

تطبيقات ميكاترونك -1-

Lecture No. 3

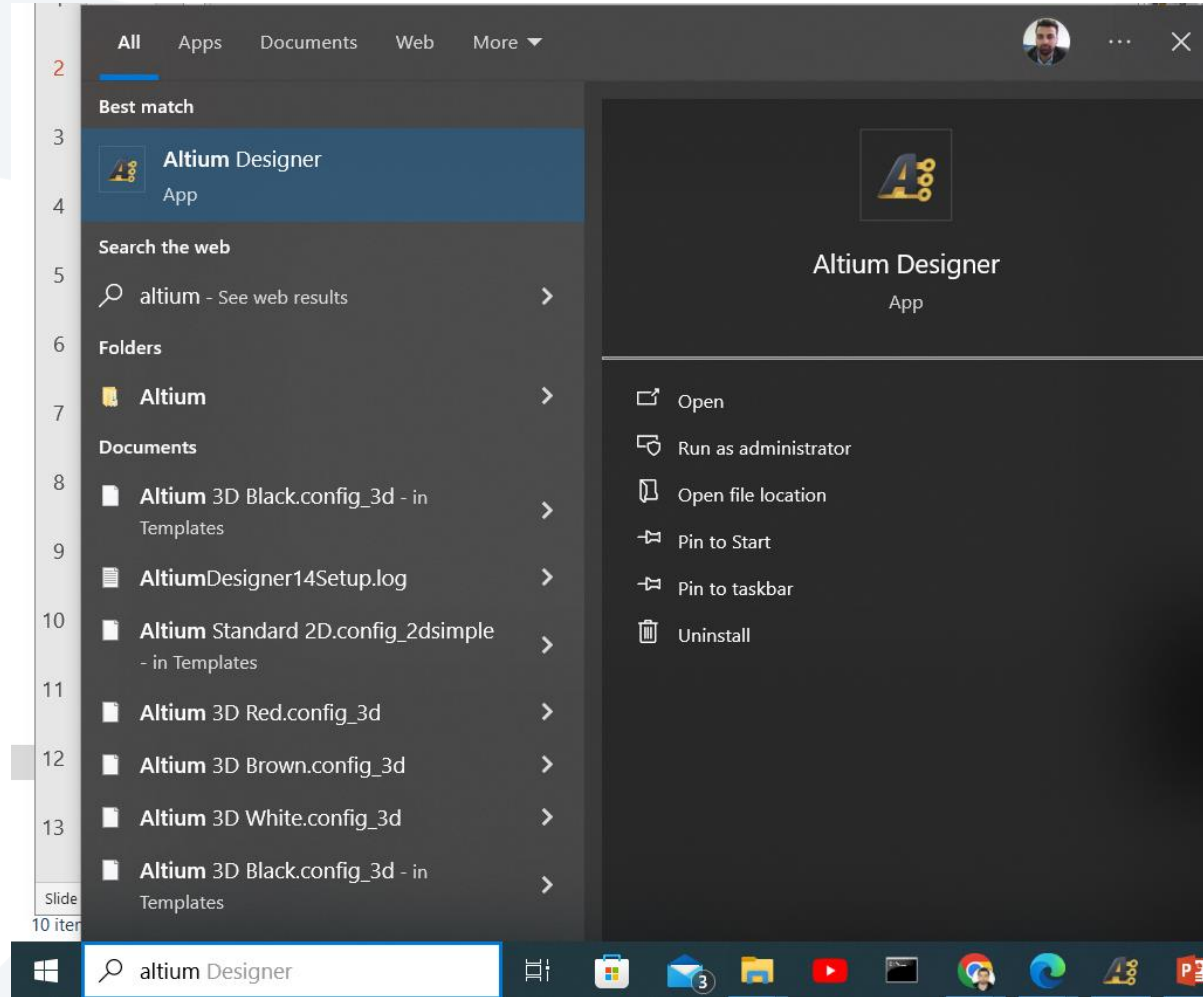
