

TUTORIAL SESSION 2014



Altium **Designer** 14

Technical Group

Hoda Najafi & Sunita Bhide

SETUP PROCEDURE

- Start the Altium Designer Software. ([Figure 1](#))
- Ensure that the 'Files' and 'Projects' tabs are located somewhere on the screen. Most likely they will be minimized on the left side of the window. If they are not then add them to the screen
 - a. View > Workspace Panels > System > Files
 - b. View > Workspace Panels > System > ProjectsAlternatively you may...([Figure 2](#))
 - c. Click on the 'System' box in the lower right hand corner of the designer and then click 'Files' and 'Projects' to make the tabs appear.

These will both accomplish the same goal of making the 'Files' tab and the 'Projects' tabs visible. All tabs can be dragged and dropped nearly anywhere on the designer to suit your personal preferences.

- Create a new PCB Project. File > New > Project > PCB Project – or – You may click on the 'Files' tab and then under the 'New' heading you can click 'Blank Project' ([Figure 3](#))

This will cause the 'Projects' tab to open and ask to set the name of the project. We may want to change the name and location of our project otherwise it will save as 'PCB1.PcbDoc' in default location. ([Figure 4](#))

- We will now add a schematic and a PCB to the current project.
 - a. Right click on the project > Add New to Project > Schematic ([Figure 5](#))
 - b. Right click on the project > Add New to Project > PCB ([Figure 6](#))

Finally, we should save the generated files:

- a. Right click on the name of each file>save ([Figure 7,8,9](#))

