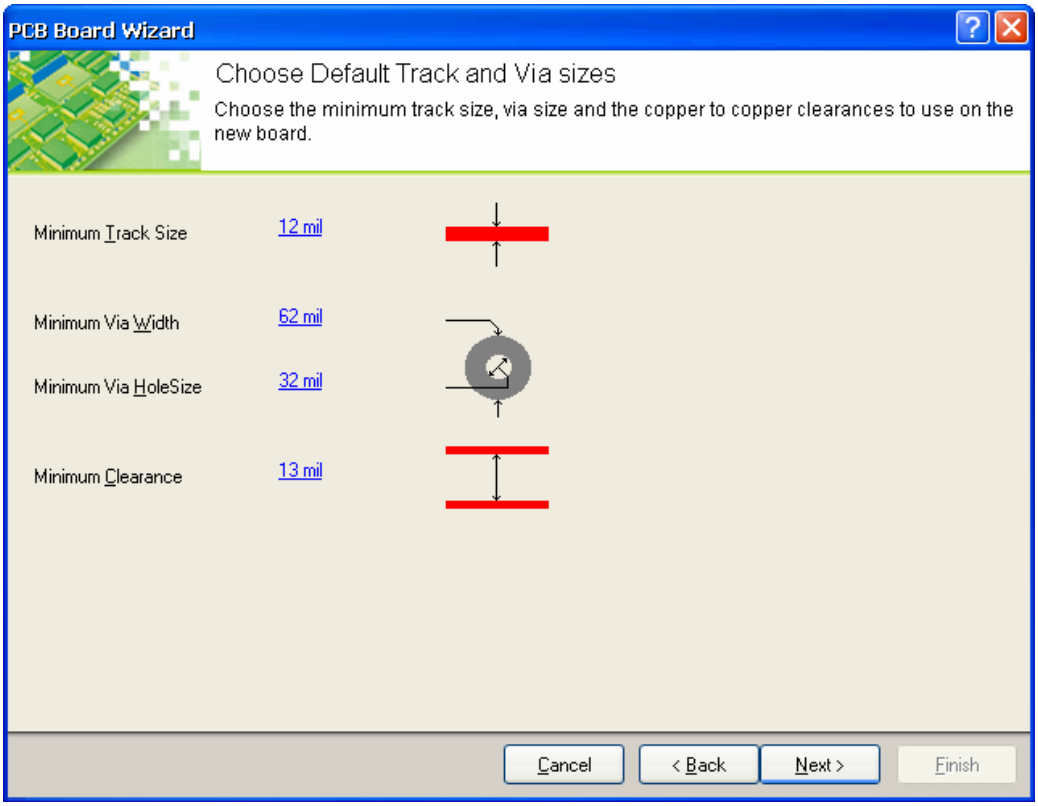



- For the **"The board has mostly:"** select **Through-hole components**.
- For the **"Number of tracks between adjacent pads,"** select **One Track**.
- Click **Next.**

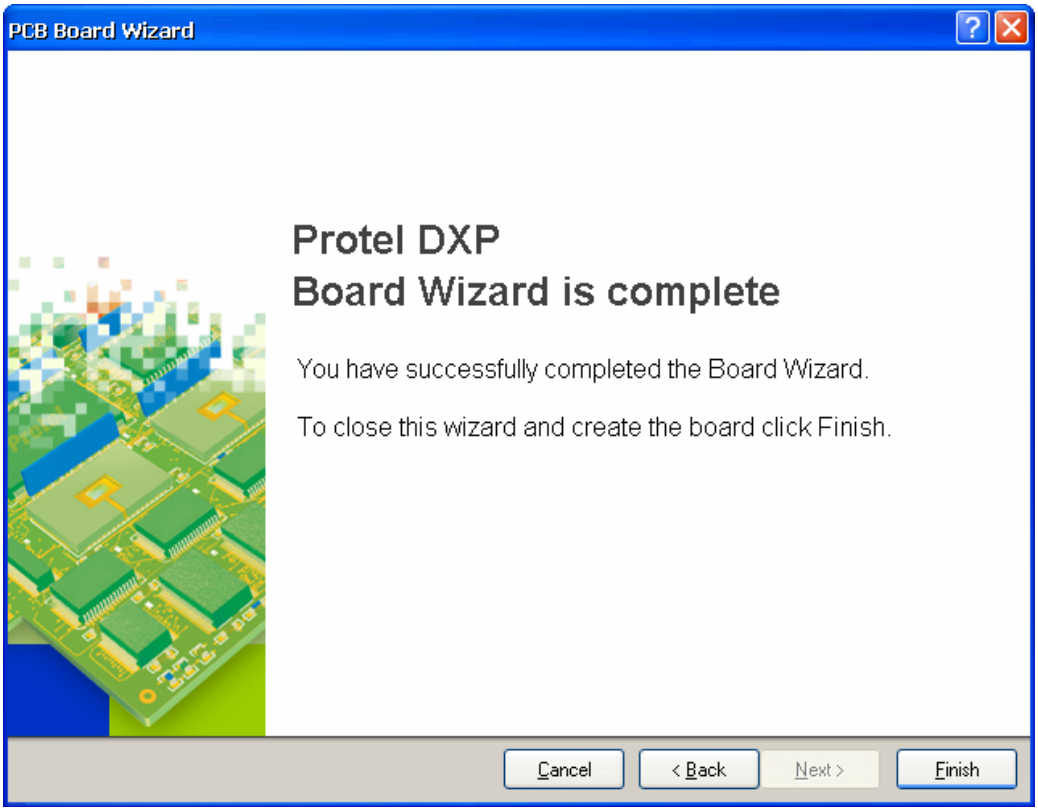


No changes needed on this window.

- Click **Next**.




- Finally click **Finish**.