PCB SCH by Altium Designer

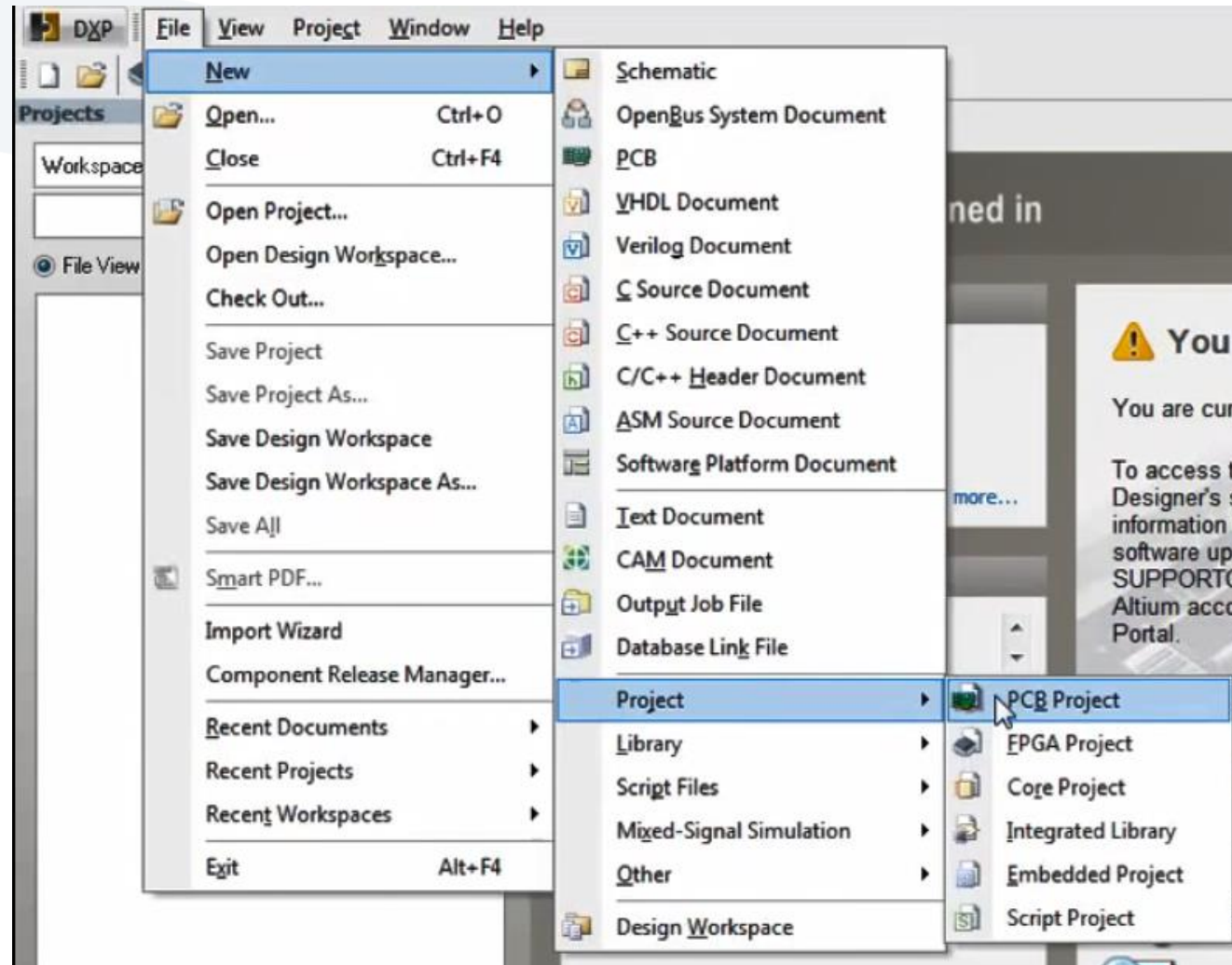