SETUP PROCEDURE

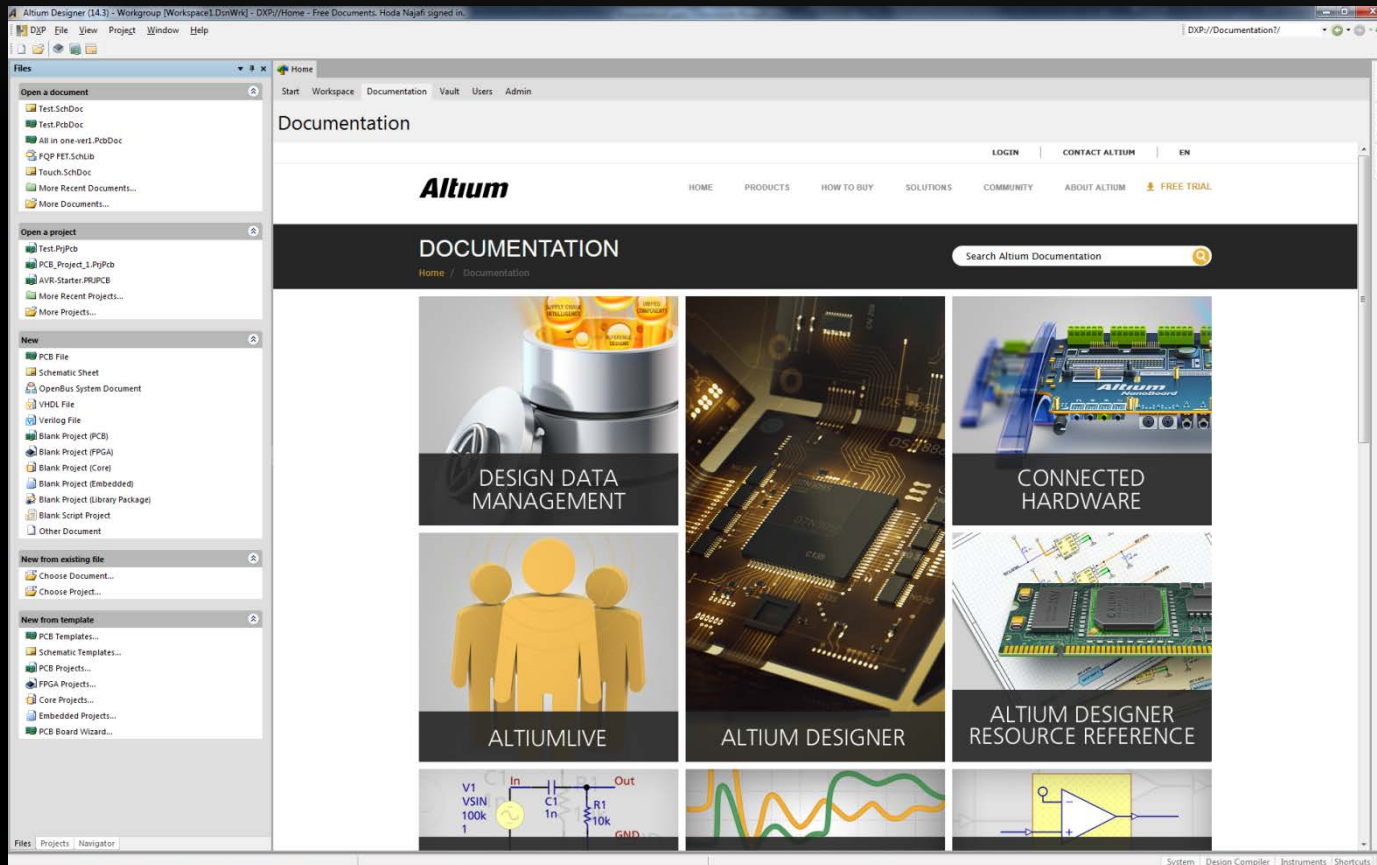


Figure 1

SETUP PROCEDURE

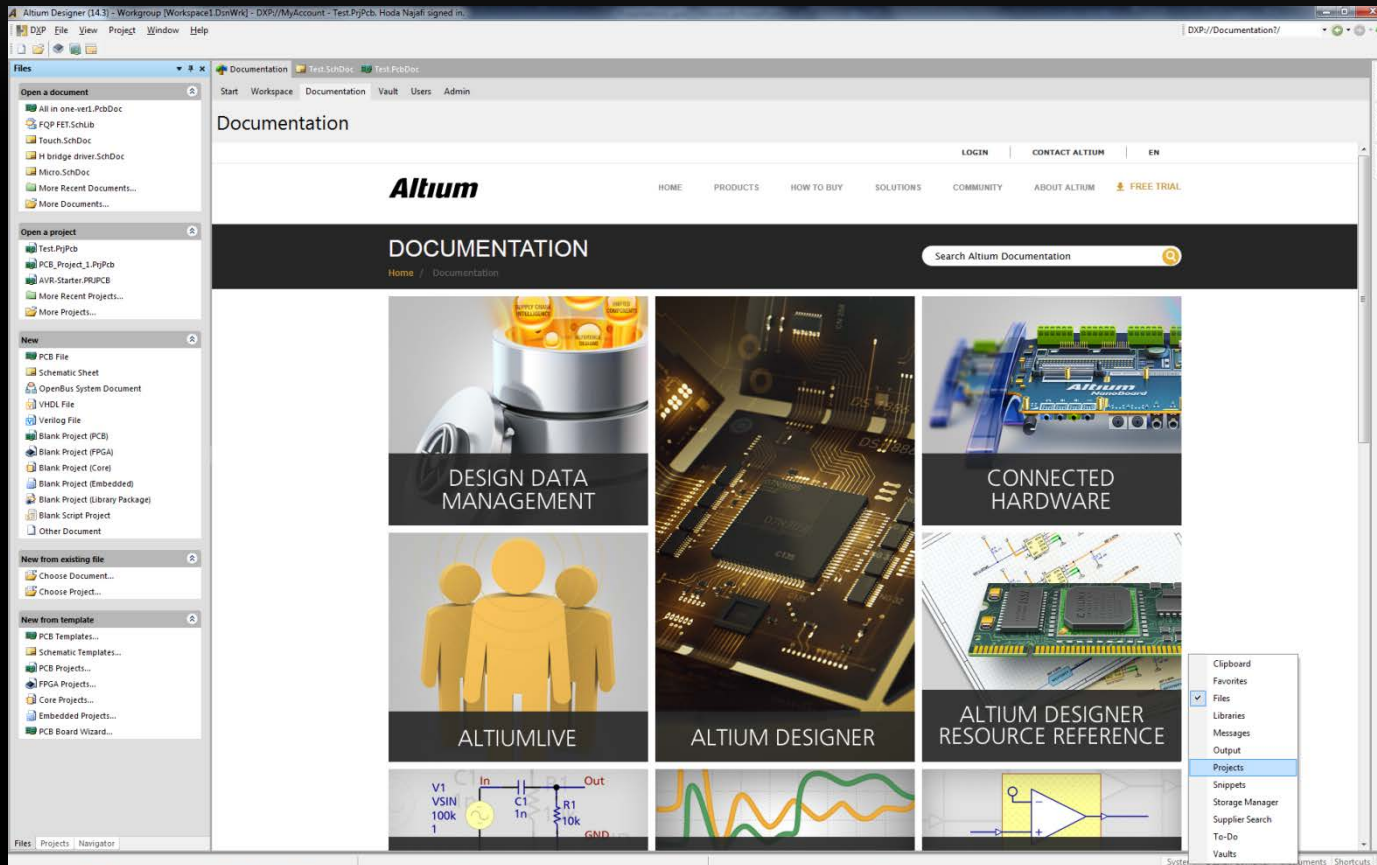


Figure 2

SETUP PROCEDURE

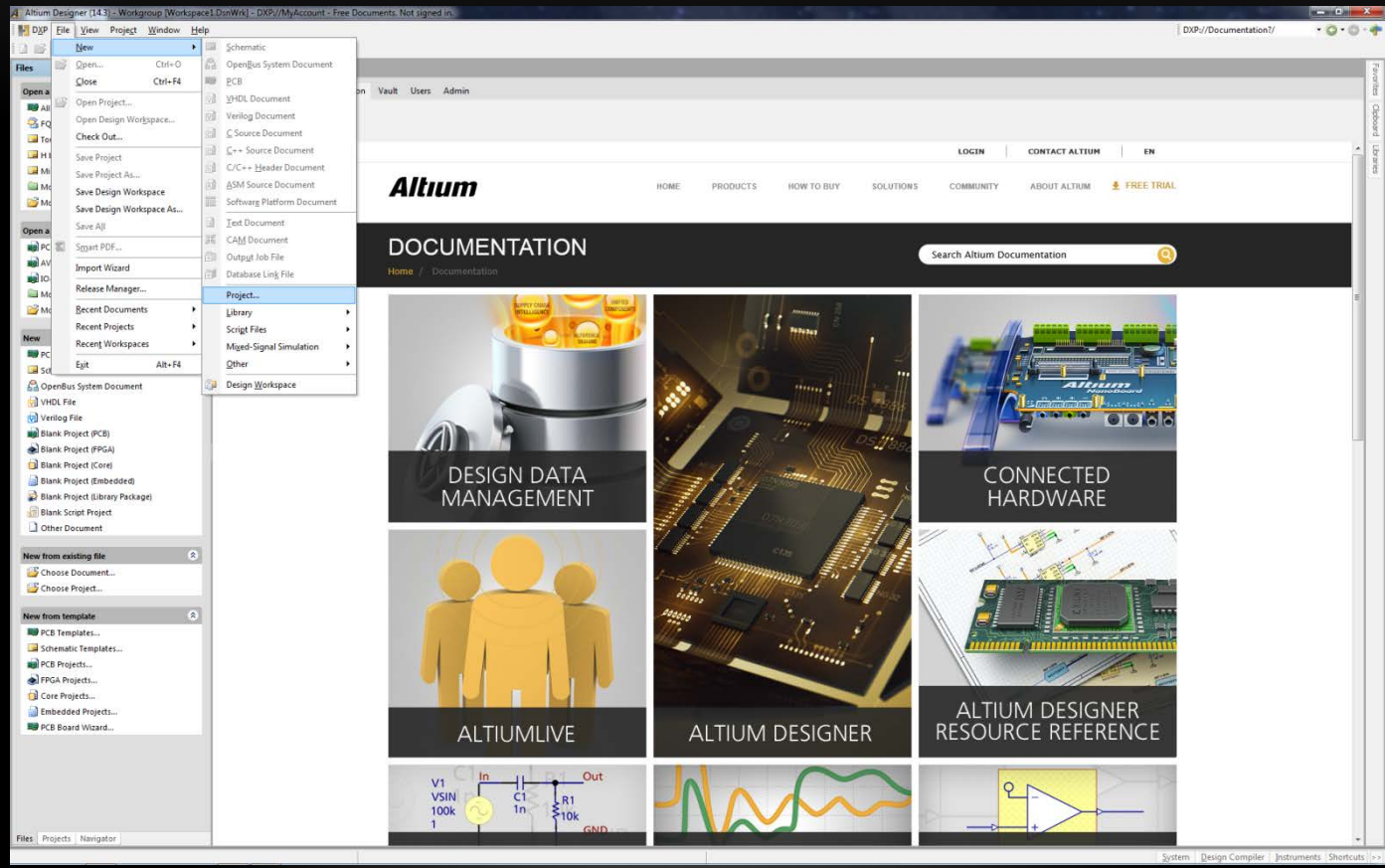


Figure 3

SETUP PROCEDURE

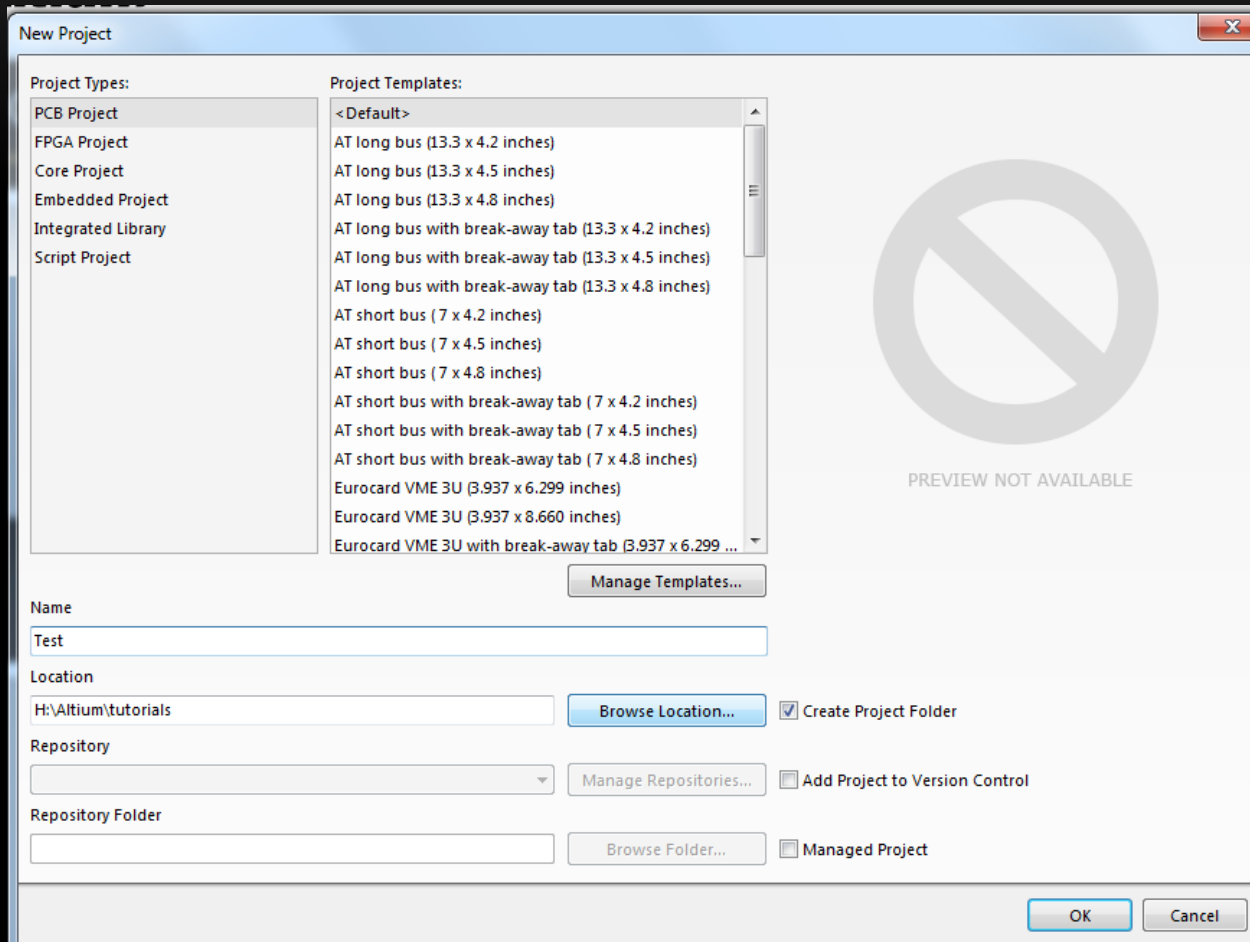


Figure 4

SETUP PROCEDURE

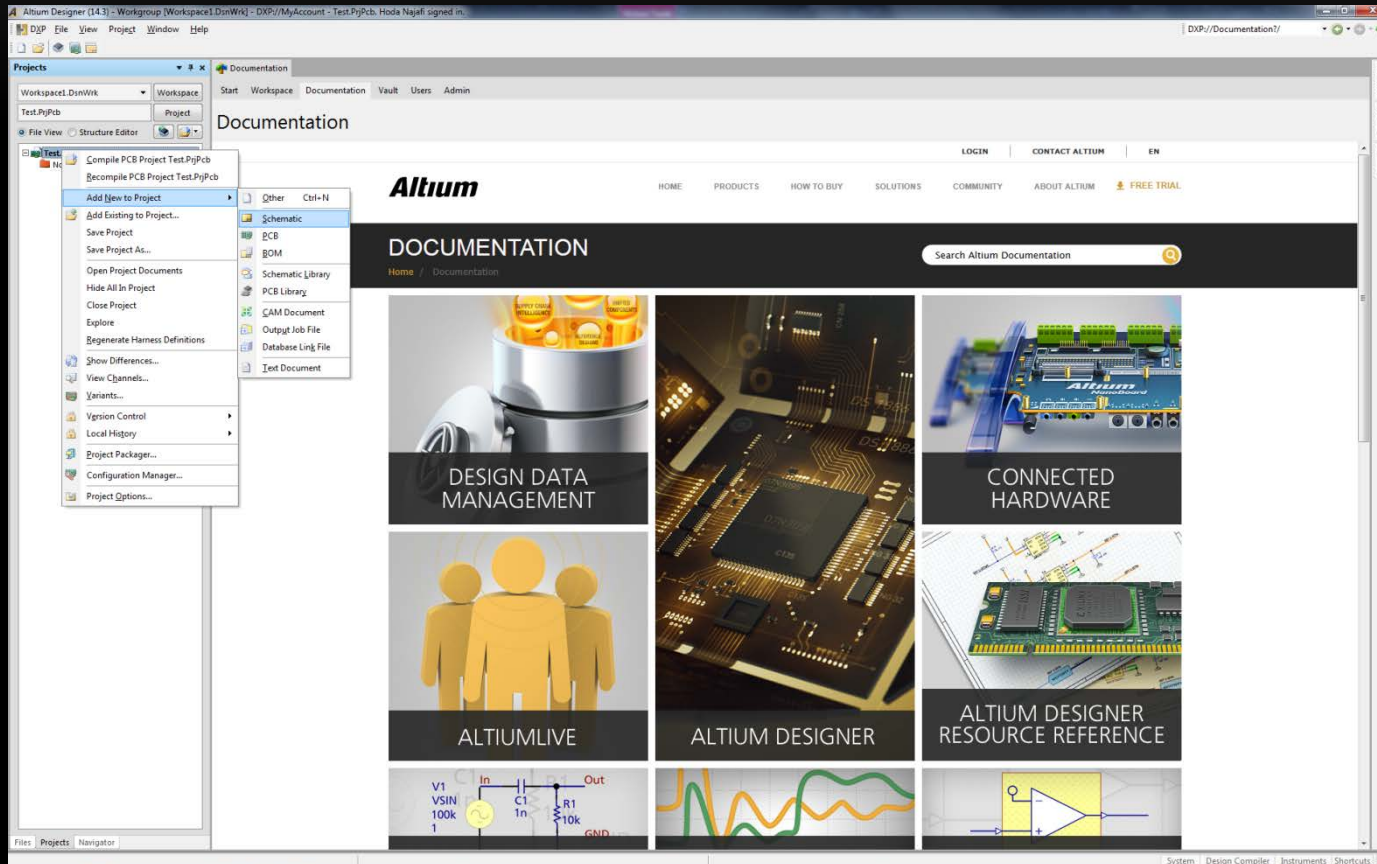


Figure 5

SETUP PROCEDURE

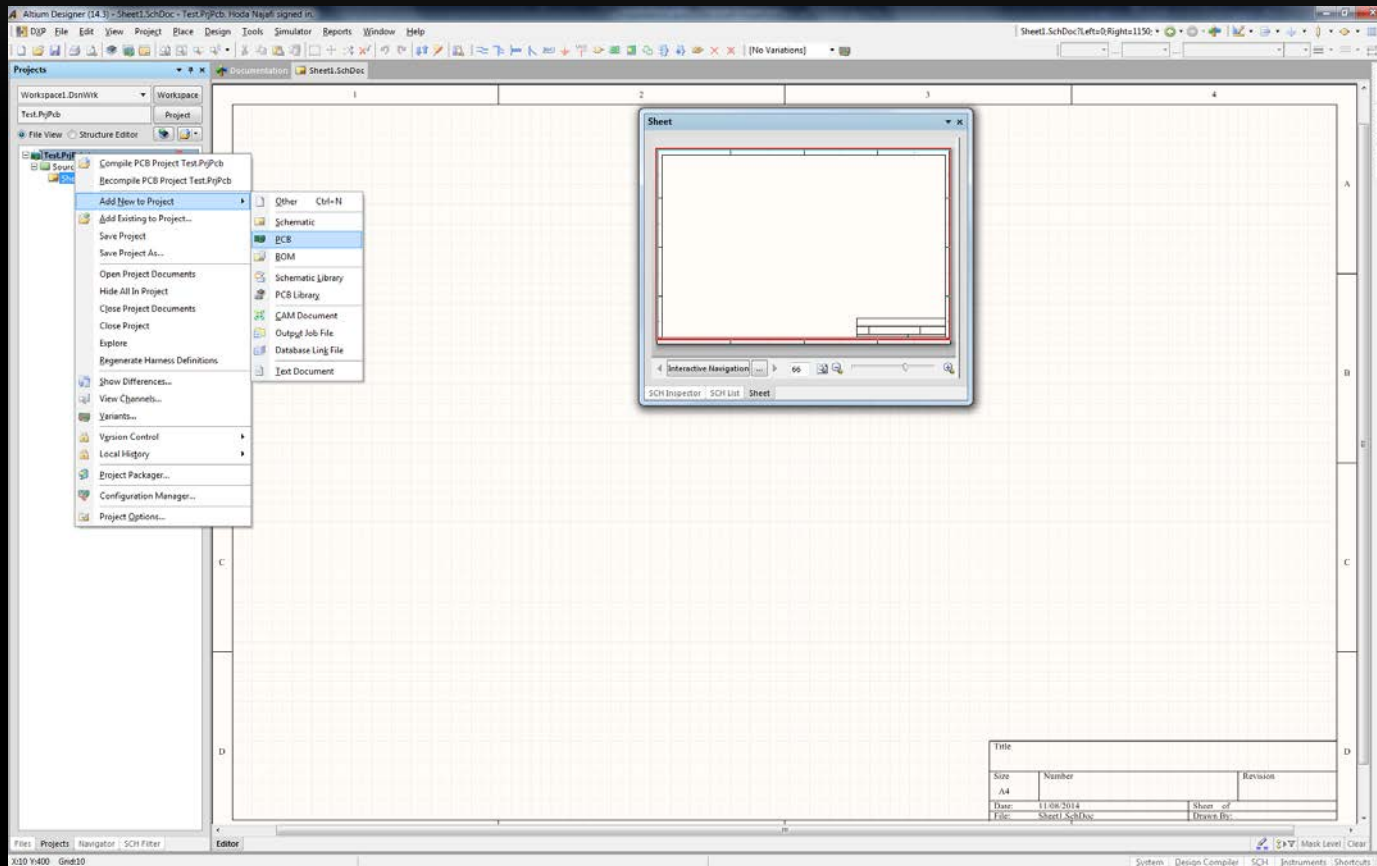


Figure 6

SETUP PROCEDURE

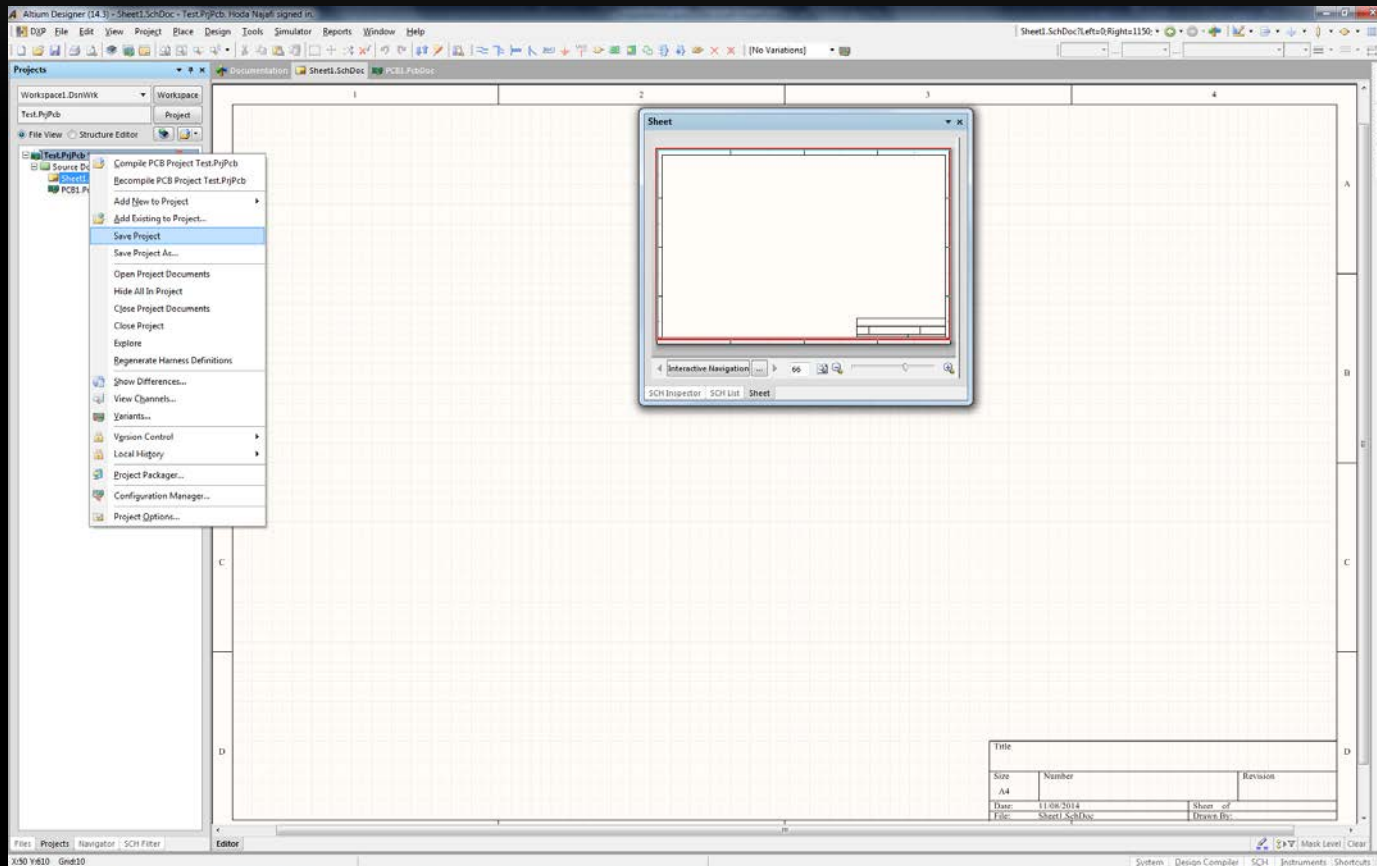


Figure 7

SETUP PROCEDURE

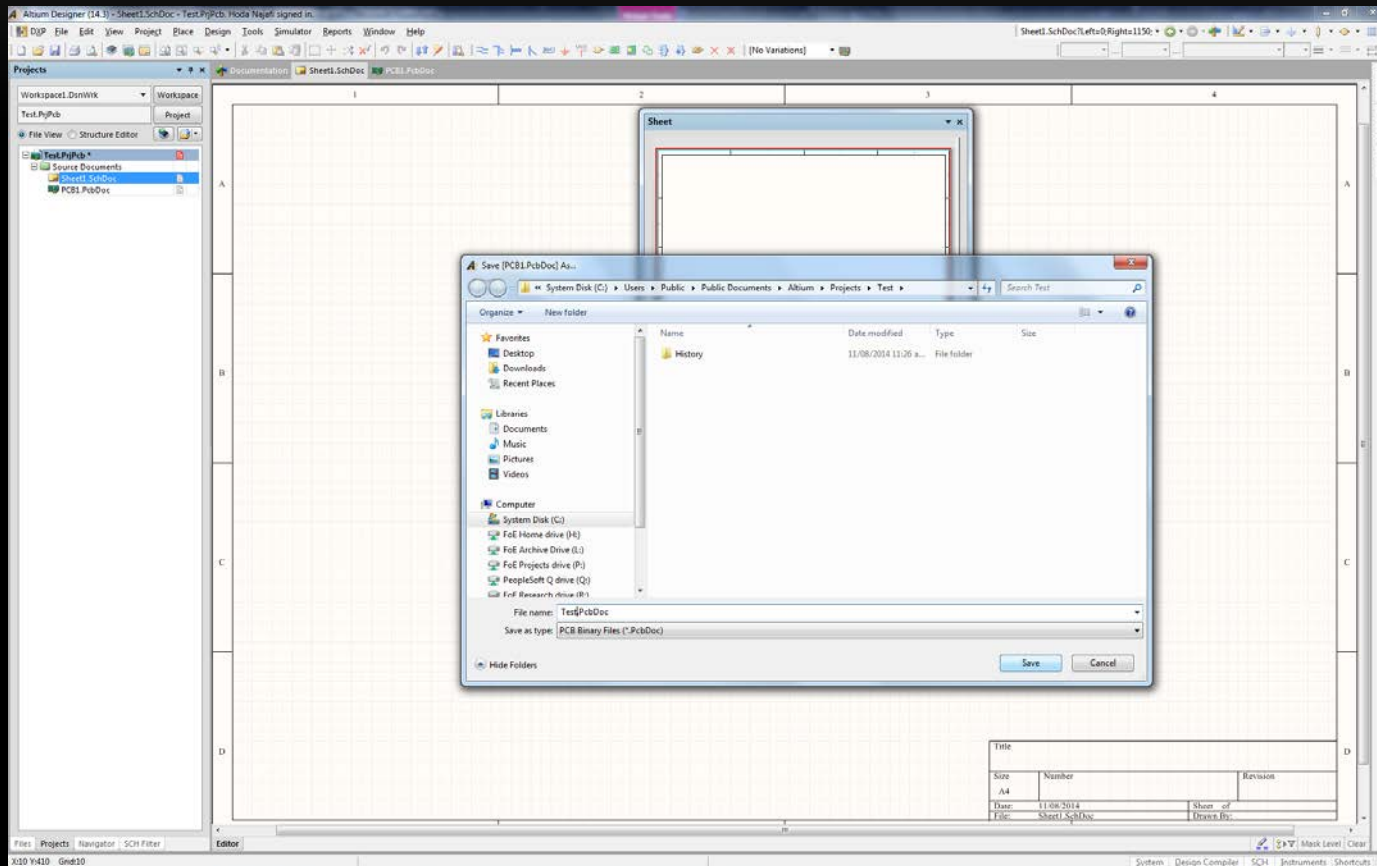


Figure 8

SETUP PROCEDURE

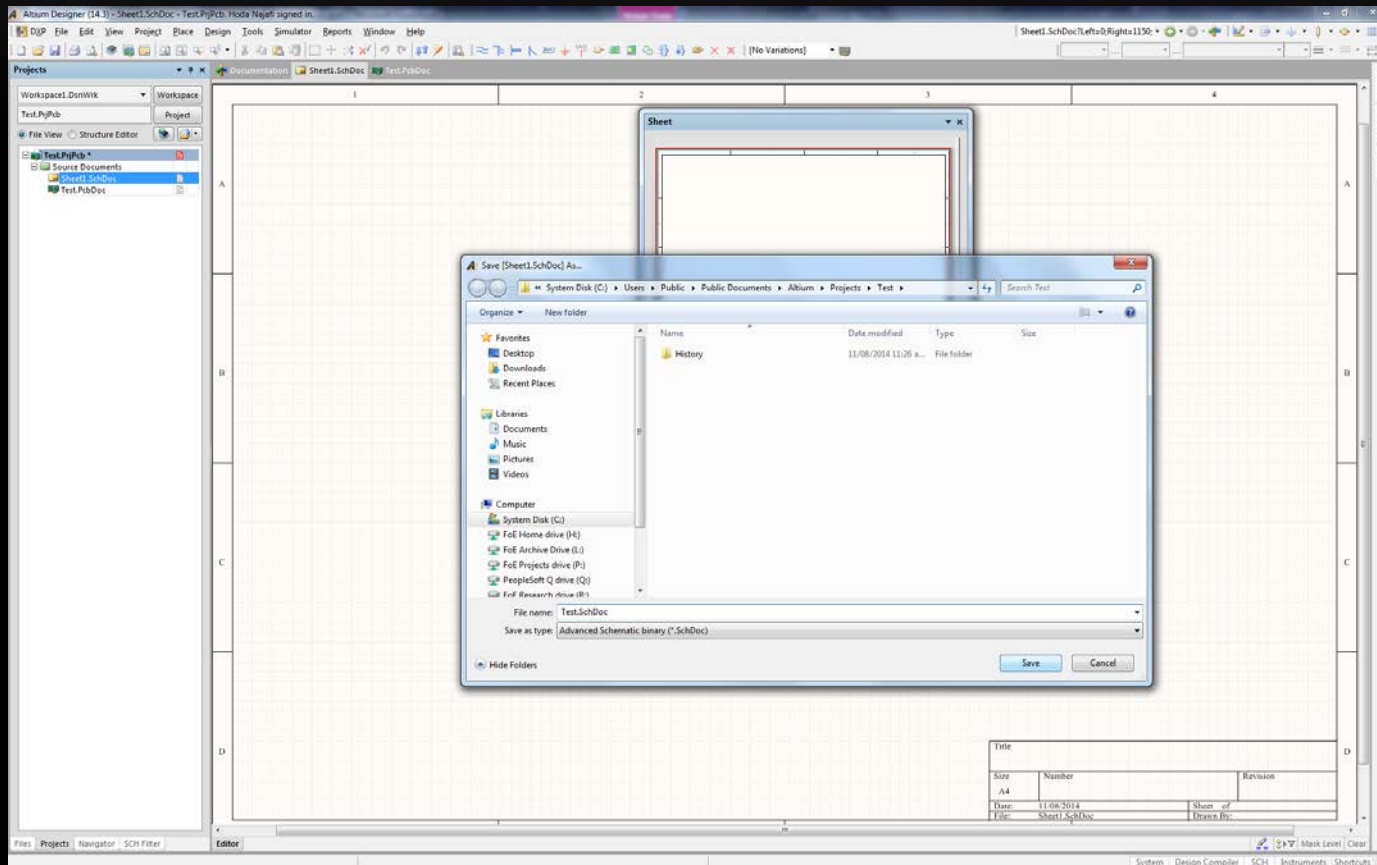


Figure 9

ADDING LIBRARIES

- Libraries are a very important part of working with your schematic. Libraries are where all of your parts will be found and selected from. If you do not have the 'Libraries' tab anywhere on your screen you should...