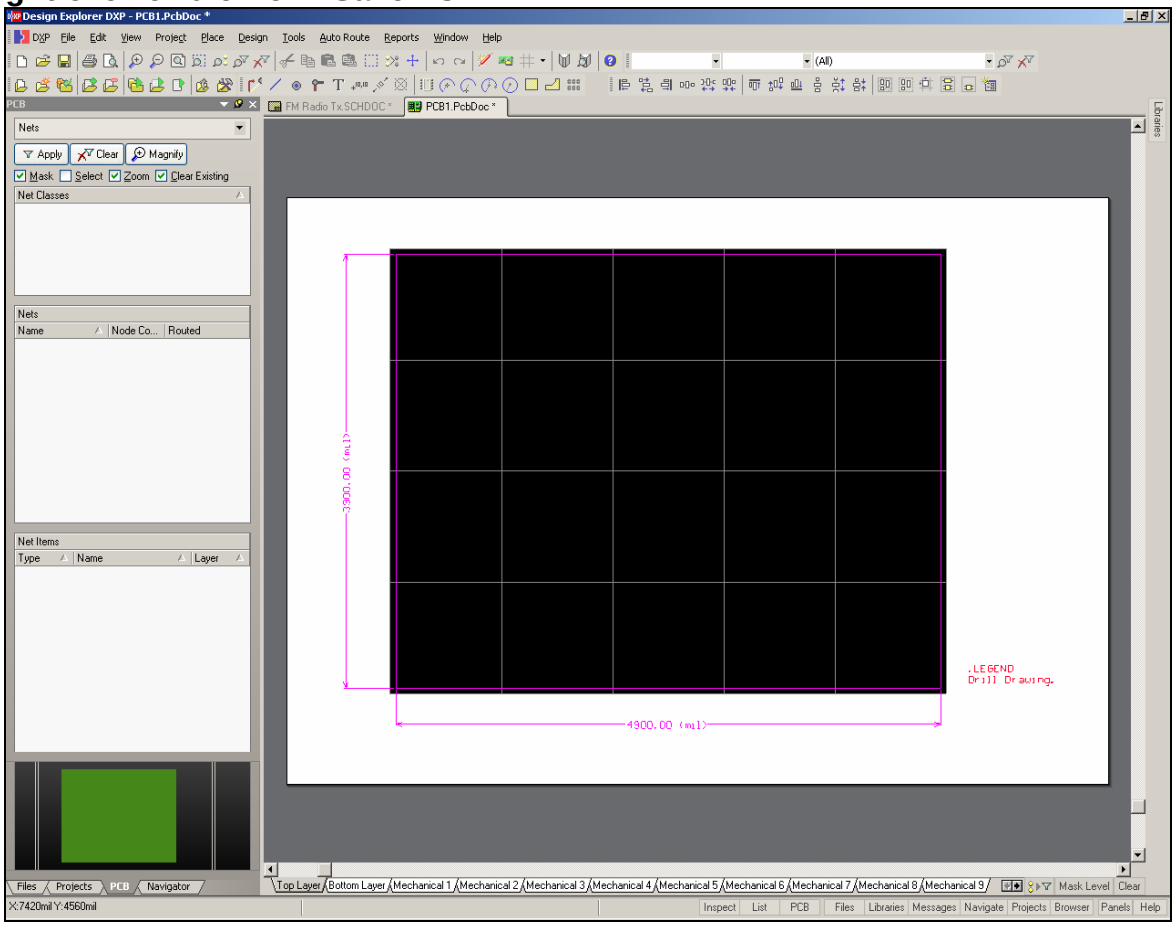


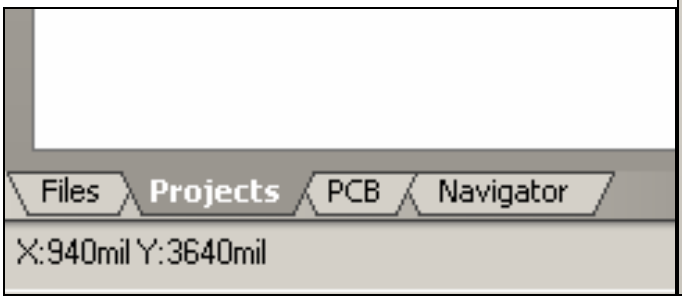
Now you should get this window to show up.

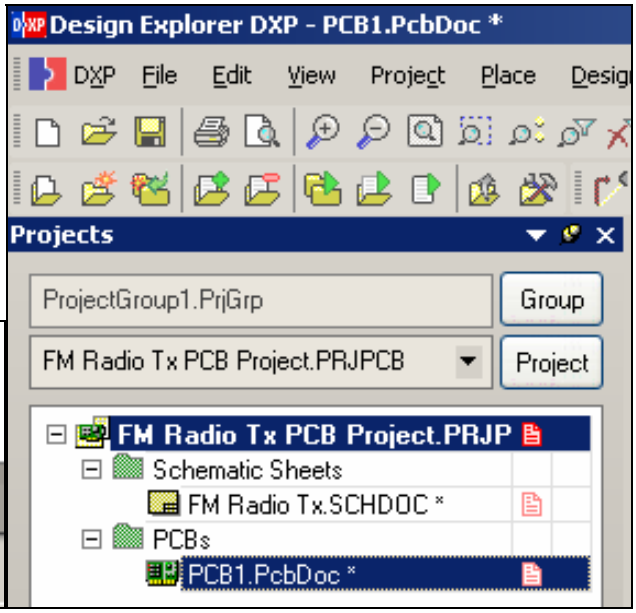
- Go to the bottom left of the window, and select the **Projects** tab.

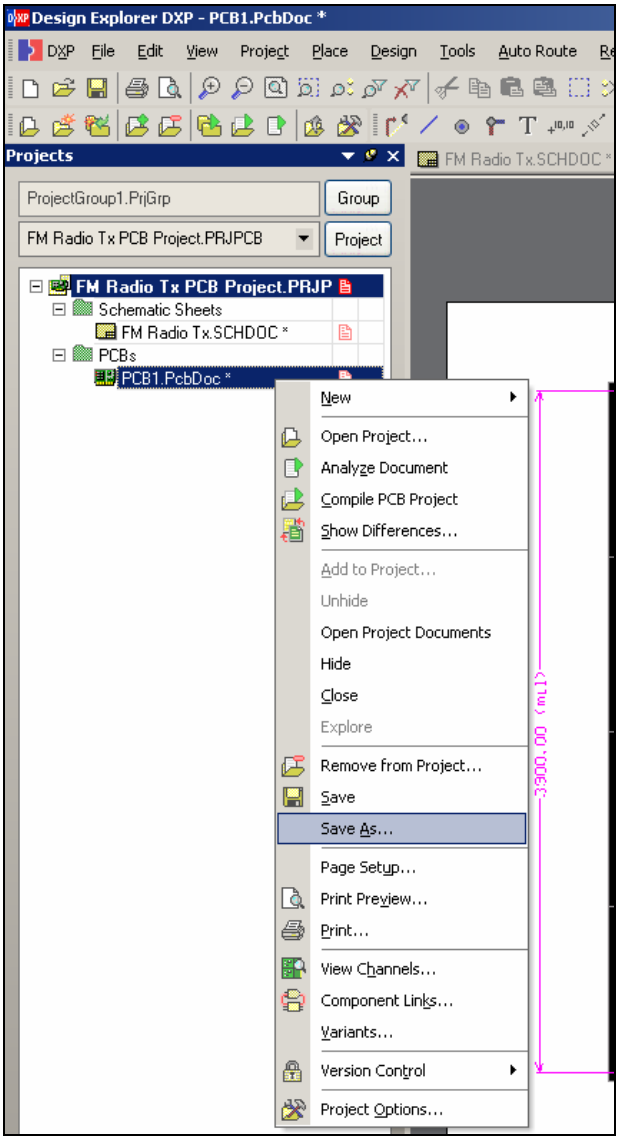
This allows you to select the new PCB file you created, so that you can save it.

- **Right-click** on the file -> **Save As**.