روبوت و أنظمة ذكية - سنة ثالثة

Dr. Eng. Essa Alghannam

Dr. Eng Fadi Mtaweg

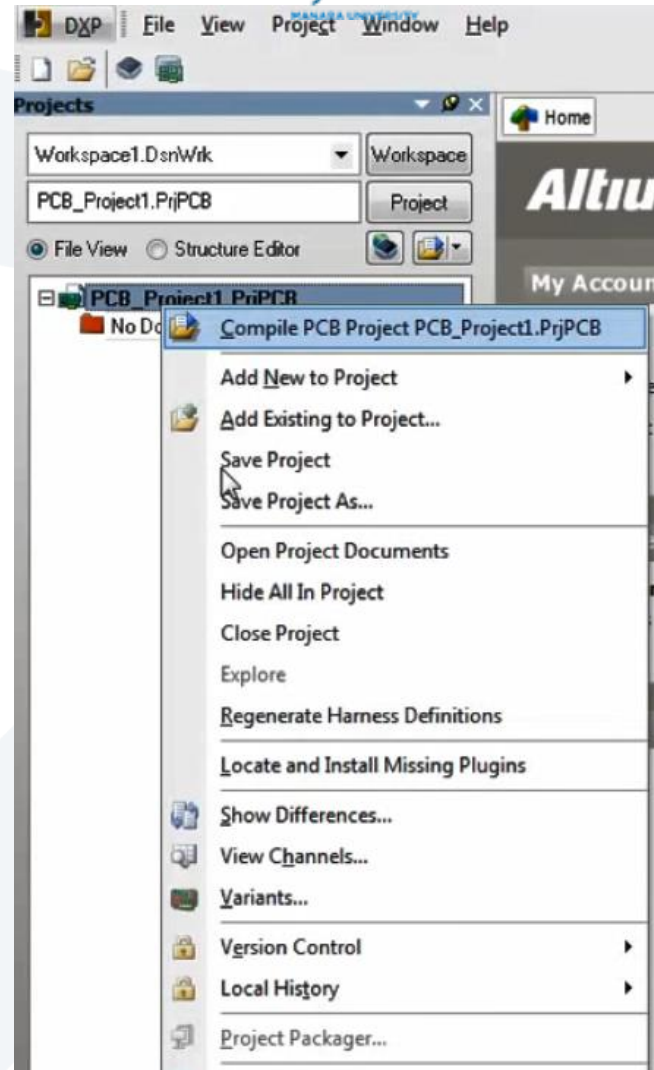
2022-2023

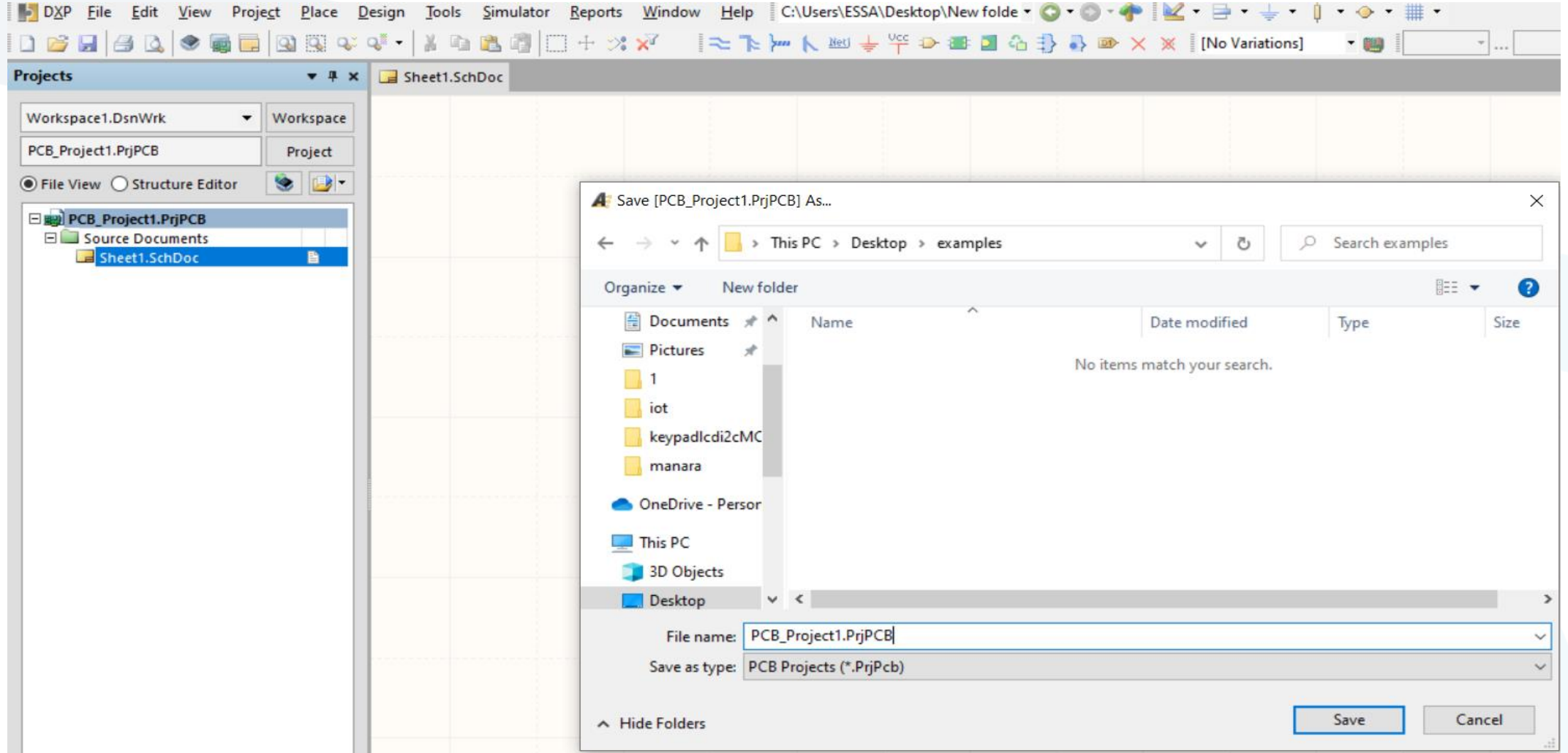