Click the 'System' box in the bottom right portion of the designer and then click the 'Libraries' button. ([Figure 10](#))

The *Libraries* panel enables you to browse and place components from the libraries currently available in Altium Designer

The libraries that are listed are dependent on how the libraries were left the last time the designer was used. Since the computers in the lab are shared there is a possibility that the libraries that you would like to use are not listed.

Components are categorized according to the manufacturers. They can be found in the address below:

C:\program files (x86)\Altium\AD14\Library

- The libraries that we will be using for this tutorial are:
 - Miscellaneous Connectors.intlib
 - ece component library.intlib

Integrated libraries combine schematic libraries with their related PCB footprints

- First download these libraries from Cecil and then install them to Altium as shown below. These can be added under Project tab or Installed tab.
- Project libraries are available only for the active project whereas installed libraries are available for all the projects. In this exercise, we are adding them under Project tab.
 - Open library panel and select project tab. ([Figure 14](#))
 - Click on 'add library' button and browse your computer to find the libraries you need. ([Figure 15](#))
 - You can see a list of your project libraries on the left side of the Altium page in projects tab.

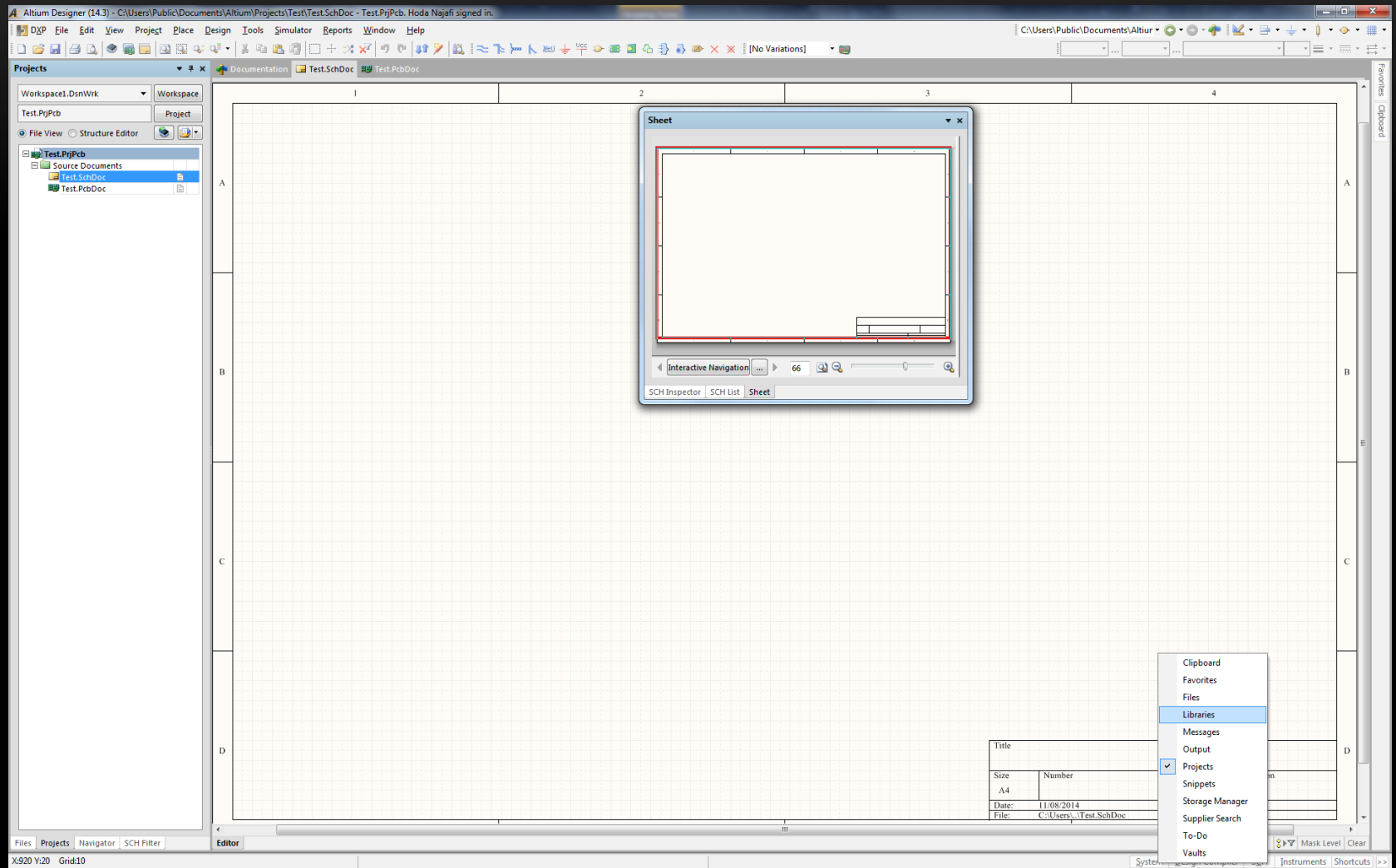


Figure 10

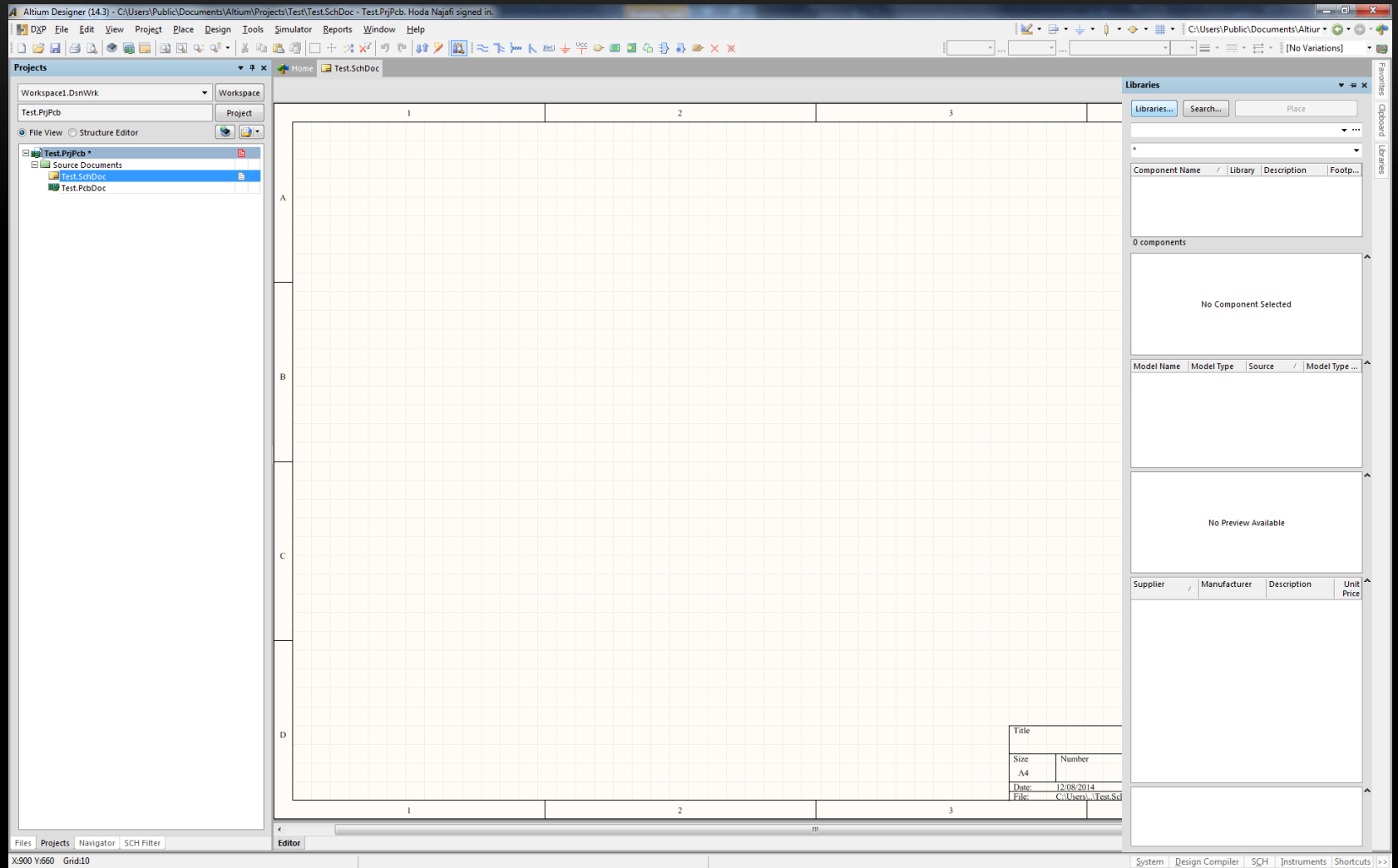


Figure 11

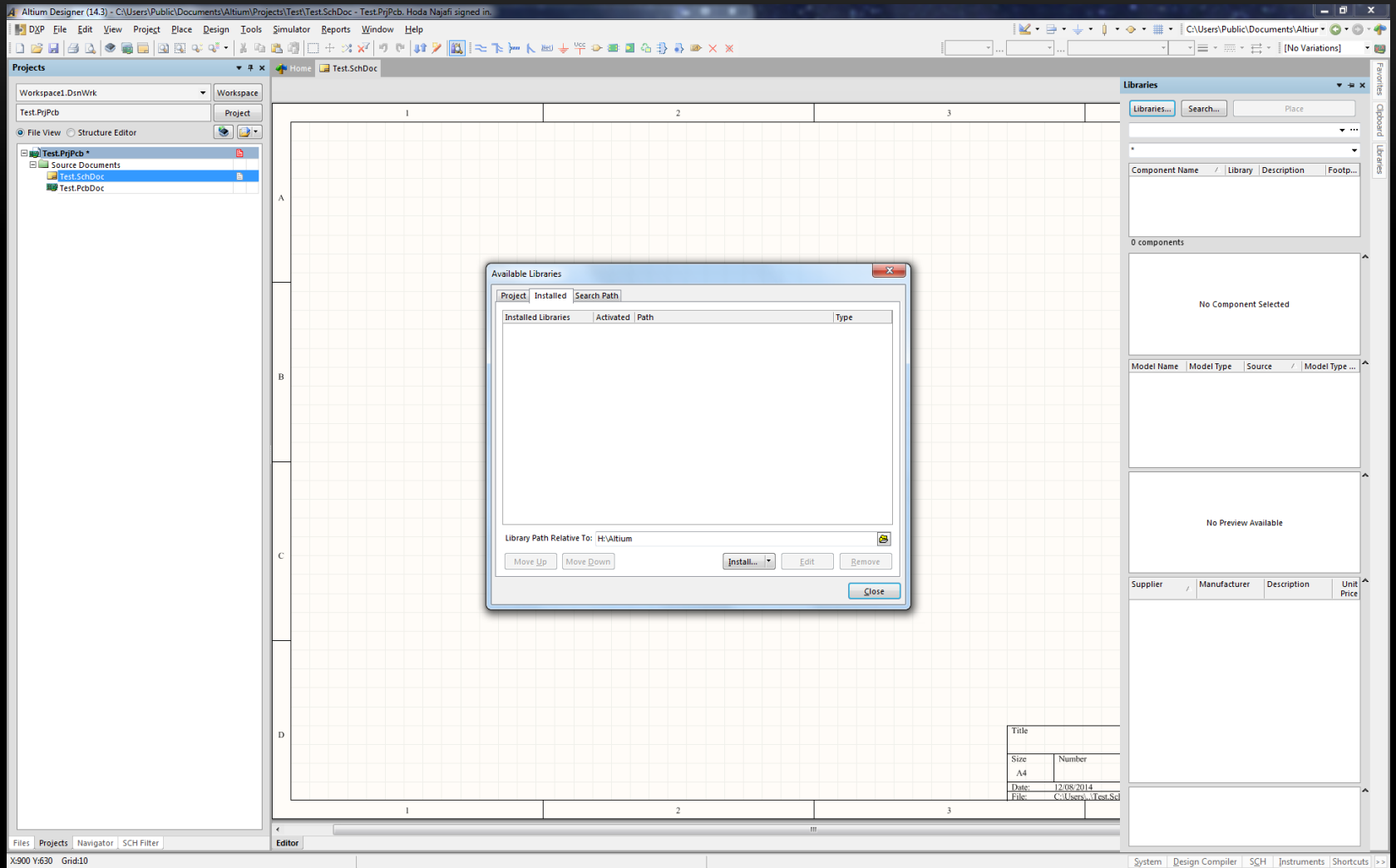


Figure 12

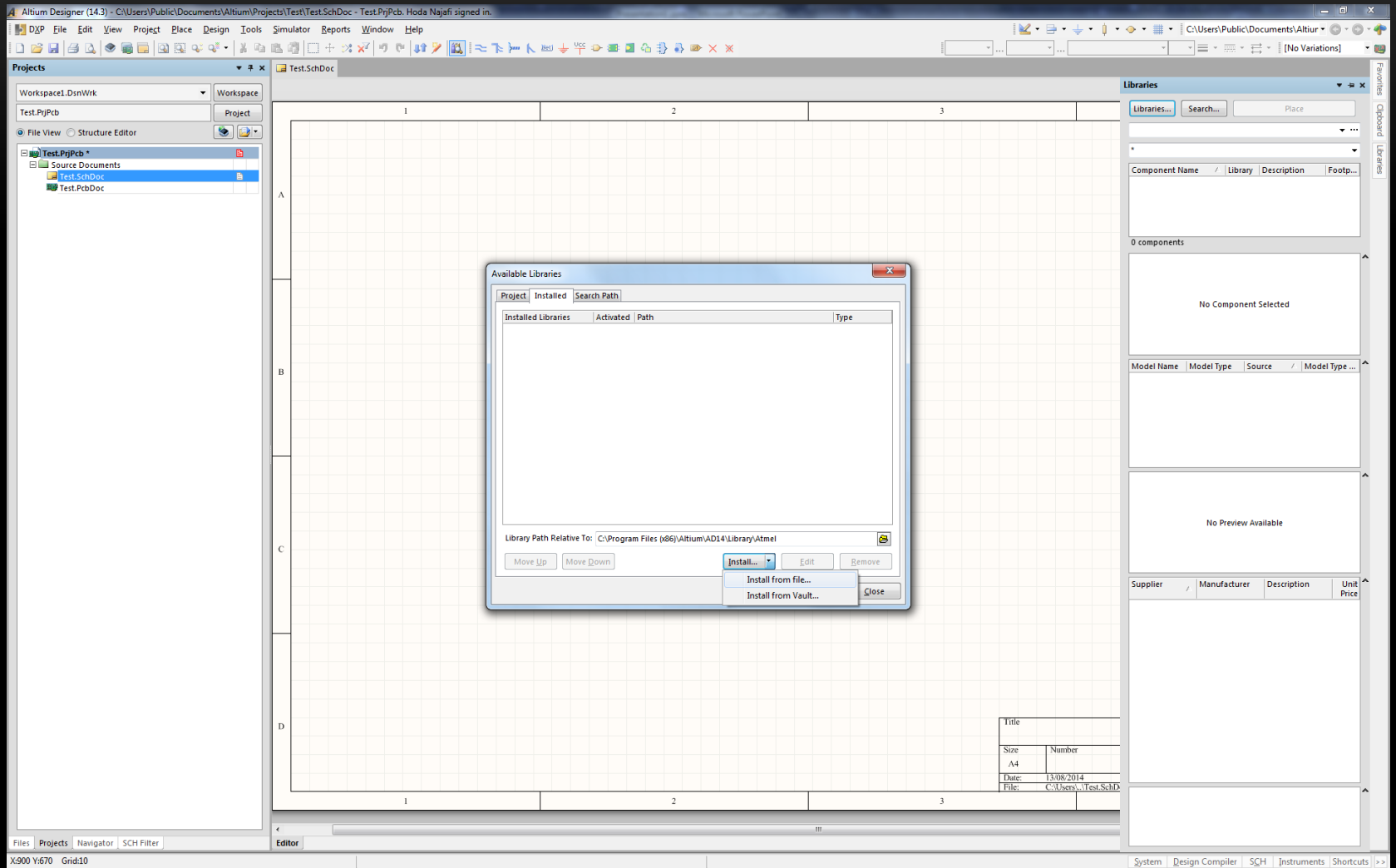


Figure 13

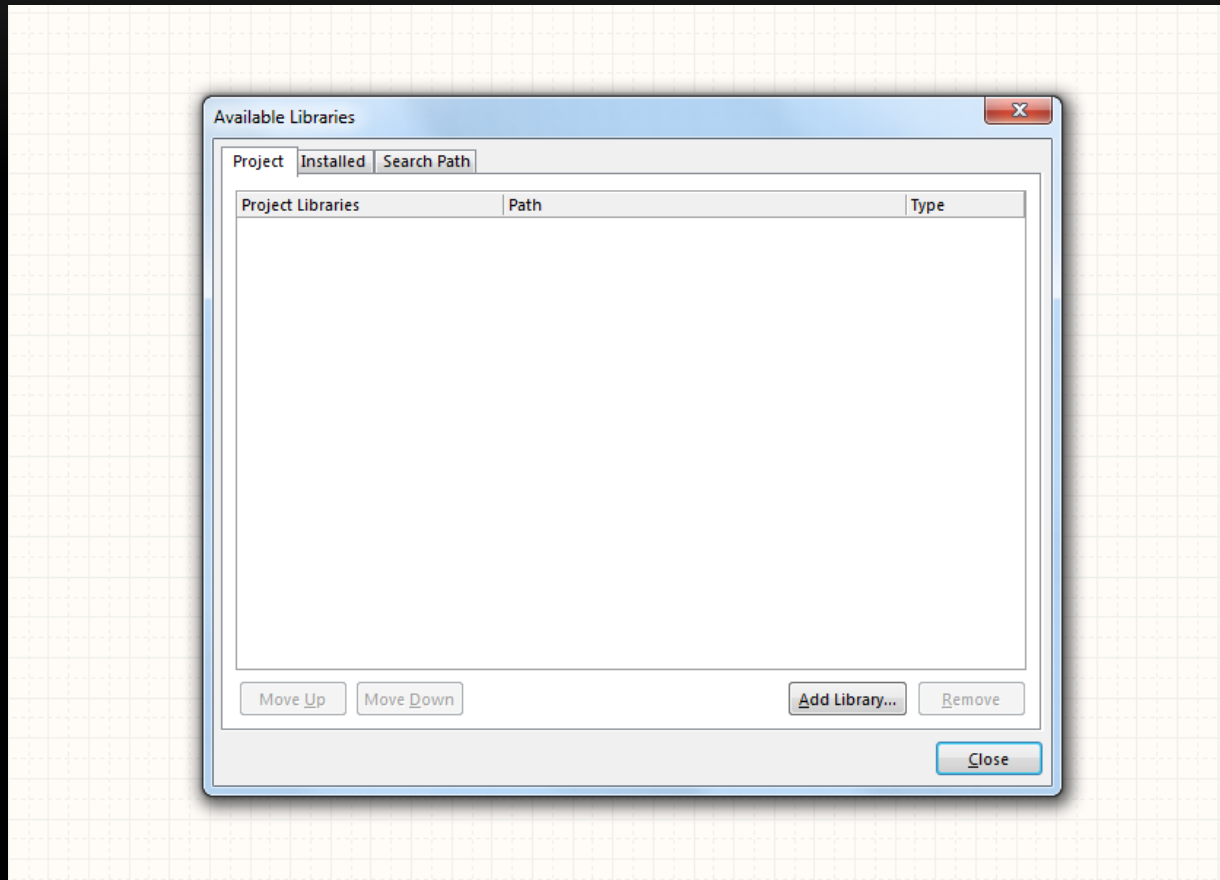


Figure 14

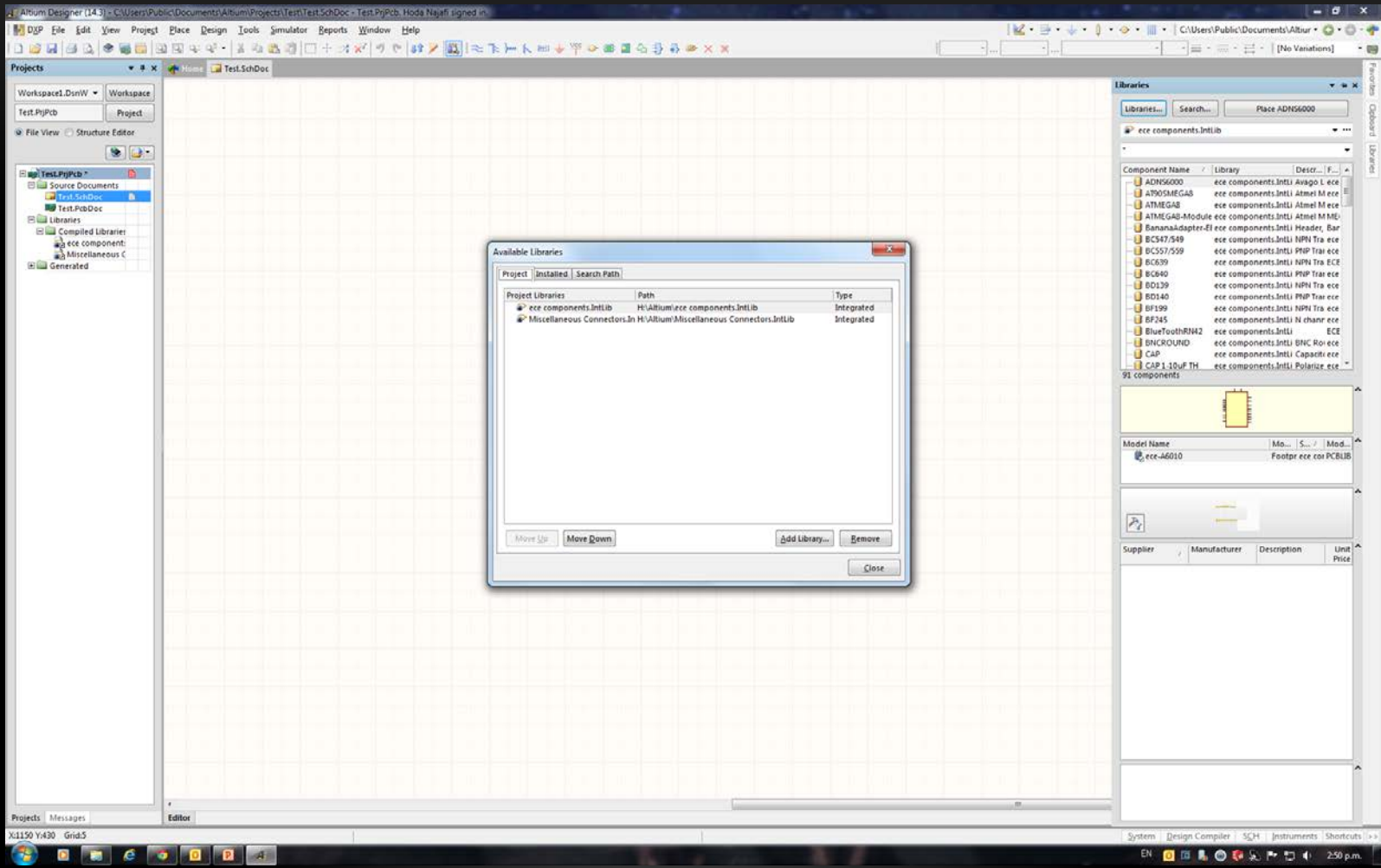
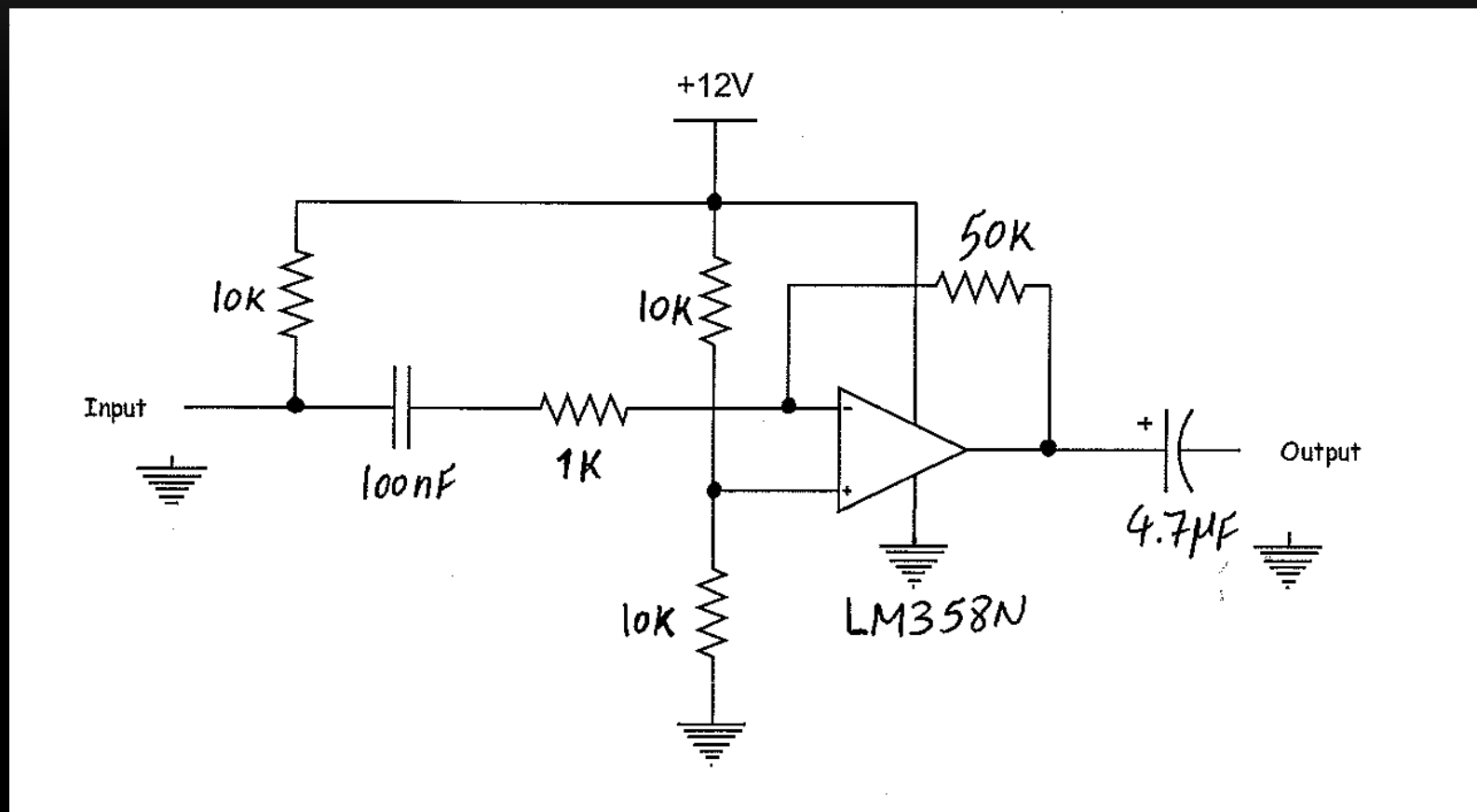


Figure 15

SCHEMATIC PRACTICE



ADDING PARTS AND MODIFYING THE SCHEMATIC



- Add a resistor to you schematic page by following these steps (below image has red letters corresponding to the steps illustrated):
 - a. Make sure you have your schematics page opened