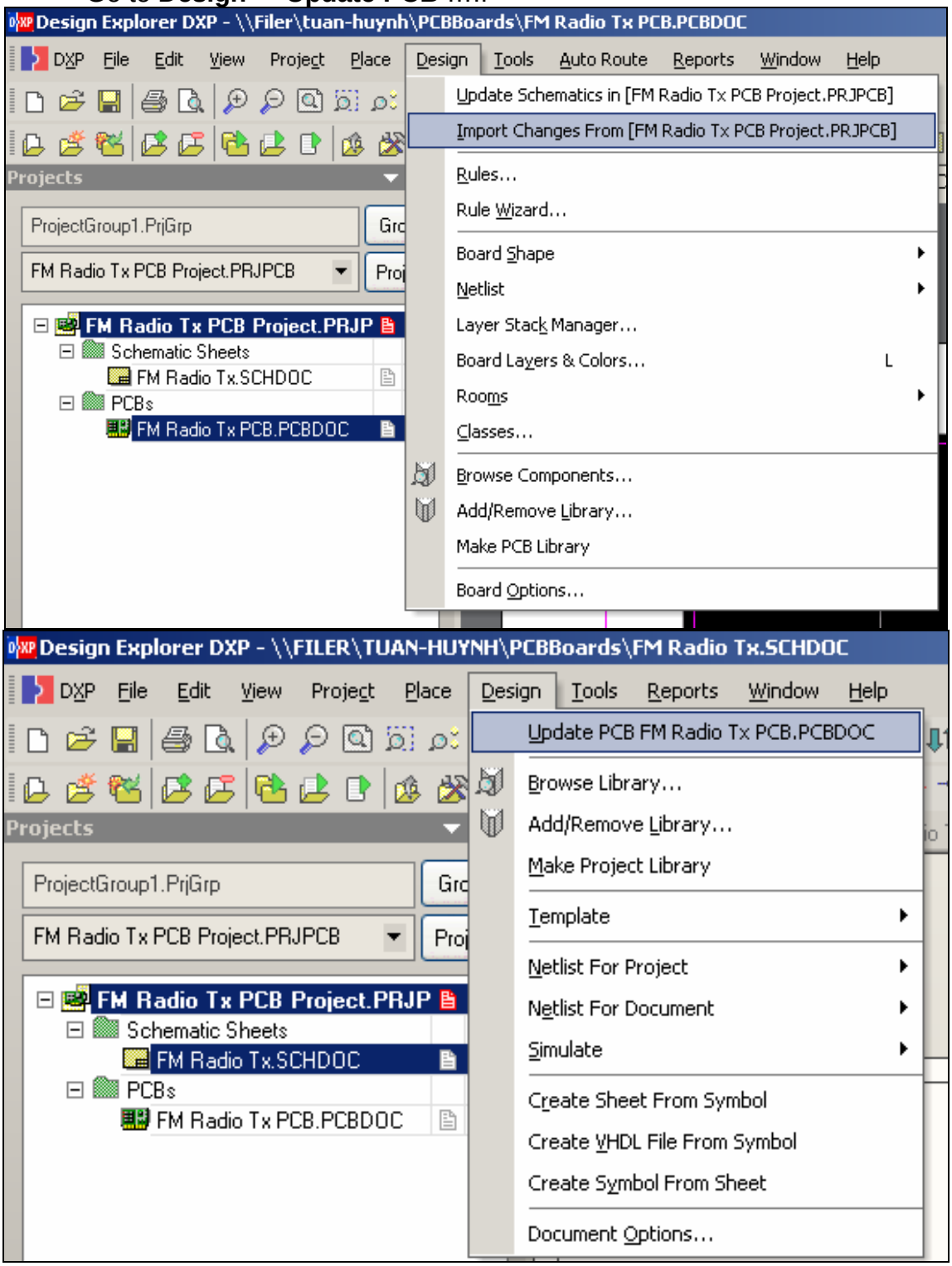




Design Explorer DXP - PCB1.PcbDoc *
DXP File Edit View Project Place Design Tools Auto Route
Projects
FM Radio Tx.SCHDOC *
ProjectGroup1.PrjGrp
Group
FM Radio Tx PCB Project.PRJPCB
Project
FM Radio Tx PCB Project.PRJP
Schematic Sheets
FM Radio Tx.SCHDOC *
PCBs
PCB1.PcbDoc *
New
Open Project...
Analyze Document
Compile PCB Project
Show Differences...
Add to Project...
Unhide
Open Project Documents
Hide
Close
Explore
Remove from Project...
Save
Save As...
Page Setup...
Print Preview...
Print...
View Channels...
Component Links...
Variants...
Version Control
Project Options...
3900.00 (mil)

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

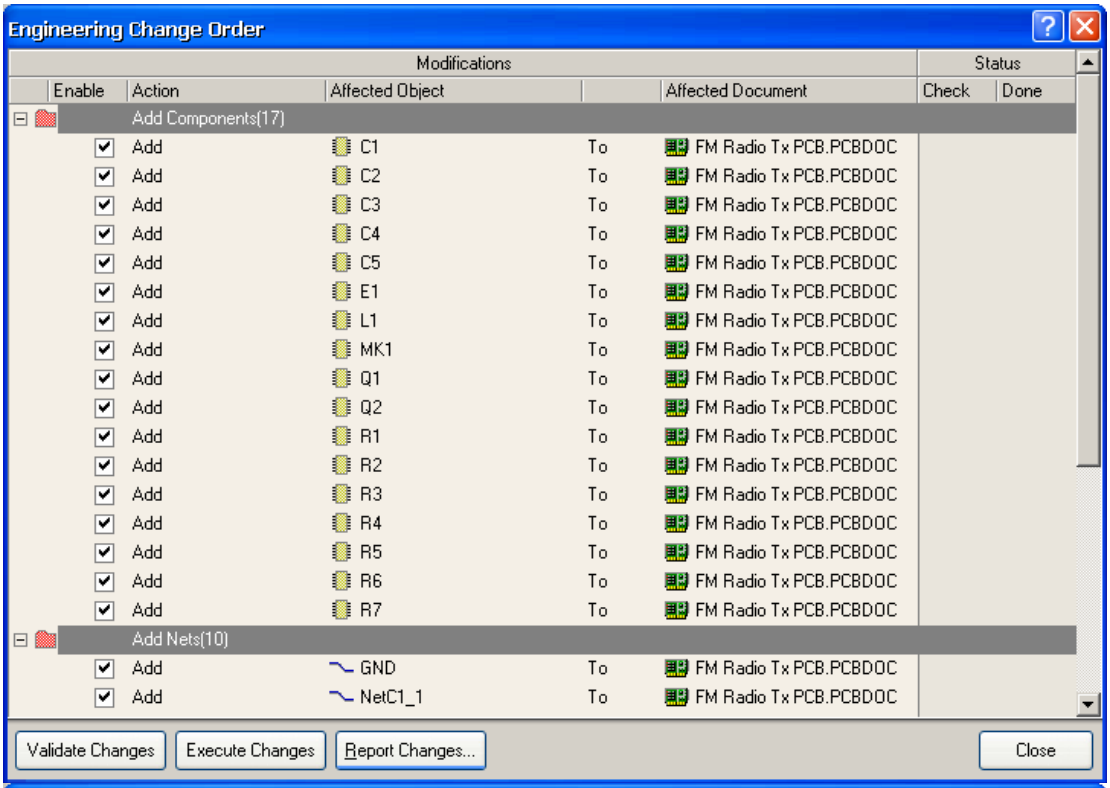
1) If you are on the **PCB** file
   - Go to **Design** -> **Import Changes From [….]**
2) If you are on the **Schematic** file
   - Go to **Design** -> **Update PCB** …..