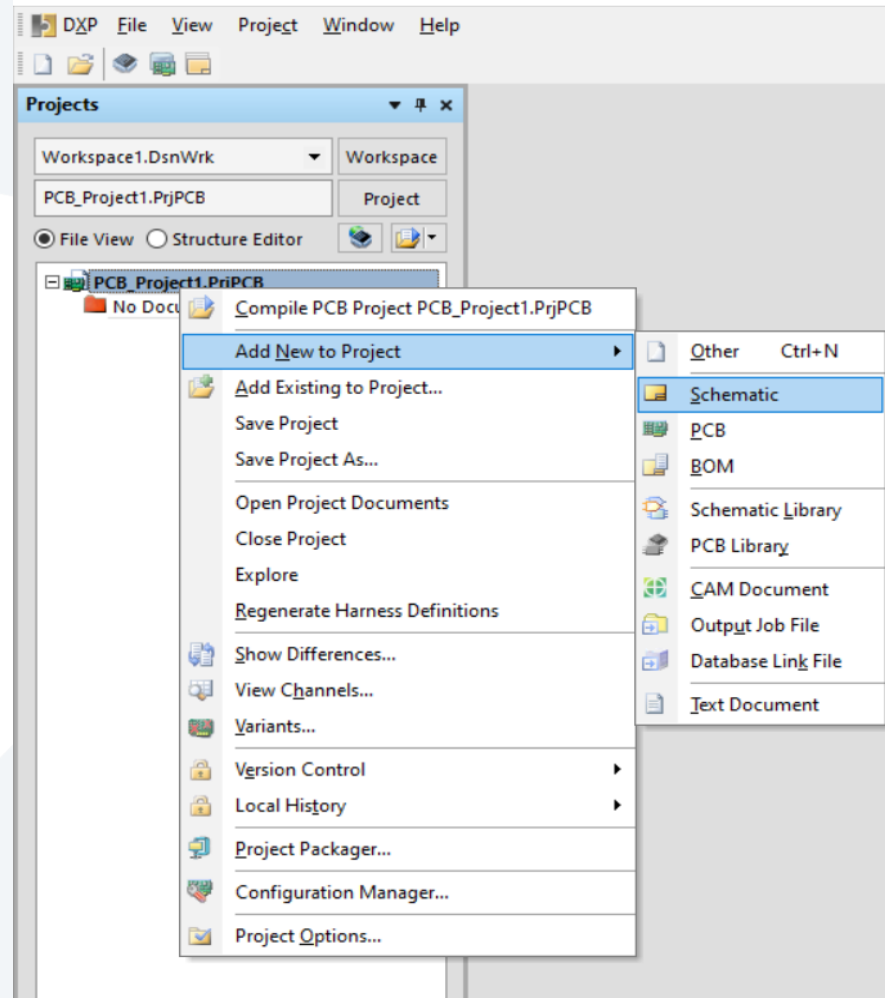