 - b. Open the 'Libraries' tab
 - c. Select the 'ece component' library from the upper drop down (if this library is not in the drop down refer to previous step)
 - d. Type 'Res1' into the dropdown textbox below the library listing
 - e. Click the 'Place Res2' button at the top of the libraries tab
 - f. Your cursor will then have a resistor attached to it and you can place it anywhere on the schematic
 - g. Once you have placed the resistor you can either hit the 'Esc' key –OR- right click your mouse to stop placing resistors

ADDING PARTS AND MODIFYING THE SCHEMATIC

- Follow the above steps and place the following parts

a. From the 'ece component' library:

- (5) res2
- (1) Cap 1-10uF TH
- (1) Cap

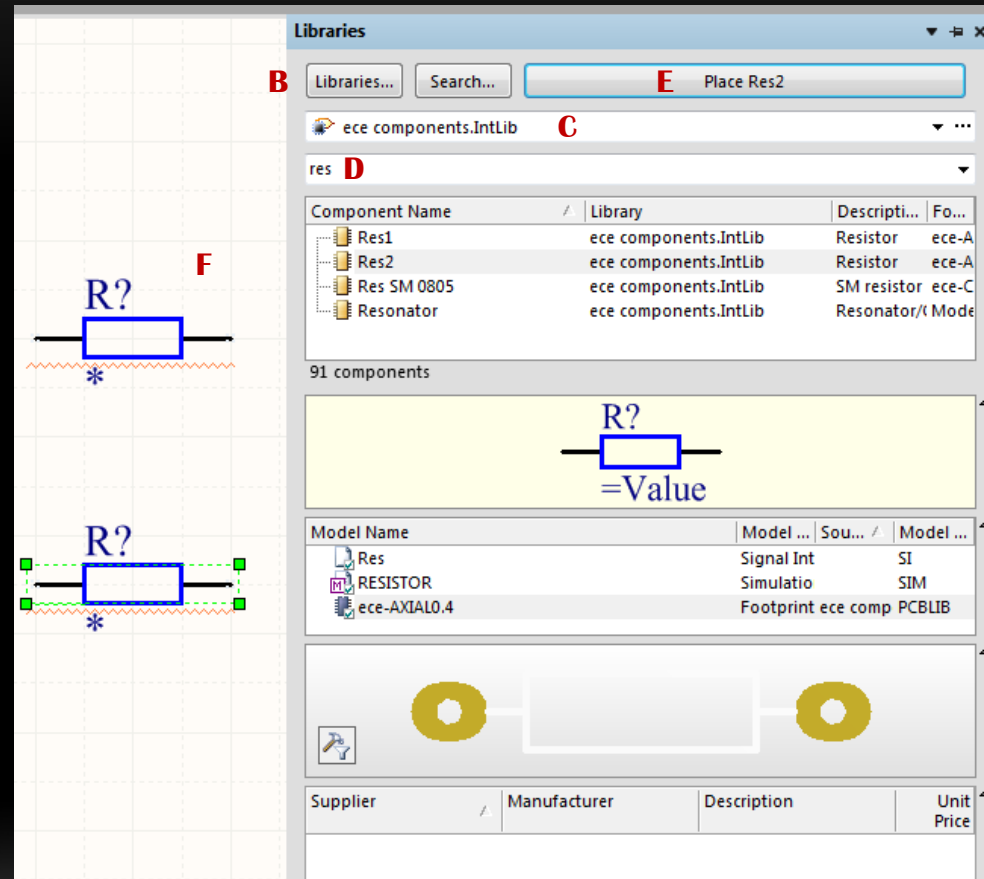
b. From the 'Miscellaneous Connectors' library:

- (1) Header 2H
- (2) Header 2

c. Search the libraries to find and place:

(1) LM358N

How to search? [See more...](#)



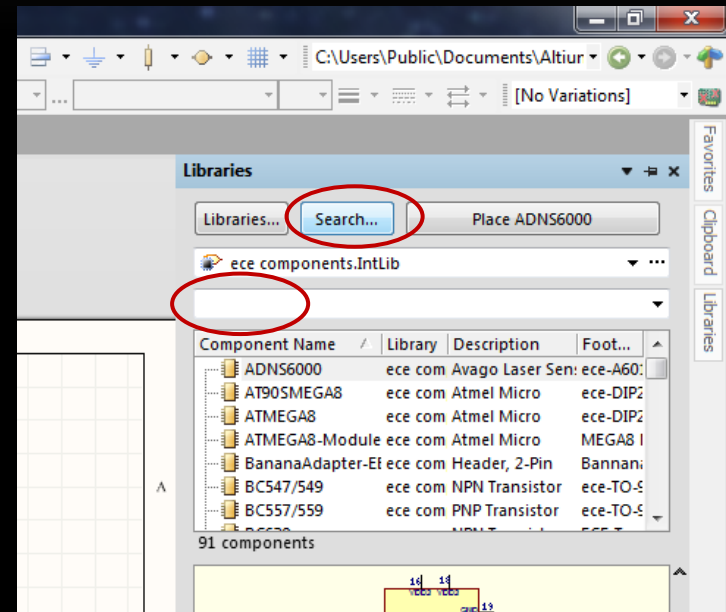
SEARCH IN LIBRARIES

- Open 'Libraries' and select 'search'
- In first row, change operator to 'contains', then write 'LM358N' in Value tab
- Make sure select 'Libraries on path' in Scope
- Brows and find the library path as:
C:\program files (x86)\Altium\AD14\Library
- Press 'Search', it takes few seconds to find.

Note: make sure the dropdown textbox below the library listing is cleared, otherwise your search may not get result.

- You may find few models which are from different manufacturers with different footprints.

Choose the one match to your design.



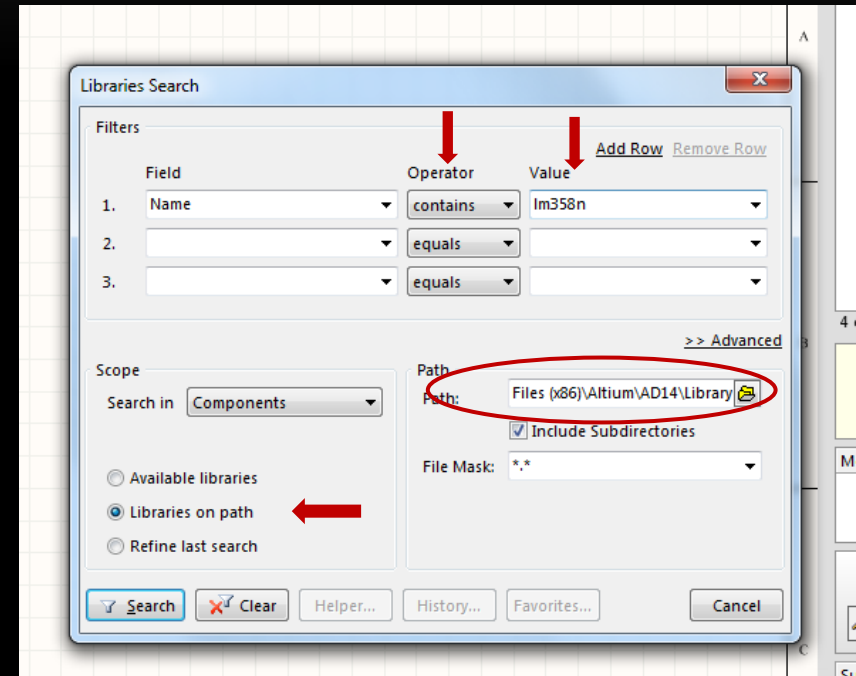
SEARCH IN LIBRARIES

- Open 'Libraries' and select 'search'
- In first row, change operator to 'contains', then write 'LM358N' in Value tab
- Make sure select 'Libraries on path' in Scope
- Brows and find the library path as:
C:\program files (x86)\Altium\AD14\Library
- Press 'Search', it takes few seconds to find.

Note: make sure the dropdown textbox below the library listing is cleared, otherwise your search may not get result.