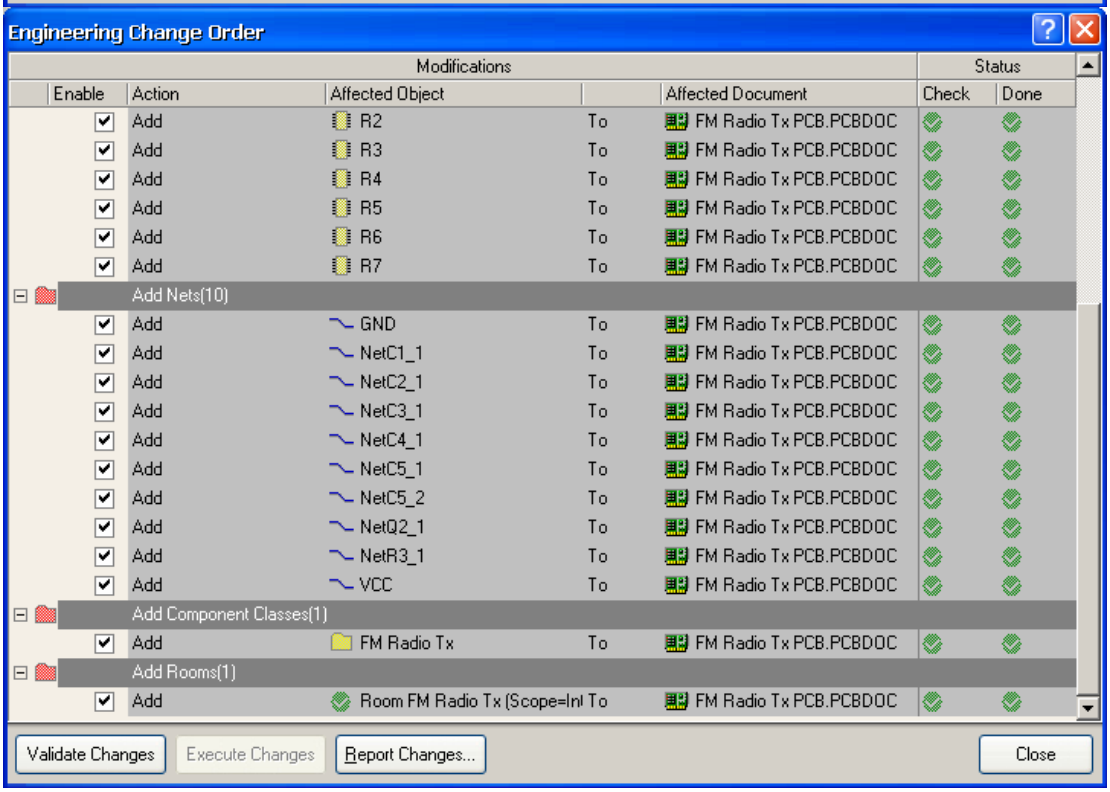


The **Engineering Change Order** window should pop up.

- 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.

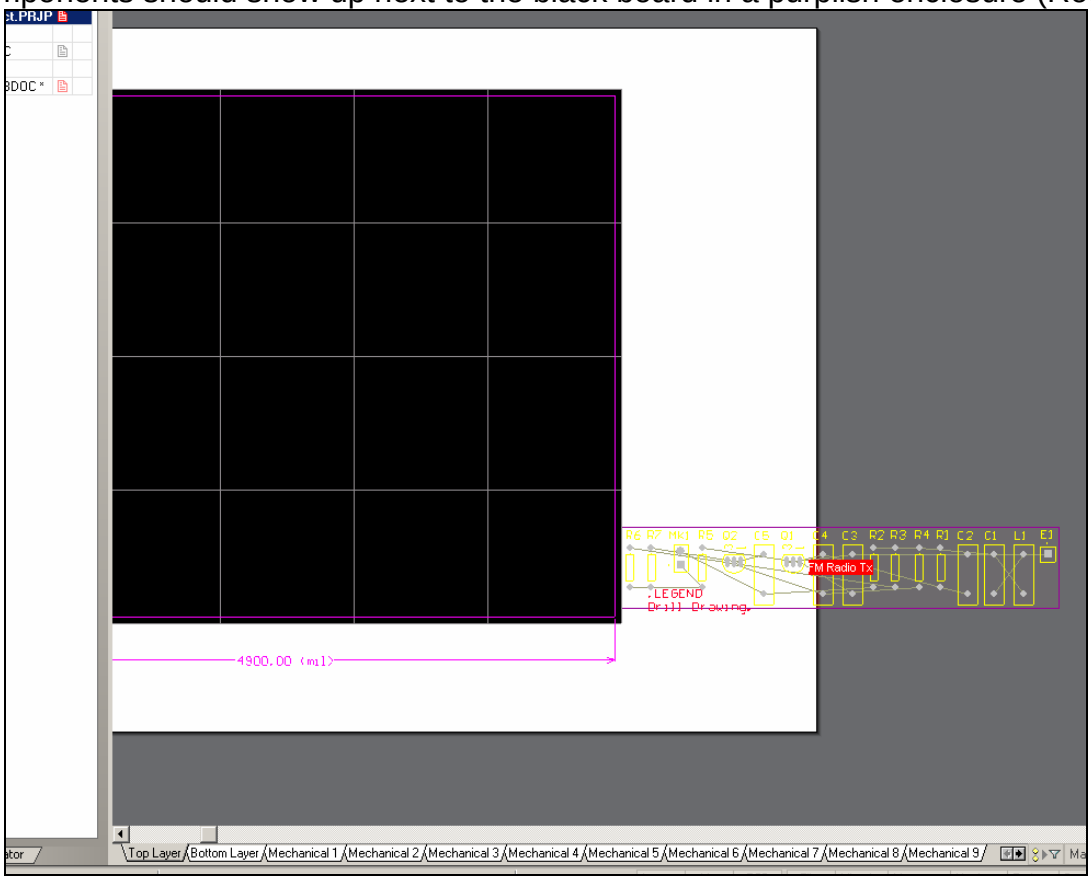


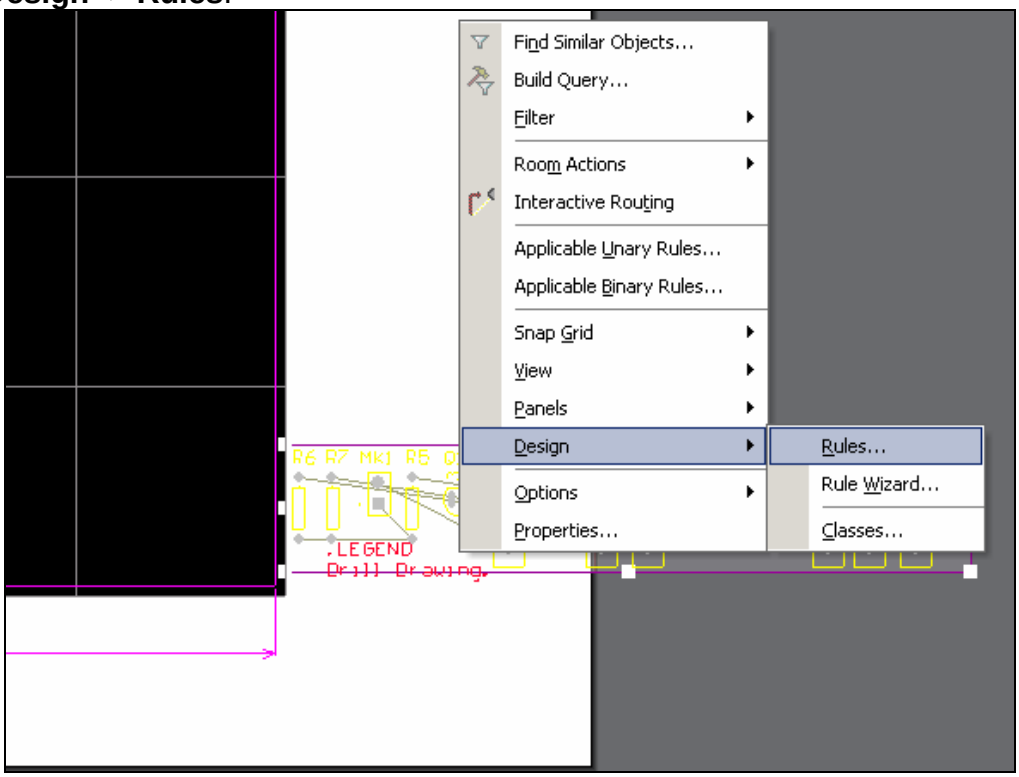
After you've verified that there are green check marks,