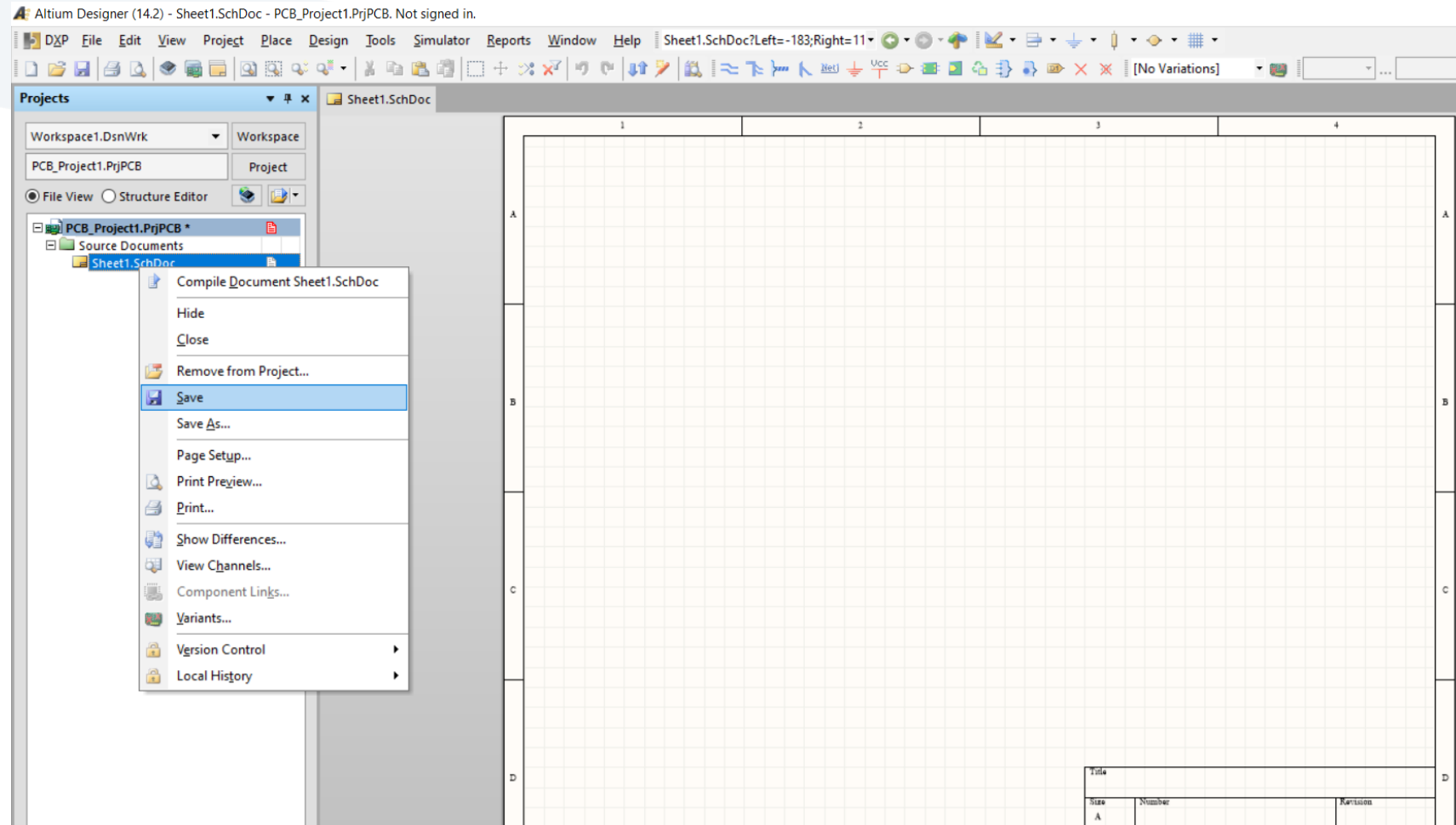
جامعة
المنارة

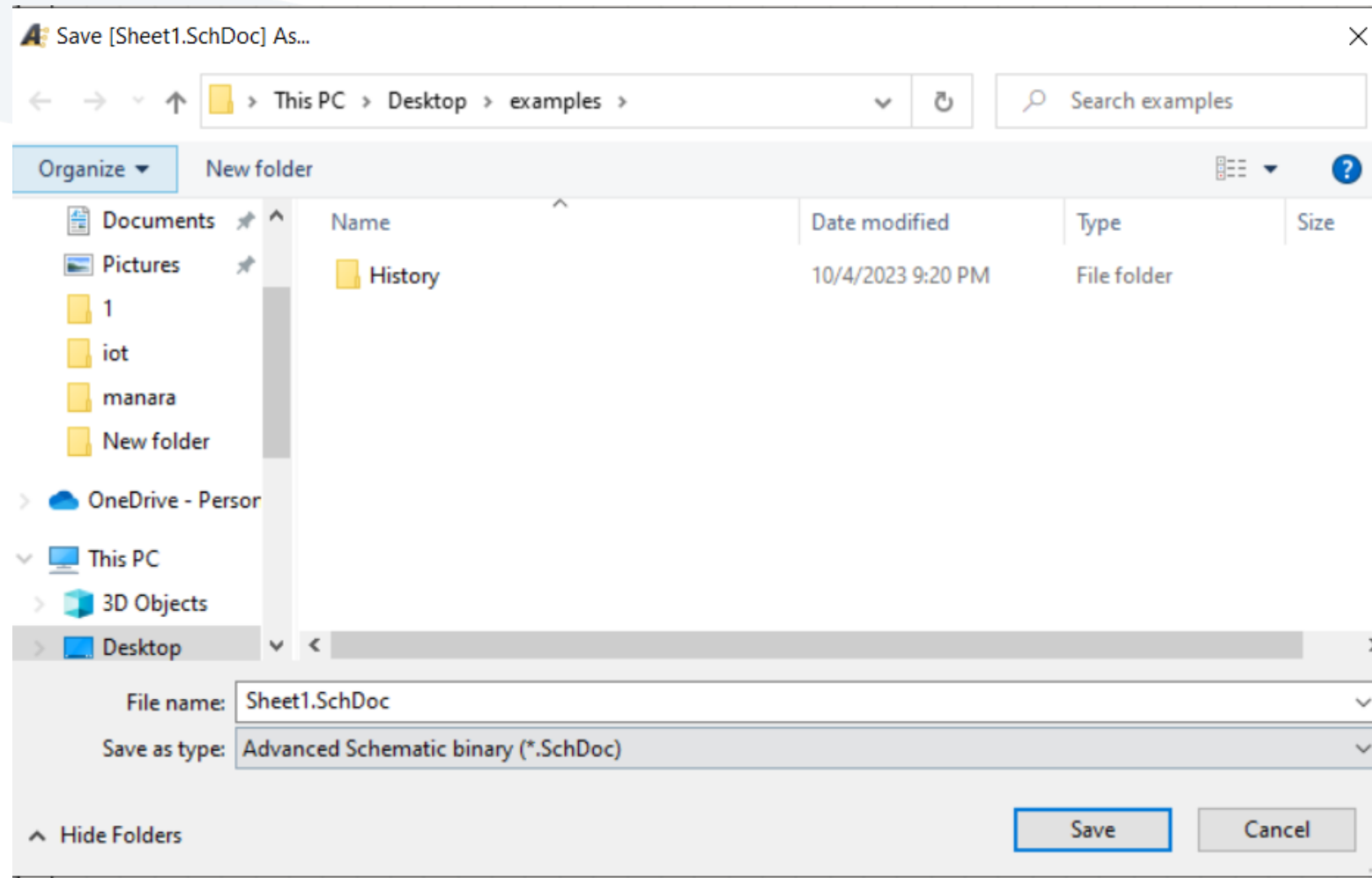