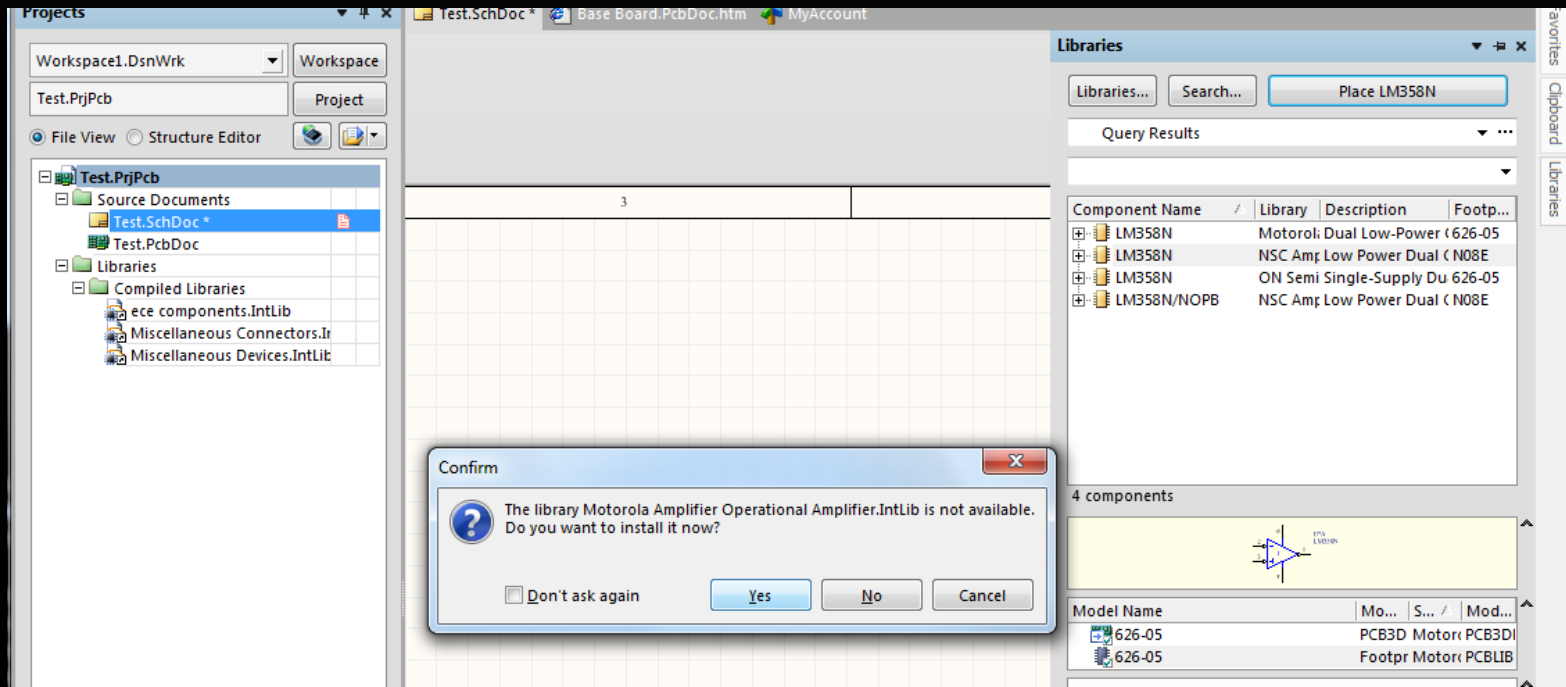
- You may find few models which are from different manufacturers with different footprints.

Choose the one match to your design.



SEARCH IN LIBRARIES

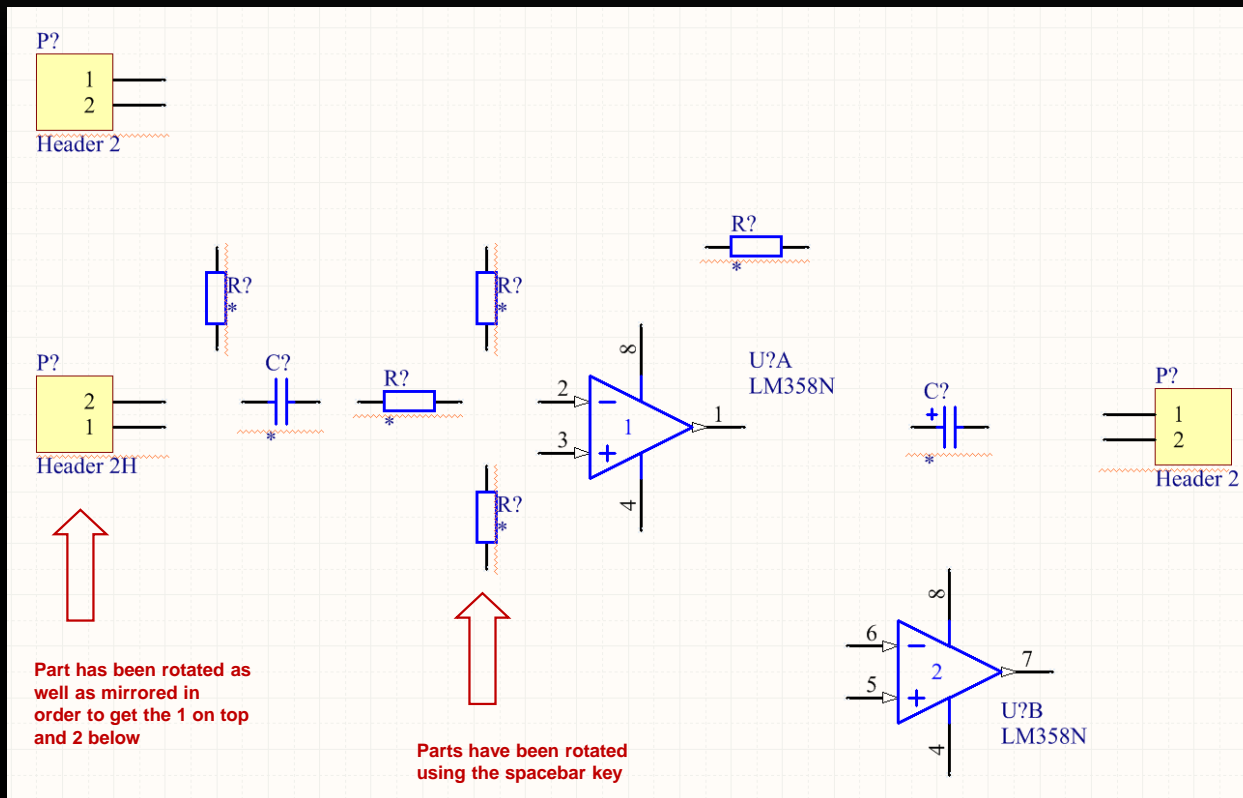
- Once you press 'Place LM358N', Altium will ask you if you want to install the related library or not. you should choose 'yes'.



- LM358N has two parts, A and B and we should place both by clicking twice.

ADDING PARTS AND MODIFYING THE SCHEMATIC

Notice: The difference between the 'Header 2' and the 'Header 2H' parts are only in the footprint created on the PCB later. Some parts will need to be rotated and this can be accomplished by pressing the **spacebar** while you are placing the part or when the part is selected. Try to place the parts more or less like the following screenshot.

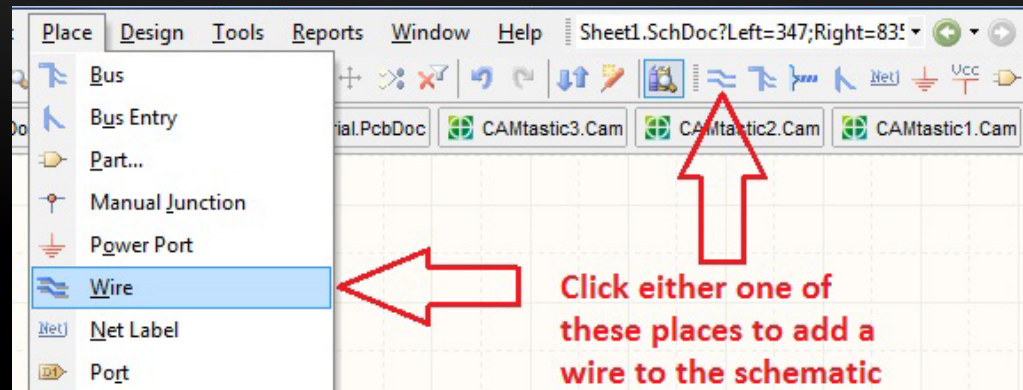


Notice: The 'Header 2H' part has been rotated as well as mirrored in order to get the 1 pin on top when the part is in this orientation. Rotation is accomplished with the **spacebar** key while the mirror is added by right-clicking the part and checking the 'mirrored' checkbox. This does not make any difference on the PCB file it just makes your schematic easier to view and understand.

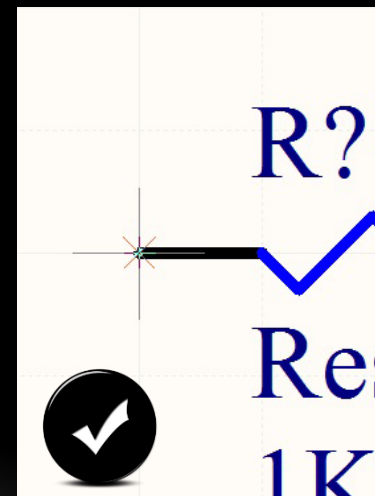
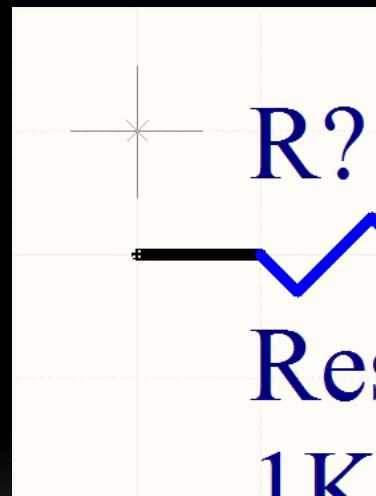
ADDING PARTS AND MODIFYING THE SCHEMATIC

It is now time to wire these parts together.

a. You can either click the 'Place Wire' icon from the upper toolbar or select **Place > Wire**. Both are illustrated here.

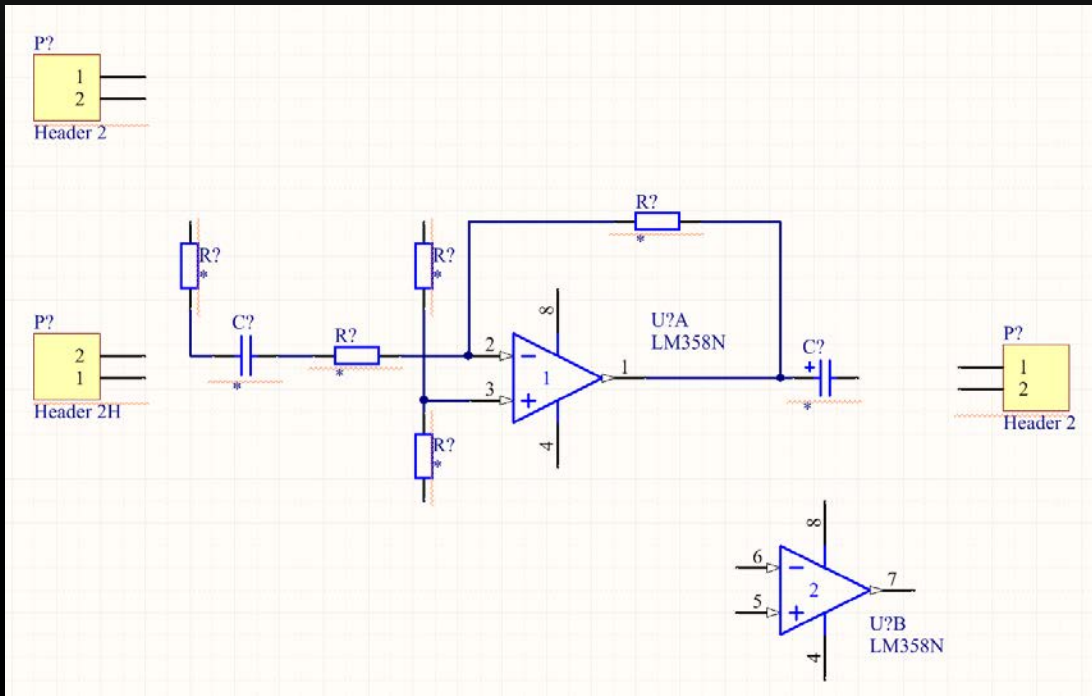


b. Once you click this icon your cursor will turn into a crosshair with a black X on it. Once you hover over a valid place to connect a wire the X will turn red. **If the X does not turn red when you place the wire a proper connection was not made and this will lead to errors later in the process.** The red X is illustrated close up here.



ADDING PARTS AND MODIFYING THE SCHEMATIC

c. Using the 'Place Wire' tool add wires between the components just like the image shown below

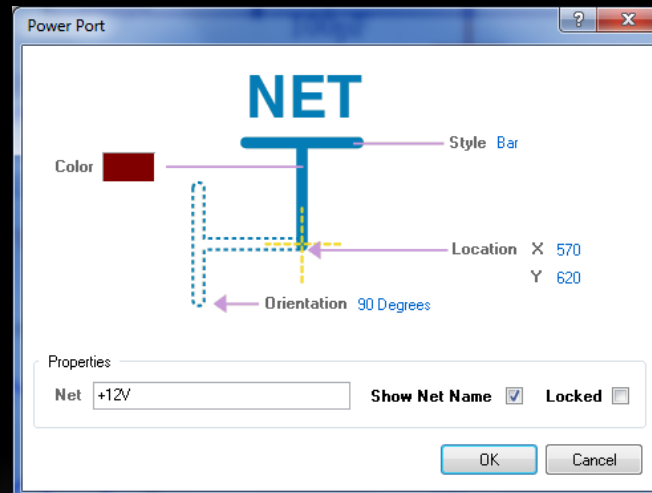


d. Once a connection between two points is made the 'Place Wire' cursor will be ready to select the next beginning point.

e. When you are finished placing wire either press the Esc key or right-click the mouse to return the cursor to normal and stop adding wires.

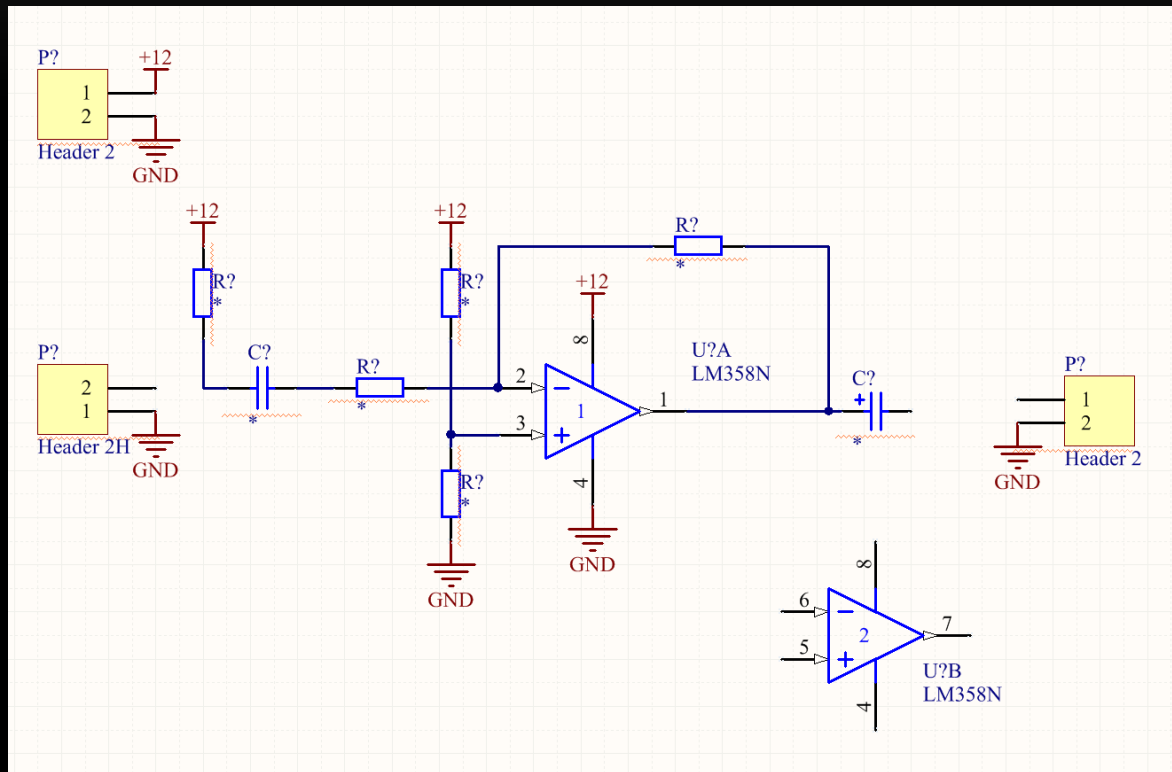
POWER AND GROUND NETS

- Now it's time to add the Power and Ground nets to the schematic
 - a. Click the 'VCC Power Port' button that is next to the 'Place Wire' button and place (4) of them in the places shown in the image on the next page
 - i. ports will be renamed from 'VCC' to '+12V'. Naming ports the same thing is the same thing as connecting them with a wire but it is a more elegant solution for keeping the schematic less congested.
 - ii. To rename a port just double click on the port and its properties dialog box will pop up. You can then change the text in the 'Net' textbox in order to change the name of the net. Shown below.