- Click **Close**

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



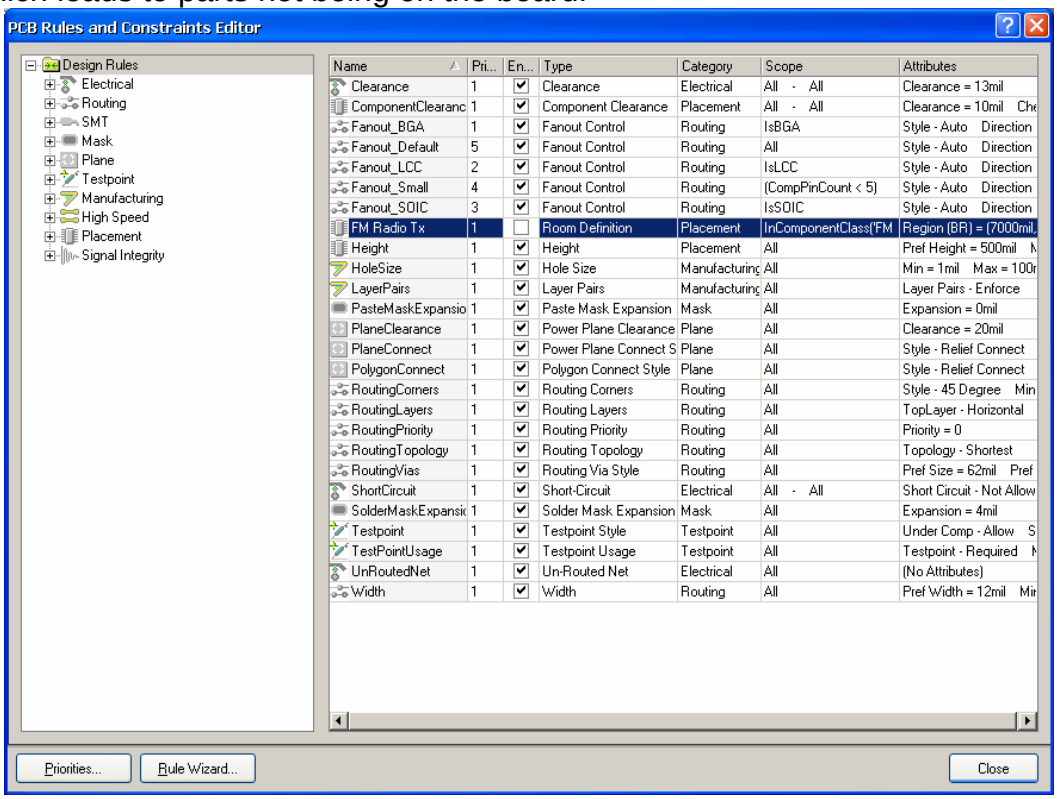

- Next, **Right-click** in the enclosure.
- Go to **Design** -> **Rules**.



The **PCB Rules and Constraints Editor** window should show up.

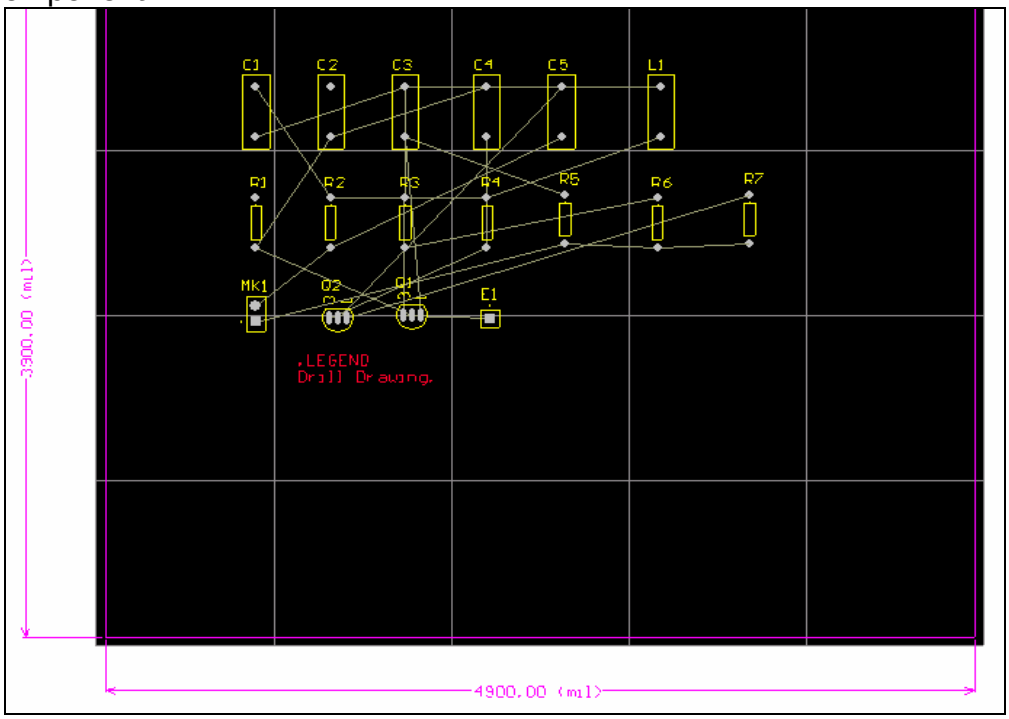
- Find **“Room Definition”** under the **Type** column and **Uncheck it**.

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.