POWER AND GROUND NETS

- Click the 'GND Power Port' button next to the 'Place Wire' button and place (5) of them in the schematic in the places shown in the image on the next page

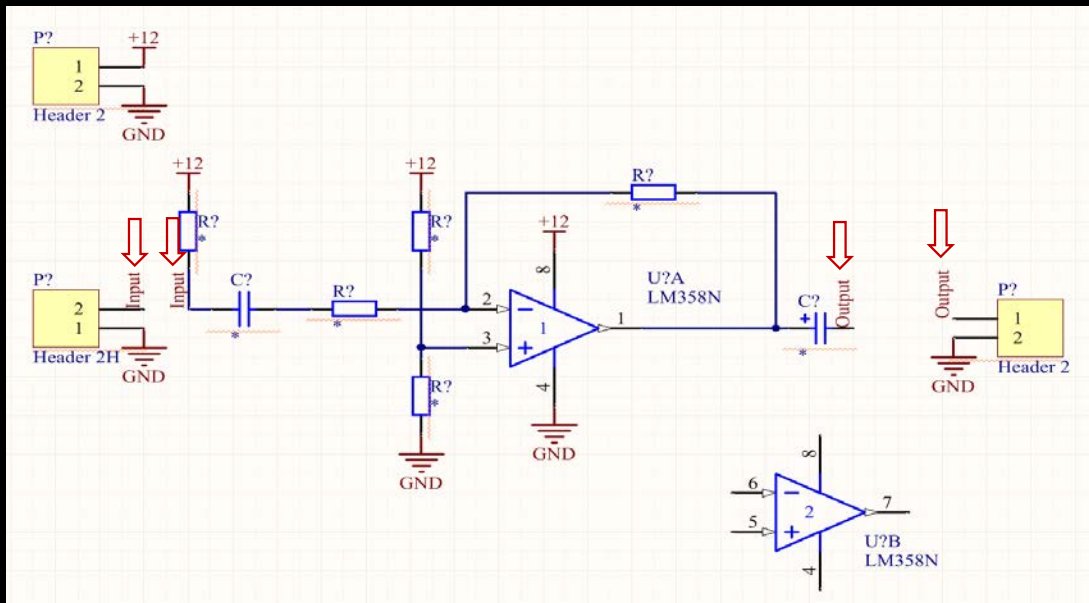


This image shows the Power (+12V) and Ground (GND) nets added to the schematic with red arrows

NET LABELS

- Now we will use named nets to add connections to our schematic without wires.
 - a. Click on the 'Place Net Label' icon next to the place wire icon –OR- click Place > Net Label.
 - b. Use spacebar to rotate the net label to make it easier to place.
 - c. Place nets in the places shown in the image below.
 - d. In order to rename the net Double click it –OR- Right click and select Properties then change the value in the text box for Net

In our case the net name will be either Input or Output



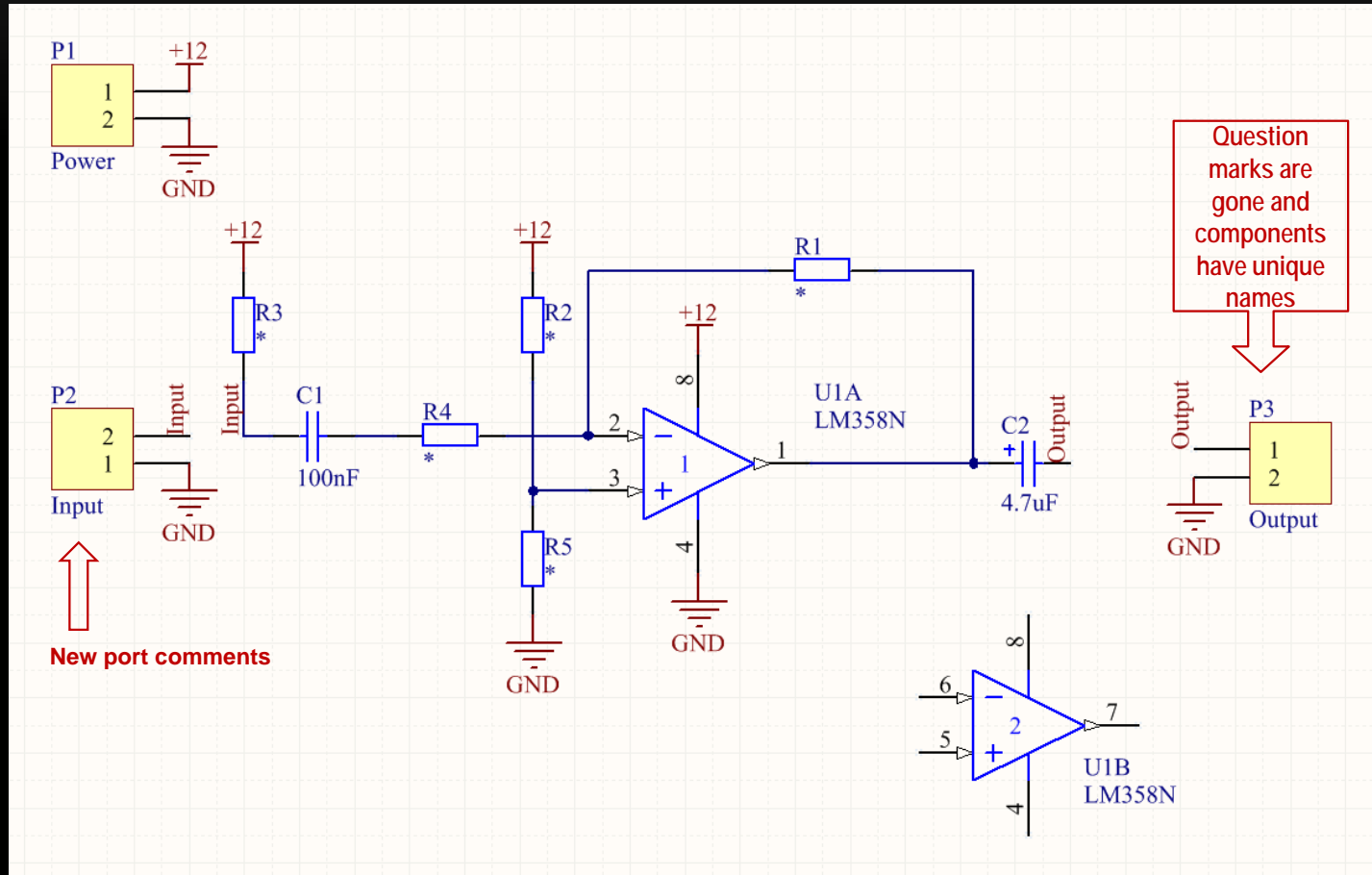
ANNOTATION

It is now time to annotate all of the unnamed components

- a. Add more descriptive comments to the ports, this will make them easier to find on the PCB later
 - i. Change the comment on the input port that reads 'Header 2H' and make it say 'Input'
 - ii. Change the comment on the output port that reads 'Header 2' and make it say 'Output'
 - iii. Change the comment on the power port that reads 'Header 2' and make it say 'Power'
- b. No two components are allowed to have the same designator we must rename them; this can be accomplished one-by-one or this can be done automatically
 - i. You can do this for each component by hand. Go to the properties of each component and change the text in the Designator field
 - ii. –OR- You can do them all at once automatically using tools in Altium (much easier)
 1. **Tools > Annotate Schematics Quietly**
 2. Click **OK** when the dialog box pops up to tell you how many changes will be made
 3. This will automatically add unique designators to each component depending on the annotation rules that are currently in place
 4. You may optionally change the annotation rules using **Tools > Annotate Schematics** if you want more control over the way the program decides to rename your designators for you. That is out of the scope of this tutorial

ANNOTATION

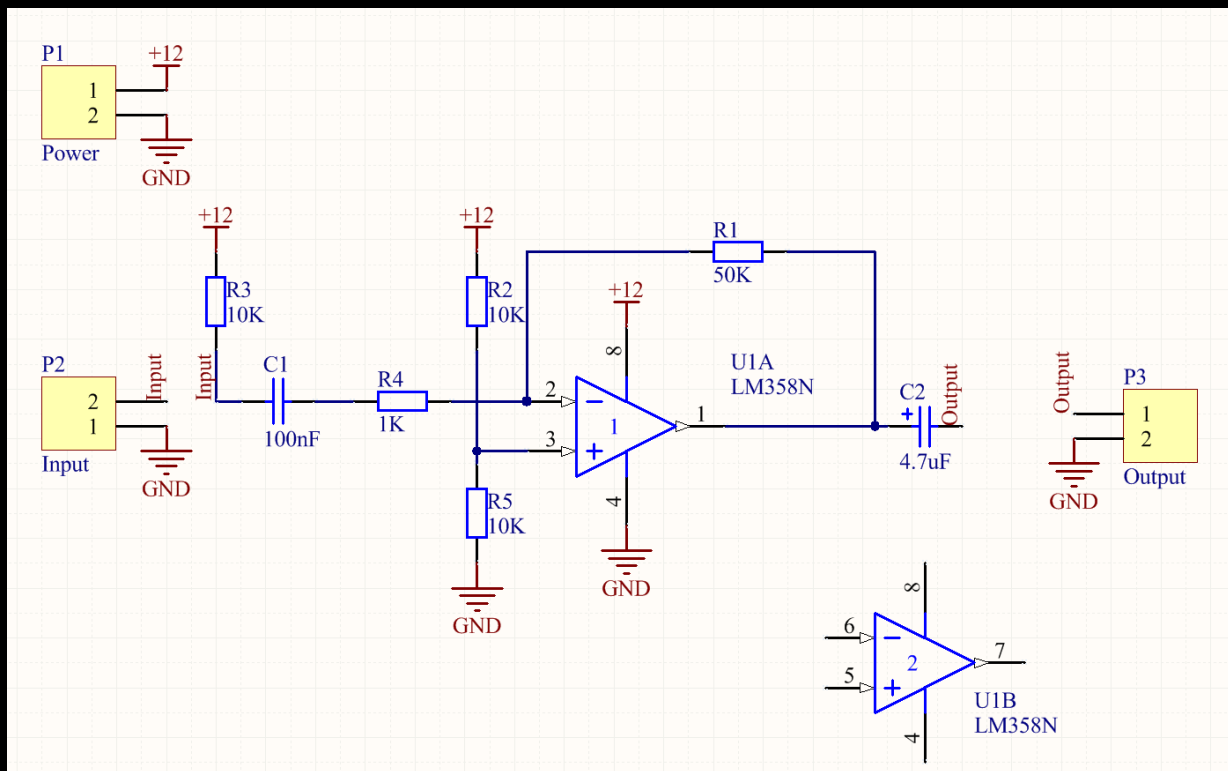
Your schematic should look something like the one below without any question marks on it



ADDING PARTS AND MODIFYING THE SCHEMATIC

Now, we should change the "value" of each component according to our design.

- Double click on each component and change the * character to the appropriate value and click OK.
- Make sure the comment is visible. (Figure 21). Your design would be similar to figure below.



ADDING PARTS AND MODIFYING THE SCHEMATIC

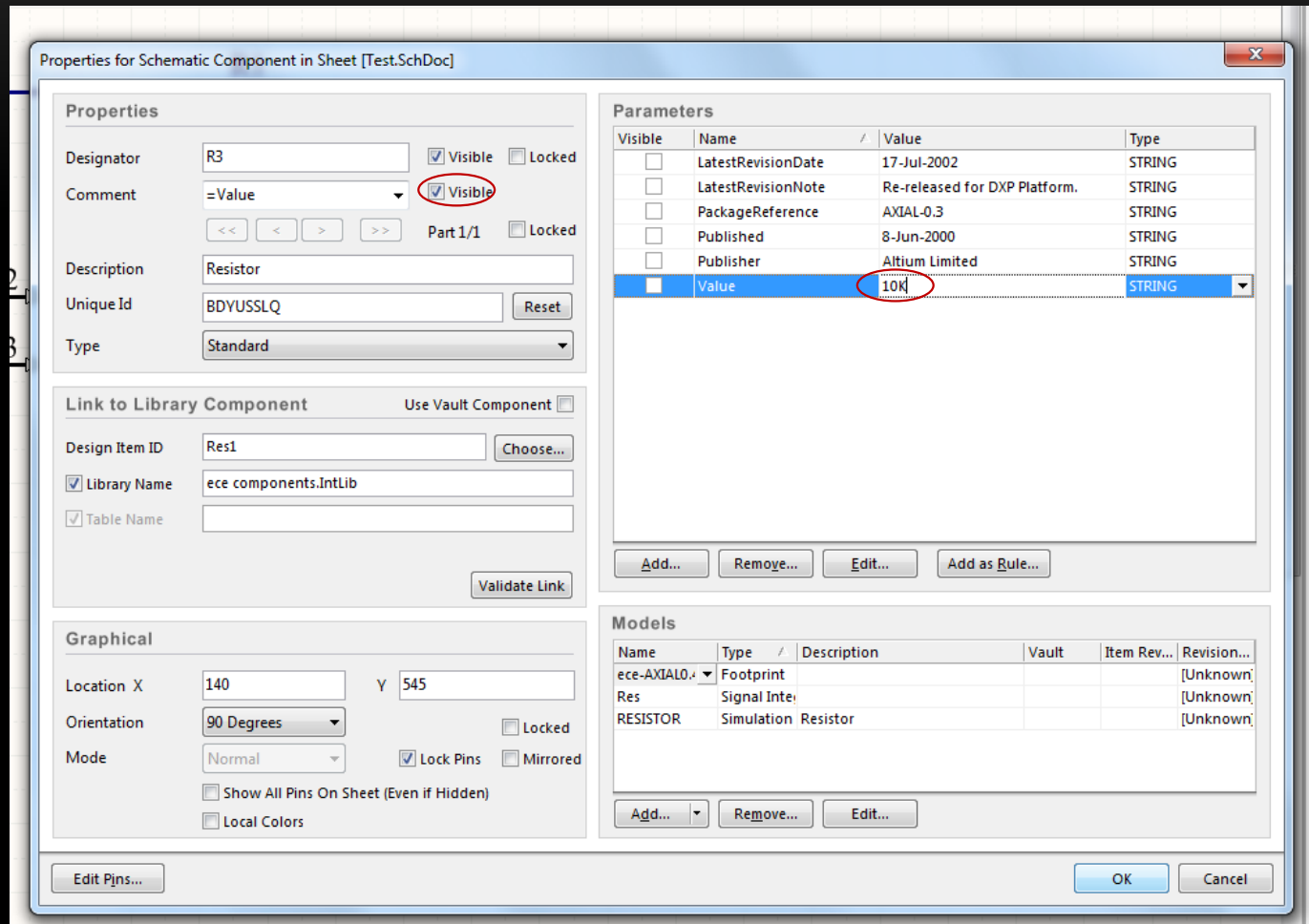


Figure 21

ADDING PARTS AND MODIFYING THE SCHEMATIC

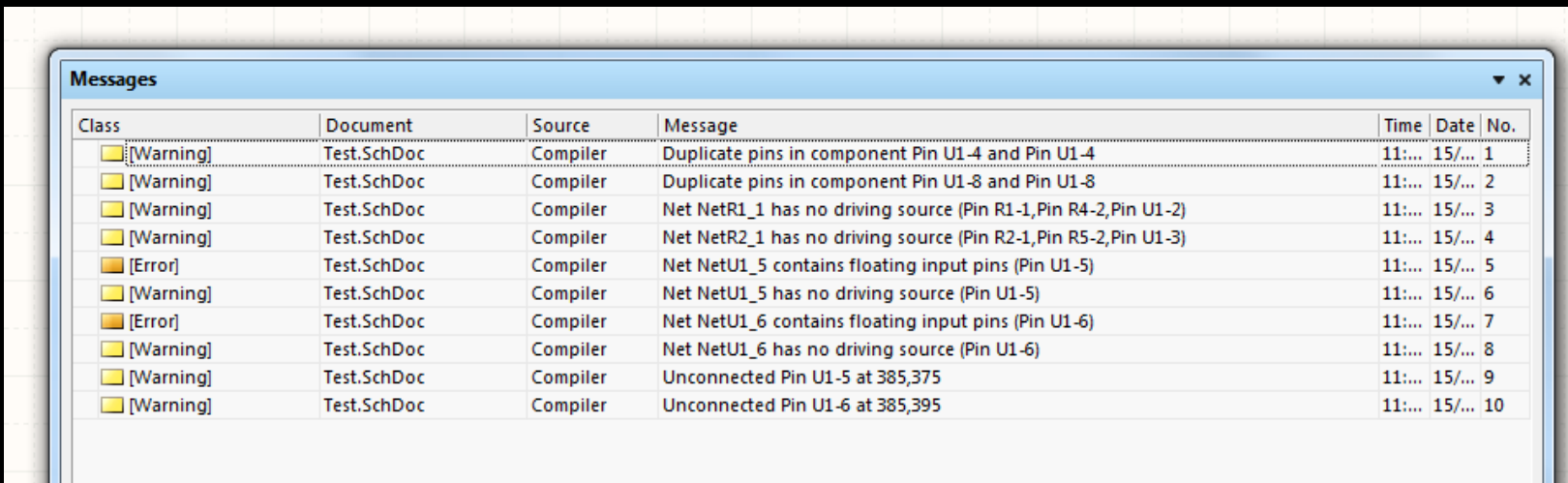
Now the Schematic is ready and we can design the PCB file.

Before transferring to PCB, make sure your schematic design has no error.

in **Project > Compile PCB project 'nameOfProject.PrjPcb'**

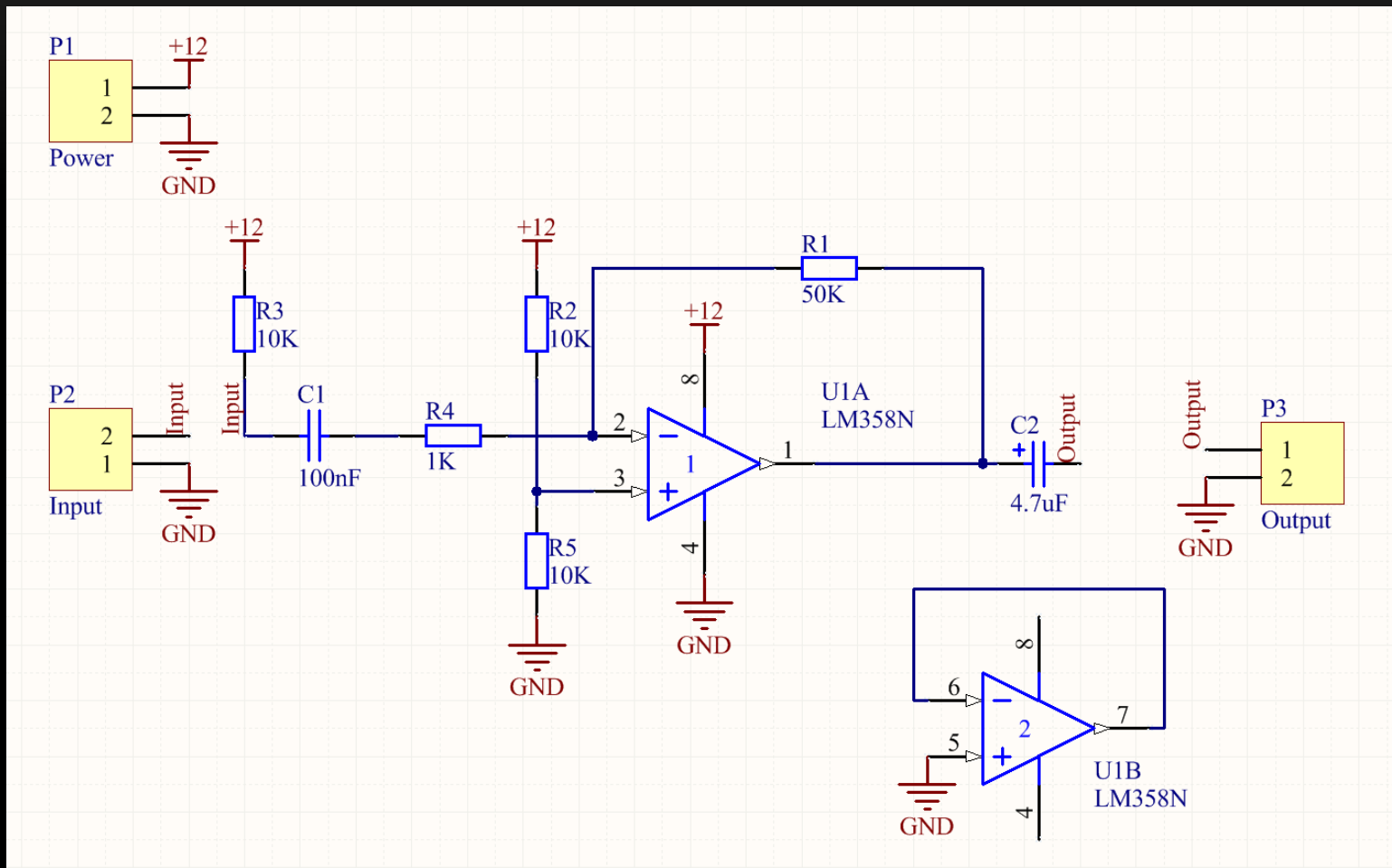
When the project is compiled, all warnings and errors will displayed in the *Messages* panel. The panel will only appear automatically if there are errors detected. If there are errors, work through each one, checking your circuit and ensuring that all wiring and connections are correct.

Here we have floating input pins which are for second part of Op-Amp and should wire as voltage follower.



Class	Document	Source	Message	Time	Date	No.
[Warning]	Test.SchDoc	Compiler	Duplicate pins in component Pin U1-4 and Pin U1-4	11:...	15/...	1
[Warning]	Test.SchDoc	Compiler	Duplicate pins in component Pin U1-8 and Pin U1-8	11:...	15/...	2
[Warning]	Test.SchDoc	Compiler	Net NetR1_1 has no driving source (Pin R1-1, Pin R4-2, Pin U1-2)	11:...	15/...	3
[Warning]	Test.SchDoc	Compiler	Net NetR2_1 has no driving source (Pin R2-1, Pin R5-2, Pin U1-3)	11:...	15/...	4
[Error]	Test.SchDoc	Compiler	Net NetU1_5 contains floating input pins (Pin U1-5)	11:...	15/...	5
[Warning]	Test.SchDoc	Compiler	Net NetU1_5 has no driving source (Pin U1-5)	11:...	15/...	6
[Error]	Test.SchDoc	Compiler	Net NetU1_6 contains floating input pins (Pin U1-6)	11:...	15/...	7
[Warning]	Test.SchDoc	Compiler	Net NetU1_6 has no driving source (Pin U1-6)	11:...	15/...	8
[Warning]	Test.SchDoc	Compiler	Unconnected Pin U1-5 at 385,375	11:...	15/...	9
[Warning]	Test.SchDoc	Compiler	Unconnected Pin U1-6 at 385,395	11:...	15/...	10

FINAL SCHEMATIC



MAKING AND MODIFYING PCB FILE

We are now ready to update the PCB file

- a. Design > Update PCB Document 'nameOfProject.PcbDoc'
- b. Scroll to the bottom of the window that pops up and uncheck the box next to the room that the program is trying to add
- c. Click **Validate Changes**, make sure all of the changes get validated
- d. Click **Execute Changes**, this will add all of the components and nets to your PCB document
- e. Close the update window ([Figure 23](#))

The PCB file should have automatically opened for you; if not, open it now

- a. Click **View > Fit Sheet** to see the board as well as all of your components
- b. You will notice that the components are added outside of the board area (black area)
- c. The components can be dragged onto the board as you place them

Engineering Change Order					Status		
Enable	Action	Affected Object		Affected Document	Check	Done	Message
Add Components(11)							
<input checked="" type="checkbox"/>	Add	C1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	C2	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	P1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	P2	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	P3	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	R1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	R2	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	R3	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	R4	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	R5	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	U1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
Add Nets(9)							
<input checked="" type="checkbox"/>	Add	+12	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	GND	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	INPUT	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	NetC1_2	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	NetC2_1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	NetR1_1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	NetR2_1	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	NetU1_6	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Add	OUTPUT	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
Add Component Classes(1)							
<input checked="" type="checkbox"/>	Add	Test	To	PCB2.PcbDoc	<input checked="" type="checkbox"/>		
Add Rooms(1)							
<input type="checkbox"/>	Add	Room Test (Scope=InComponentClass(Te: To		PCB2.PcbDoc			

Only Show Errors

Figure 23

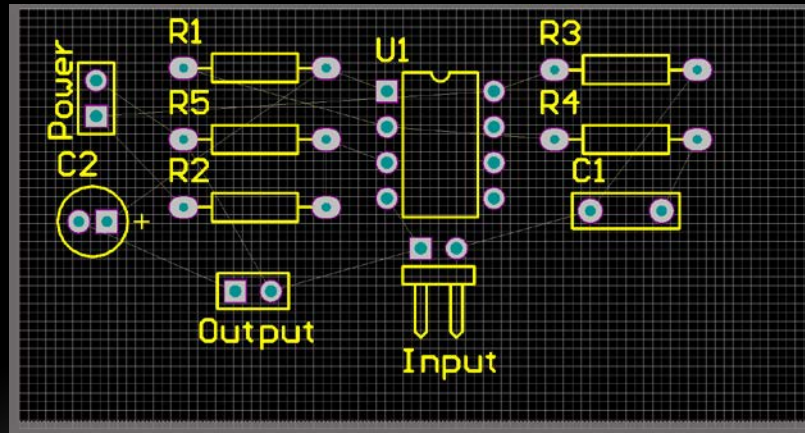
COMPONENT PLACEMENT

We will now arrange the components on the board

- a. Go to the PCB document
- b. Begin dragging the footprints of the components over to the board
 - i. The footprints and the designators can once again be rotated using **Spacebar** while they are being dragged
 - ii. To get a better view at any time you can use **View > Fit Document** or you can use **Page Up** and **Page Down** (or **Ctrl+Scroll**) to zoom in and out at any time
 - iii. The designators of the components can be dragged independently of the component in order to make the PCB document easier to read at times
 - iv. The comments or the designators from the schematic file can be added to or removed from the PCB by opening the properties of the component on the PCB document and checking or unchecking the hide box under the 'Designator' and 'Comments' sections.
 - v. As you drag a component around the board you will notice thin lines connecting the nodes of the component to other nodes around the board. These lines will jump to the closest connection that can be made on the board. Take notice of these lines and attempt to keep them short and if they are crossed try rotating the component until the lines are straight.
 - vi. Try to arrange your components in a compact design as many manufacturing houses will charge you depending on the size of the board. You will get better at this the more you use the software. For an example of the layout look at the image below

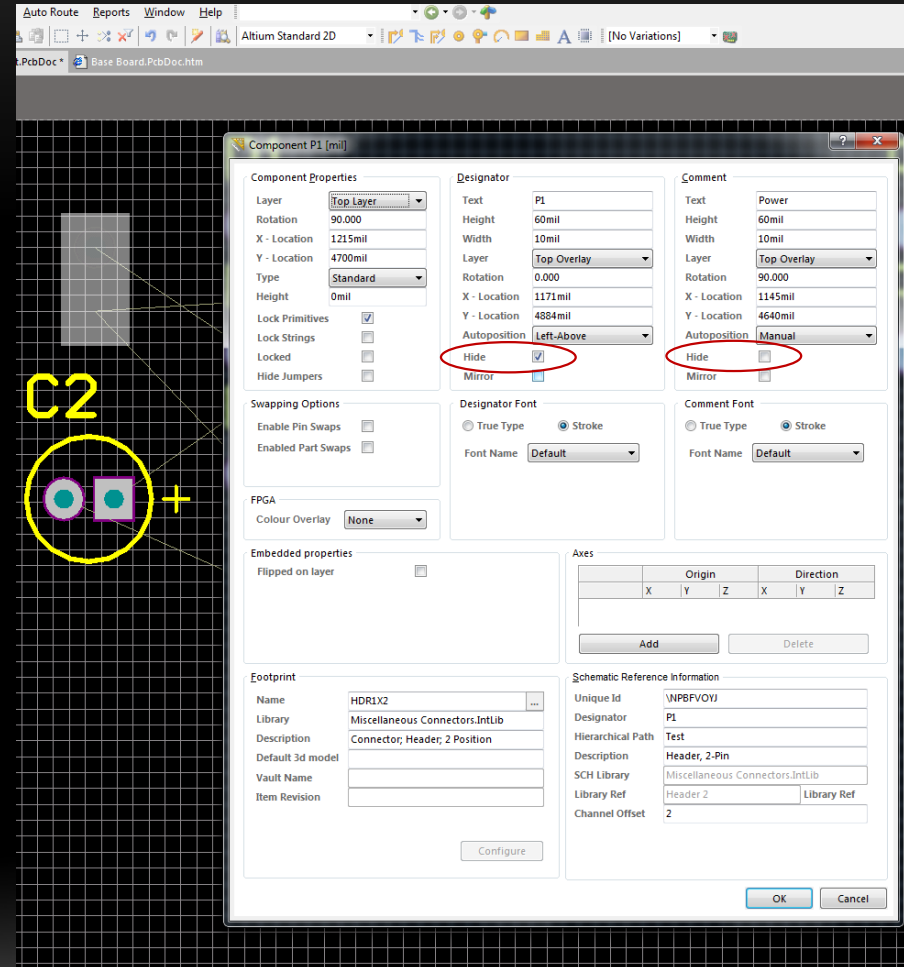
REDEFINE BOARD SHAPE

- Redefine the board shape (The black background on the screen is the board)
 - a. **View > Board Planning Mode**
 - b. a green screen will appear. Then go to **Design > Redefine Board Shape**
 - c. The cursor will turn into a crosshair
 - d. Zoom in to get a closer view of the grid and click the first corner of the new board shape that you want; continue to the next corner and click again; once you return to the first corner that you clicked on there will be a circle around the crosshair letting you know that you are back where you started; Once you are finished either **Right-click** or press **Esc** in order to stop redefining the board.
 - e. **View > 2D Layout Mode.** The board should look something like the image below



COMPONENT PROPERTIES

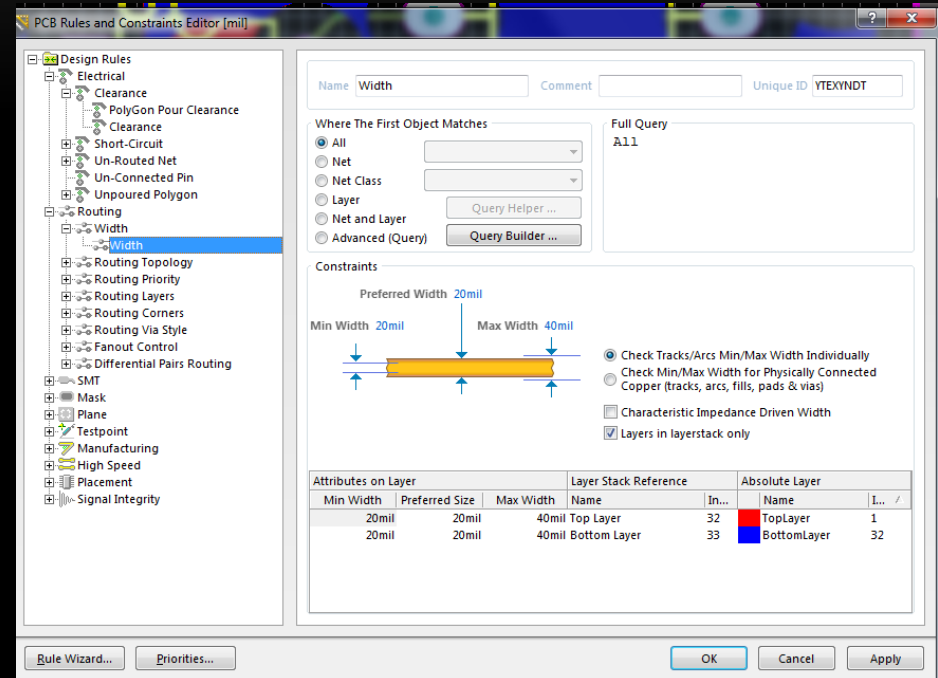
- You may hide/show Designator and Comment of each component. Double click on the component and this window will appear.



DESIGN RULES

Change the design rule of the board to define Track Width and Clearance

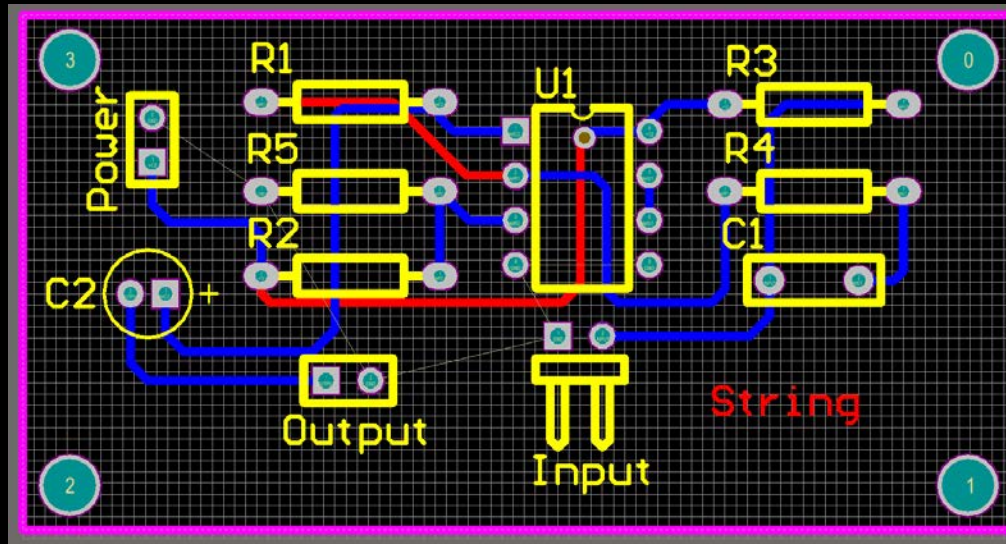
- a. Open the rule wizard using **Design > Rule**
- b. In the 'Routing' select 'width' and change the constraints to:
 - Min Width: 20mil (0.5 mm)
 - Preferred Width: 20mil (0.5 mm)
 - Max Width: 40mil (1 mm)
- c. in 'Electrical' select 'Clearance' and change the 'Minimum Clearance' to 20mil (or 0.5mm).



Note: check "ECE Protel Design Guide" file and follow the design rule settings according to the instruction. It is available on ECE webpage.

ROUTING

- Let's start routing:
- **Place > Interactive Routing** ; The cursor will turn into a crosshair
- Start from a pad and find a suitable path to the destination which is defined by a thin line.
- You can change track angles by pressing spacebar while being in 'place track' mode.
- Remember to zoom in/out to have a better view of the pads during routing.
- Using via in case of moving from top layer to bottom layer and vice versa.