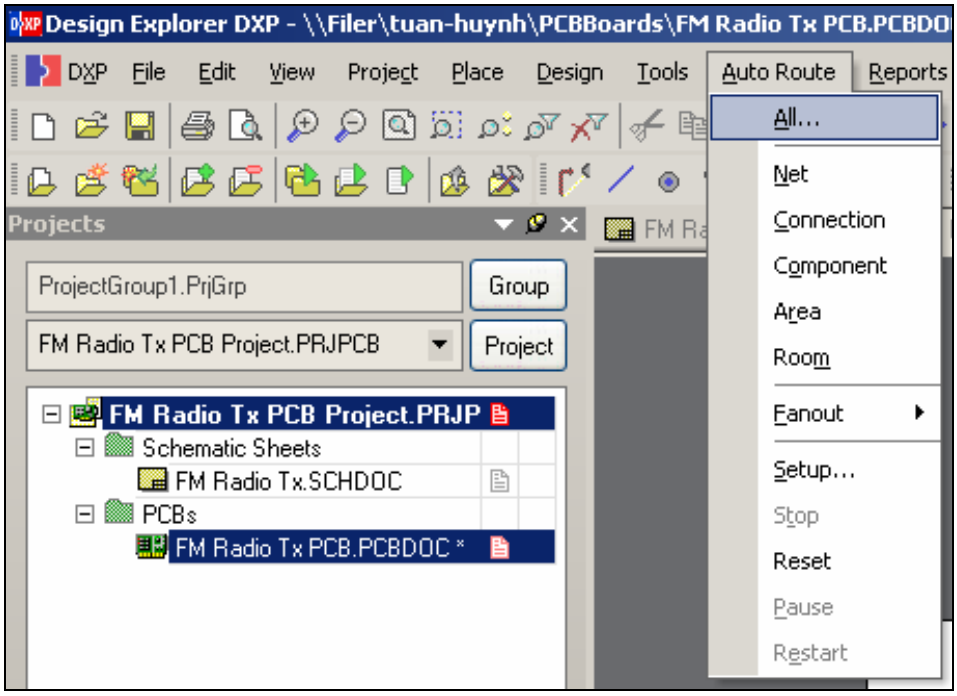

Place your components wherever you like within the black grid.

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.

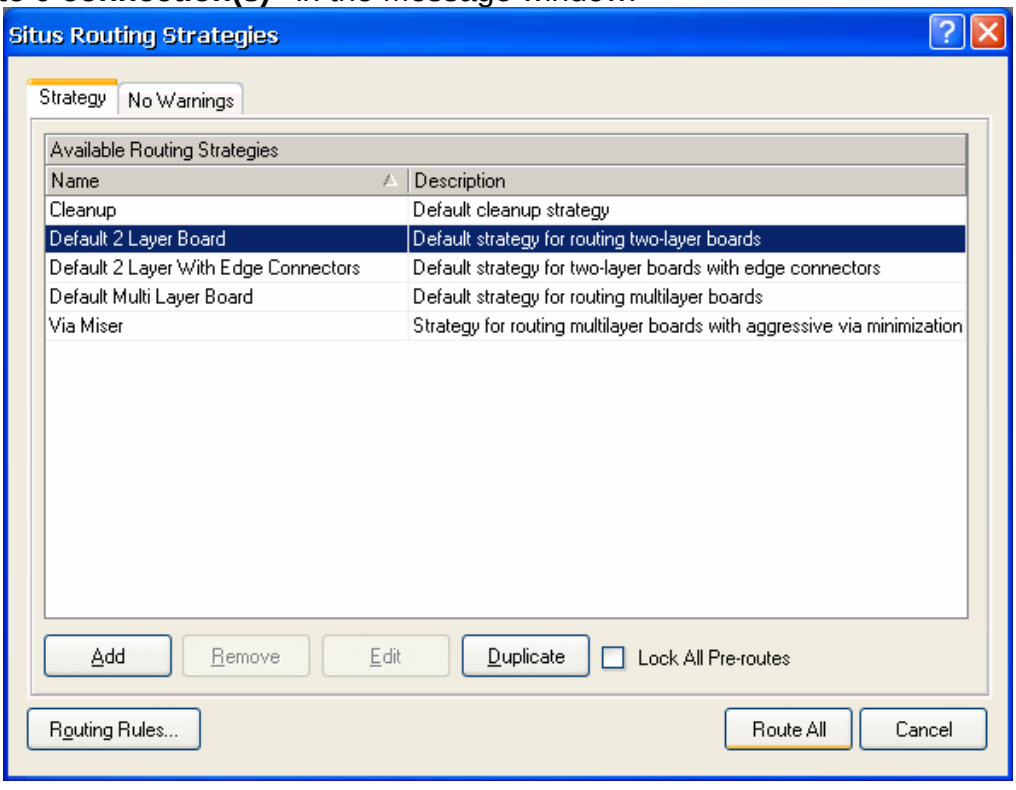
- Next, go to **Auto Route** -> **All**

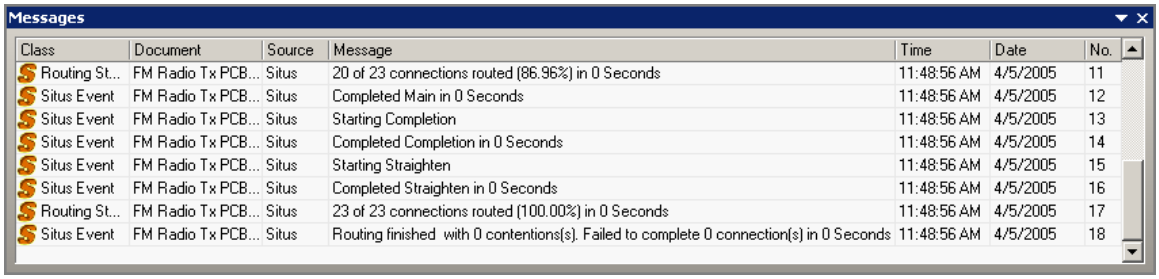



The Situs Routing Strategies window should show up.

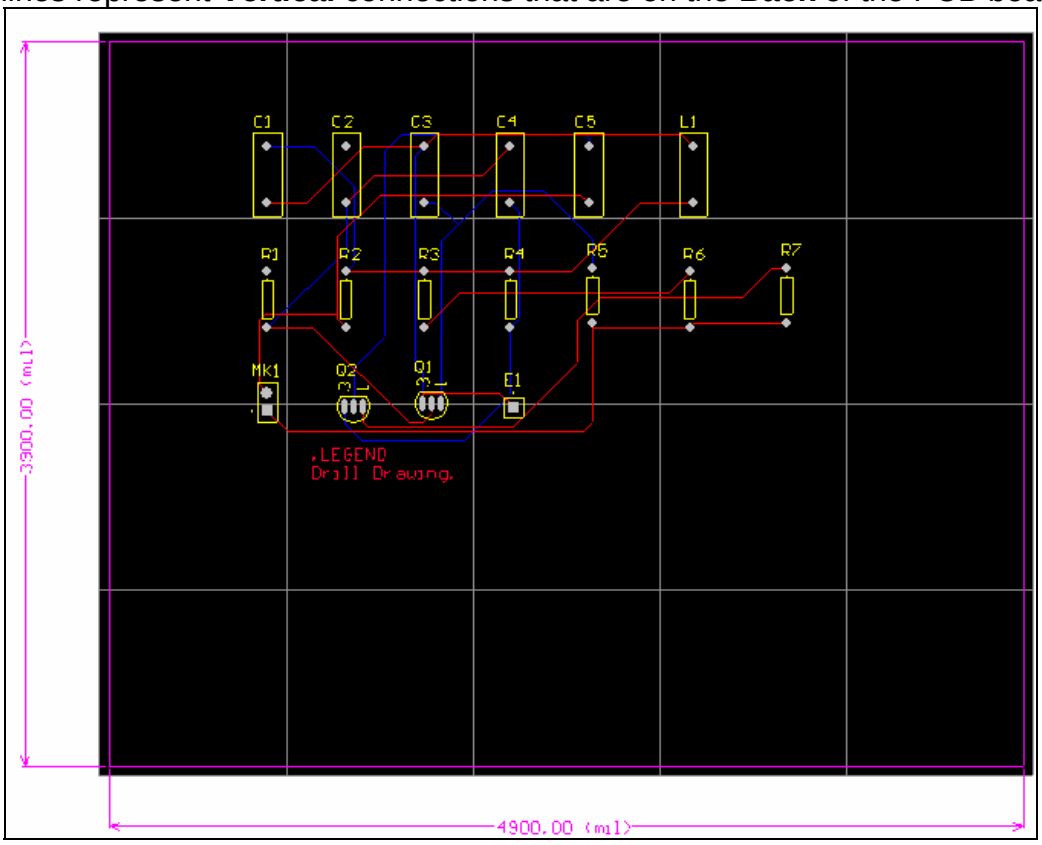
- Keep it like it is **"Default 2 Layer Board".**
- Click **Route All**.