CONFIGURATION

Add your initials and a revision number

- a. Use **Place > String** to add the string to an empty space on your board
- b. Double click the string to open its properties
 - i. Change the text field to your course name and group number
i.e. 'Course name - Group number'

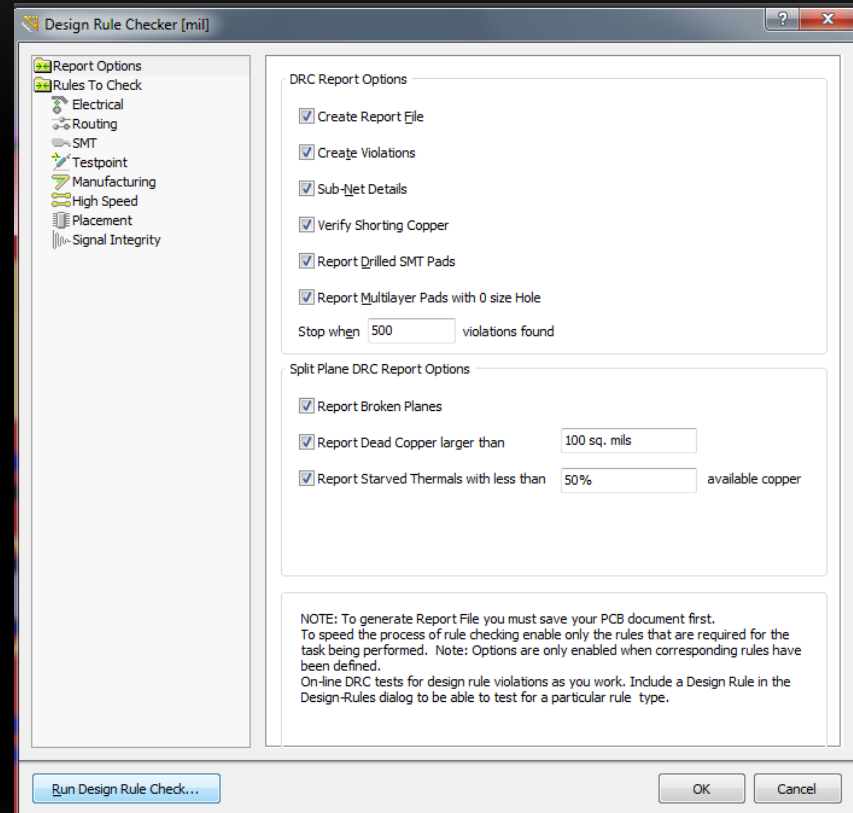
Add a keepout layer to the perimeter of your board

some manufactures will use this to cut your board

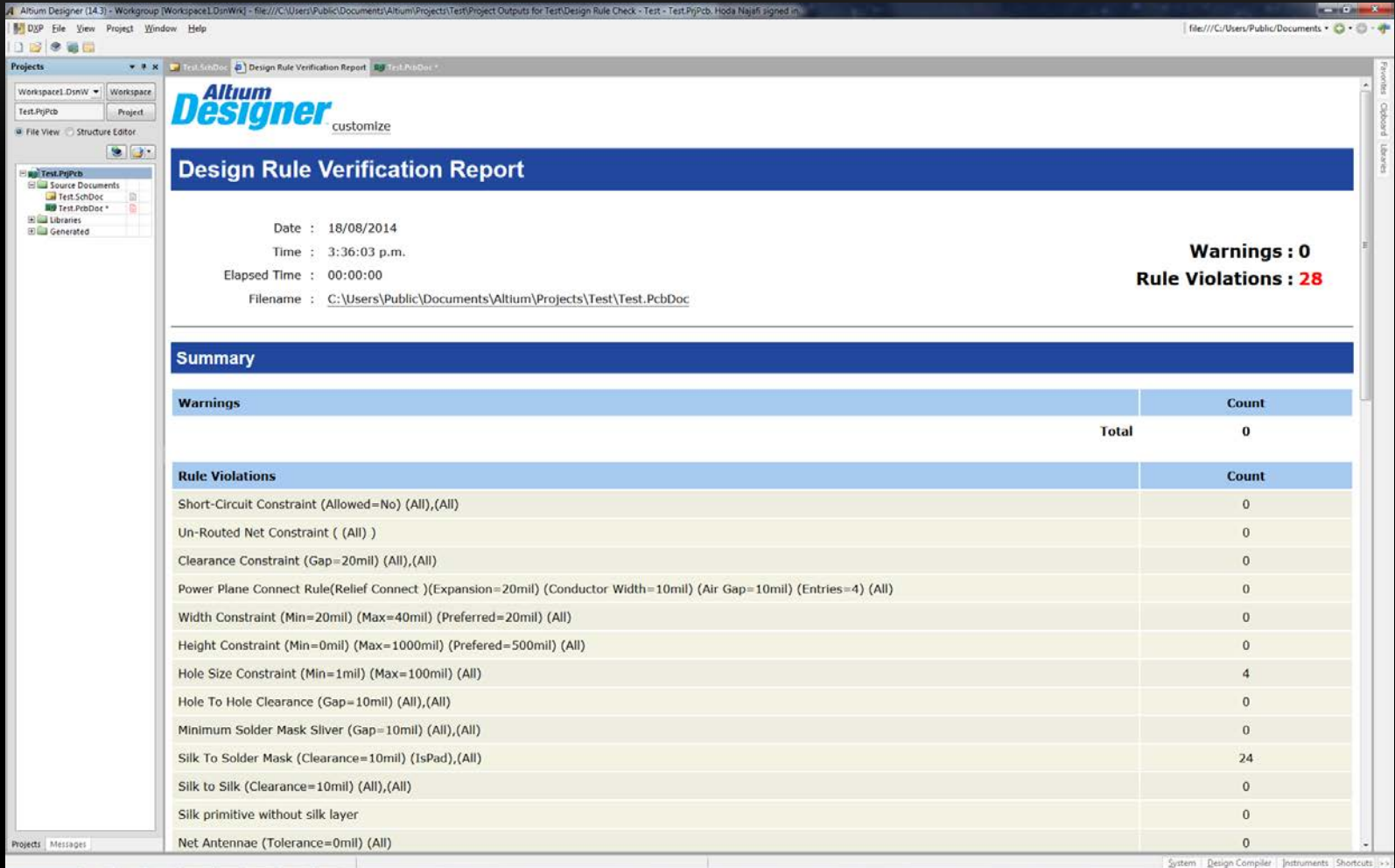
- a. Select 'Keep-Out Layer' from the tabs at the bottom of the designer window; up to this point we have been working from the 'Top Layer'
- b. b. Make sure the Keepout layer is selected then Add a Line using **Place > Line**. This will be similar to redefining the shape of the board where you can zoom in at the corners to make sure you are clicking the right spot and then zoom back out to get to the next corner more quickly.
- c. c. Add the Line to the perimeter of the board and then press **Esc** to stop adding the track. The keepout Line should be pink because it is on the keepout layer.

DESIGN RULE CHECK

- Design Rule Checking (DRC) is a powerful automated feature that checks both the logical and physical integrity of your design. Checks are made against any or all enabled design rules.
- Go to **Tools > Design Rule Check**
- The report lists each rule that was tested, as specified in the *Design Rule Checker* dialog. Each violation that was located is listed with full details of any reference information, such as the layer, net name, component designator and pad number, as well as the location of the object.
- When you run Design Rule Check, a new window will open. ([Figure 22](#))



DESIGN RULE CHECK



The screenshot shows the Altium Designer interface with a Design Rule Verification Report open. The report title is "Design Rule Verification Report" and it includes the following information:

- Date : 18/08/2014
- Time : 3:36:03 p.m.
- Elapsed Time : 00:00:00
- Filename : C:\Users\Public\Documents\Altium\Projects\Test\Test.PcbDoc

Summary statistics:

- Warnings : 0
- Rule Violations : 28

The report contains two tables:

Warnings	Count
Total	0

Rule Violations	Count
Short-Circuit Constraint (Allowed=No) (All),(All)	0
Un-Routed Net Constraint ((All))	0
Clearance Constraint (Gap=20mil) (All),(All)	0
Power Plane Connect Rule(Relief Connect)(Expansion=20mil) (Conductor Width=10mil) (Air Gap=10mil) (Entries=4) (All)	0
Width Constraint (Min=20mil) (Max=40mil) (Preferred=20mil) (All)	0
Height Constraint (Min=0mil) (Max=1000mil) (Preferred=500mil) (All)	0
Hole Size Constraint (Min=1mil) (Max=100mil) (All)	4
Hole To Hole Clearance (Gap=10mil) (All),(All)	0
Minimum Solder Mask Silver (Gap=10mil) (All),(All)	0
Silk To Solder Mask (Clearance=10mil) (IsPad),(All)	24
Silk to Silk (Clearance=10mil) (All),(All)	0
Silk primitive without silk layer	0
Net Antennae (Tolerance=0mil) (All)	0

Figure 22

ALTIUM PCB POLYGON POUR INSTRUCTION

After your PCB has been routed and you are ready to pour copper on the top and bottom layers:

- Click the Place tool bar button and select "PolyGon Pour...".
- Set the layer to Top Layer and also set the "Connect to Net" to your ground net. Tick 'Remove Dead copper' and 'Is Poured' then Click OK. [\(F.20\)](#)
- Your mouse pointer will change to a cross hair. Use this pointer to draw a box around your PCB. After drawing a box around your PCB right click your mouse to exit the draw mode.
- Altium will fill the empty areas of your top layer on your PCB with copper connected to ground.
- We need to repeat this procedure for the bottom layer.

ALTIUM PCB POLYGON POUR INSTRUCTION

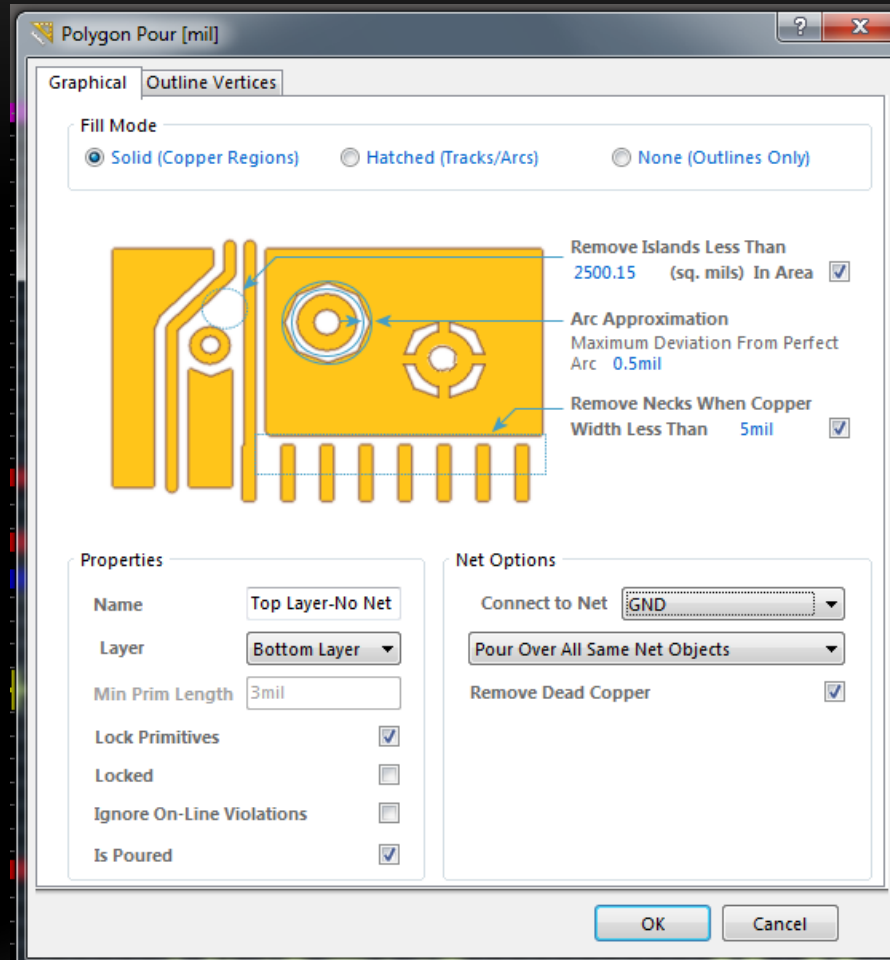
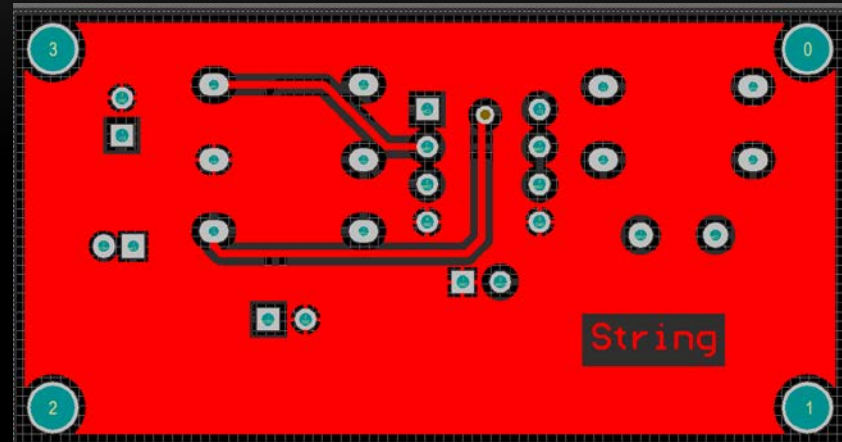


Figure 20

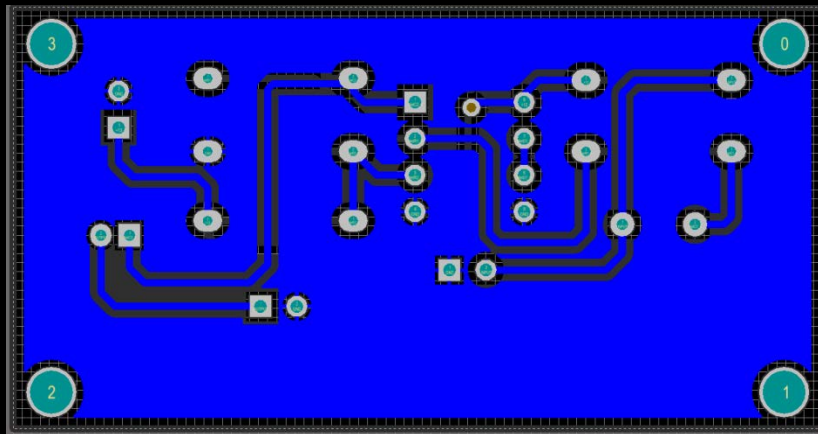
ALTIUM PCB POLYGON POUR INSTRUCTION

- The board will be like image below:

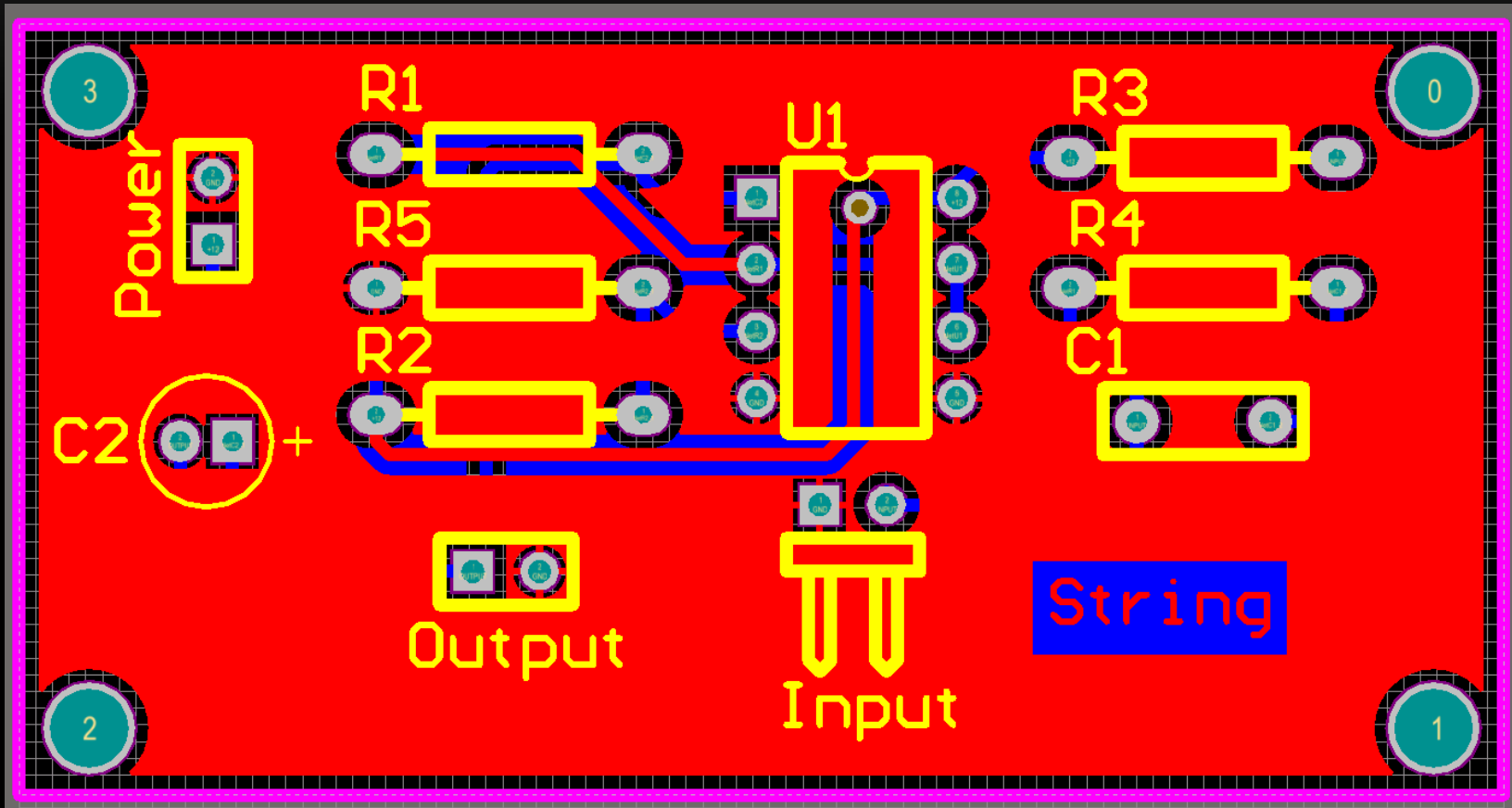
Top Layer



Bottom Layer



FINAL VIEW



ANY QUESTION?