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.



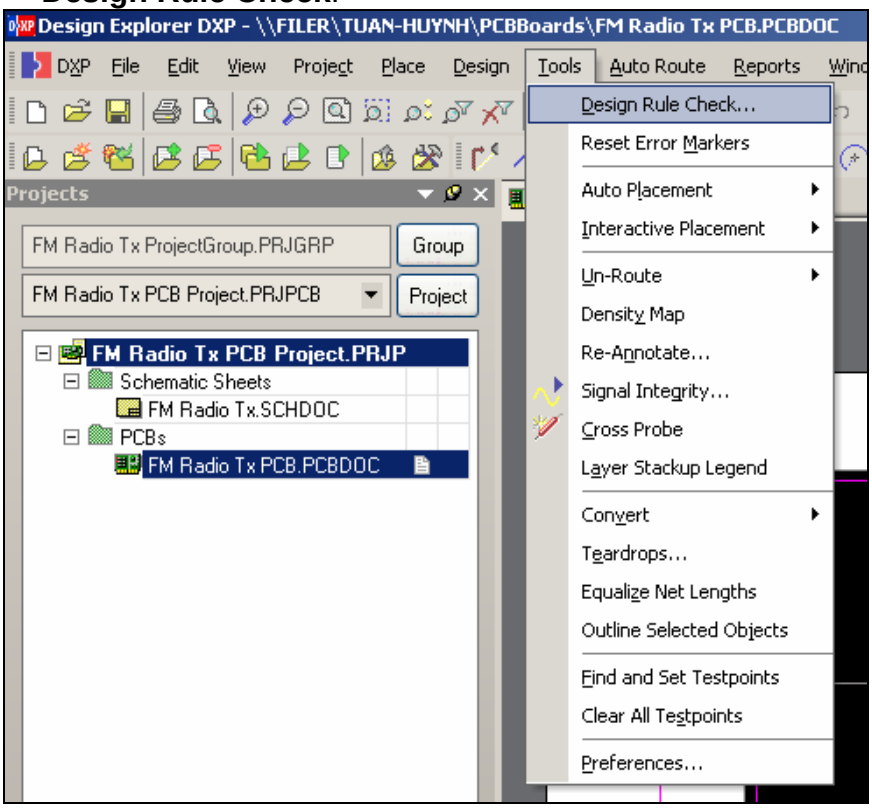



If you look at your board, all the white connections between the components have changed to red and blue lines.
The **Red** lines represent **Horizontal** connections that are on the **Front** of the PCB board.
The **Blue** lines represent **Vertical** connections that are on the **Back** of the PCB board.



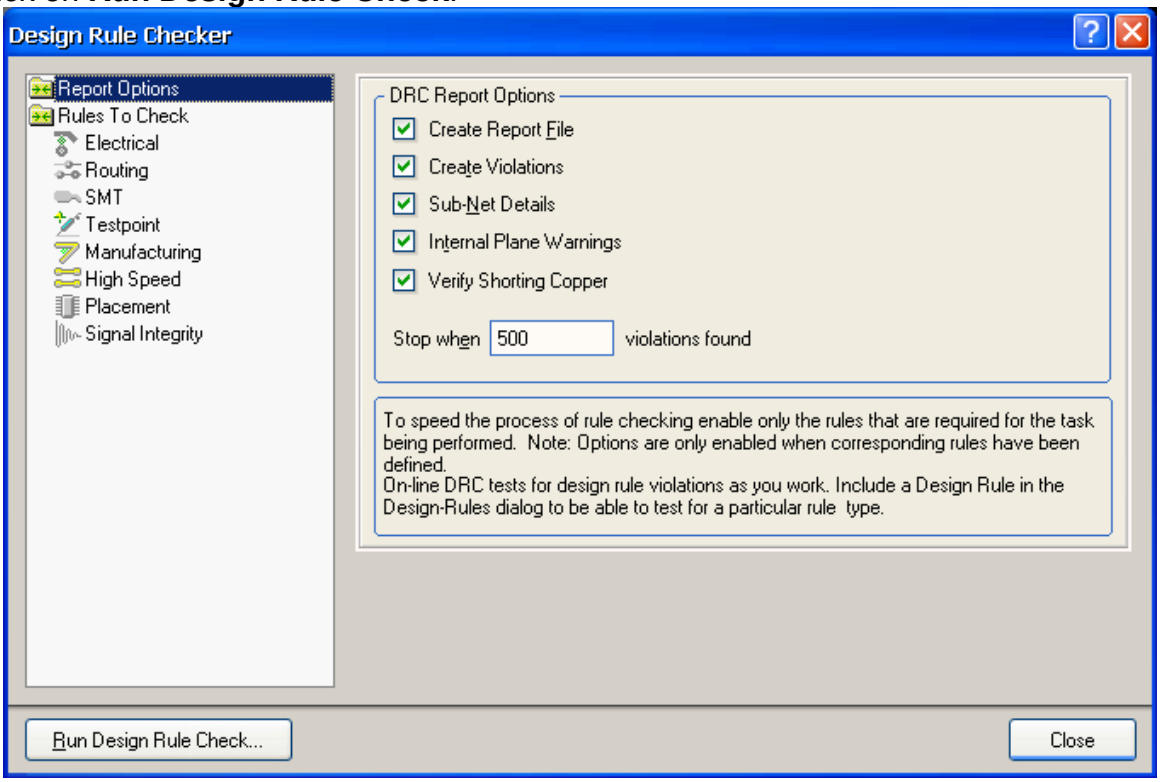
The next step is to run a Design Rule Check on the board to make sure it meets the requirements of the software to be a good board.

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



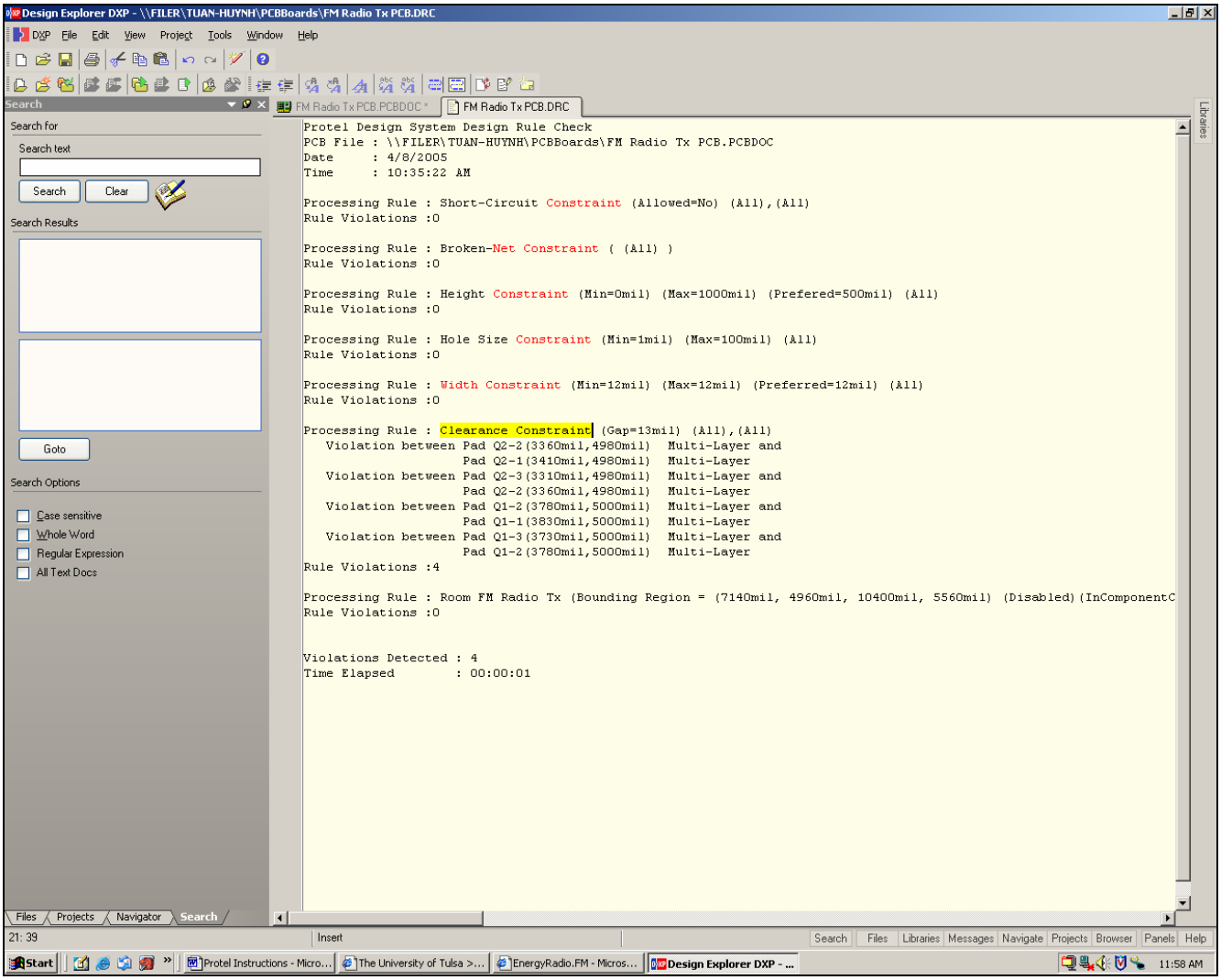

A Design Rule Checker window should pop up.
Keep everything like it is and

- Click on **Run Design Rule Check**.



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
-

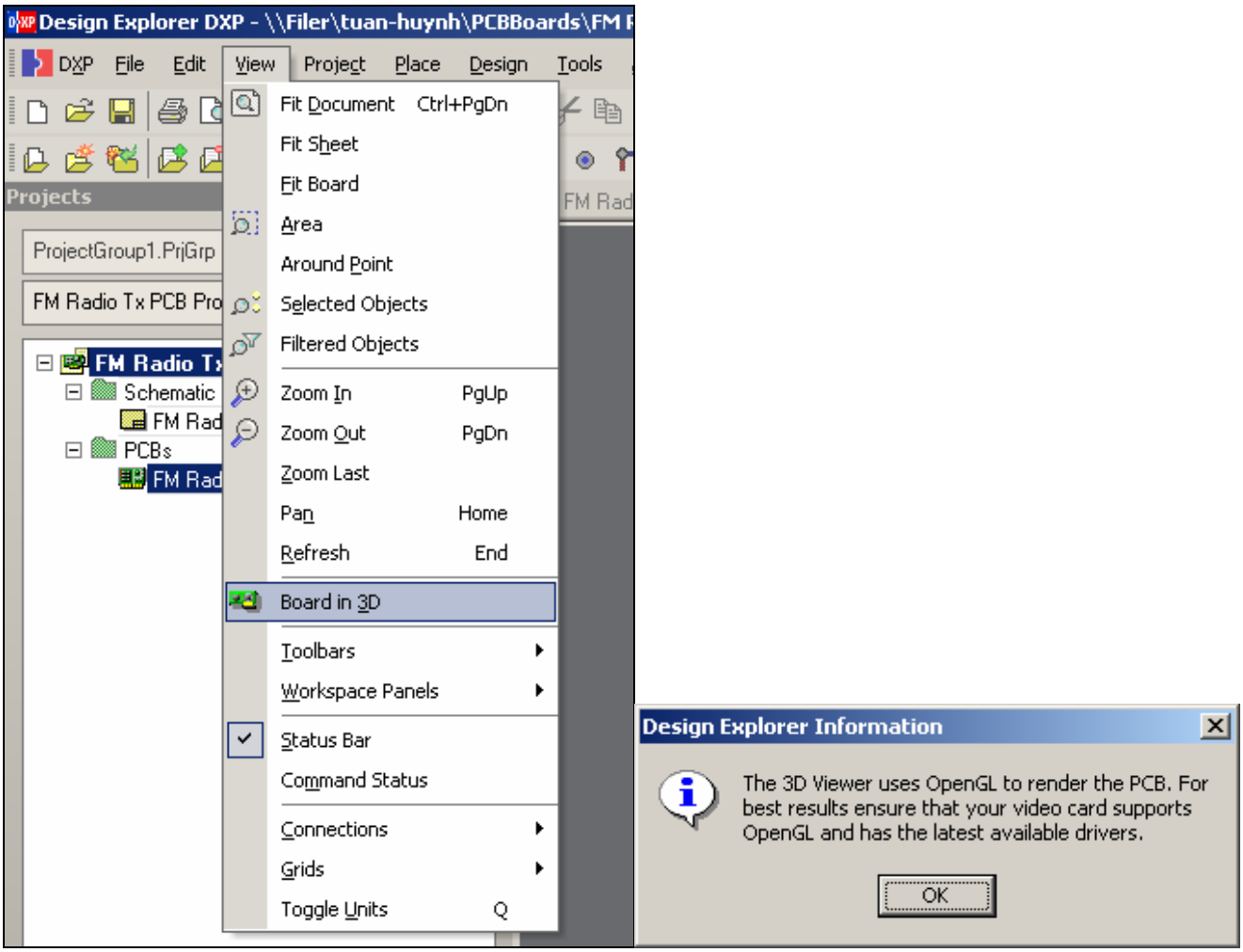


After you have verified that your board meets the Design Rules, your board is complete. There is an extra feature that allows you to view your board in 3D.
Note: The software guesses what your board may look like with components on it.

- Go to **View** -> **Board in 3D**.

A Design Explorer Information window will pop up.

- Click **OK**.



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.