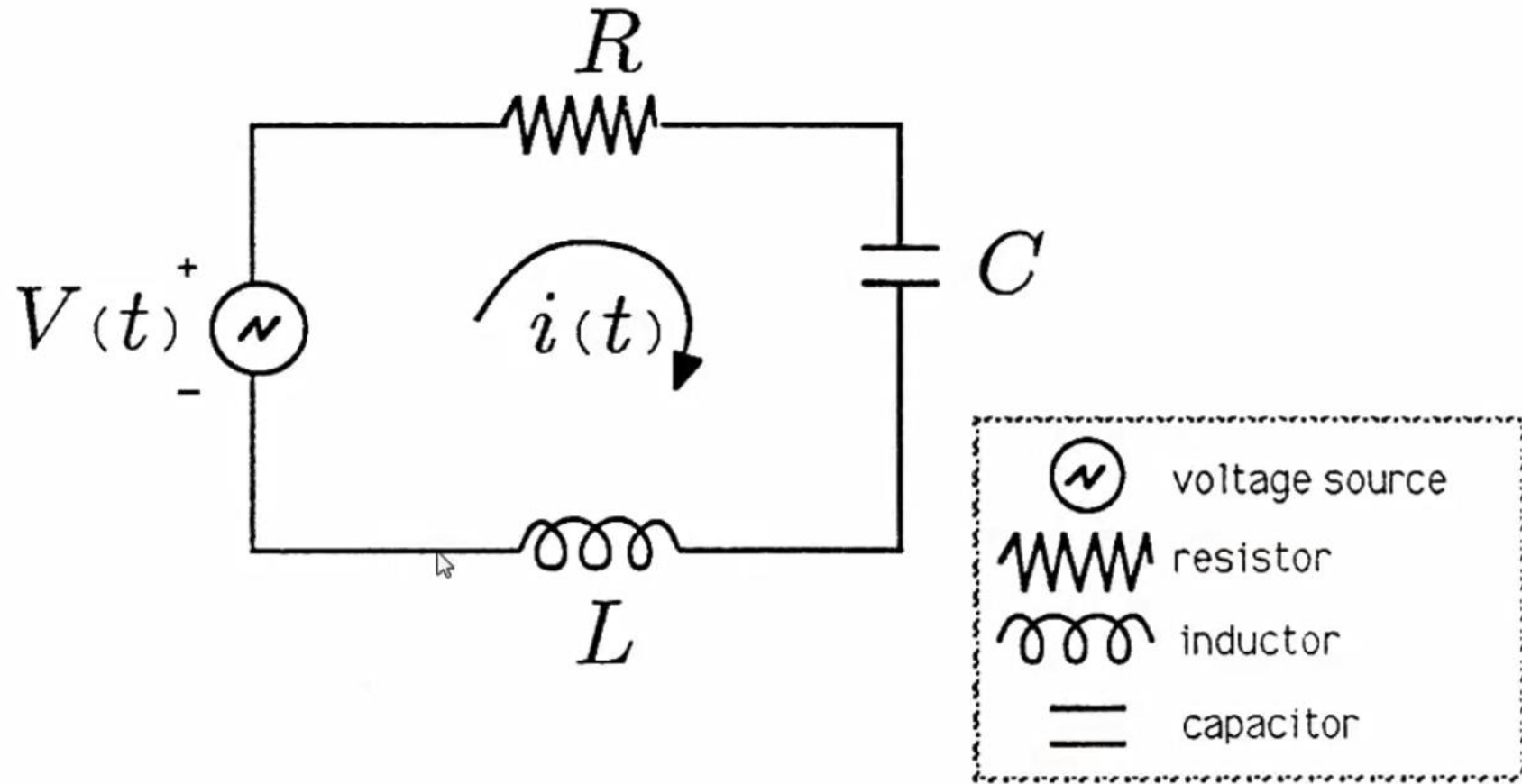


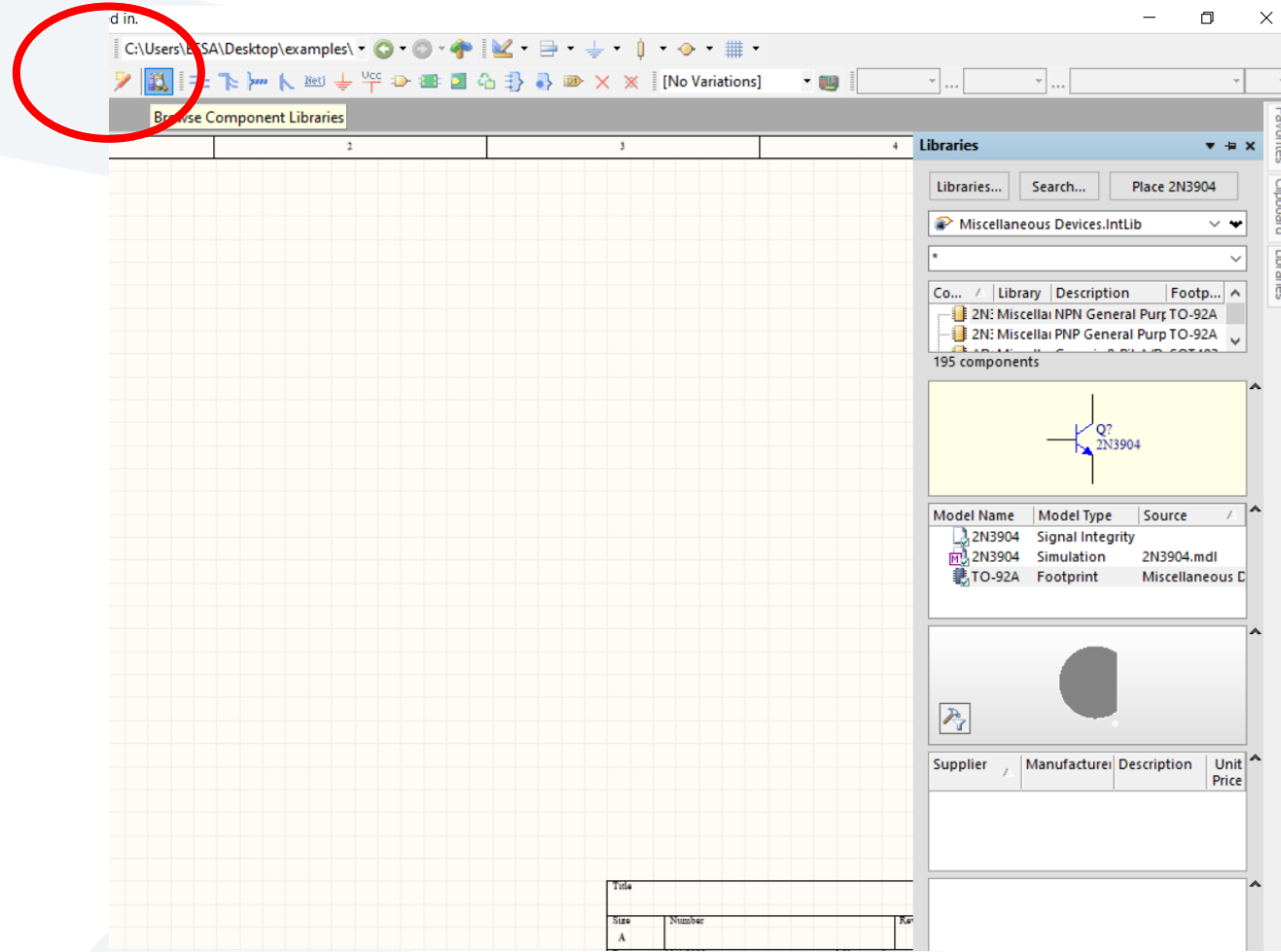
Altium Designer (14.2) - PCB_Project1.PrjPCB. Not signed in.

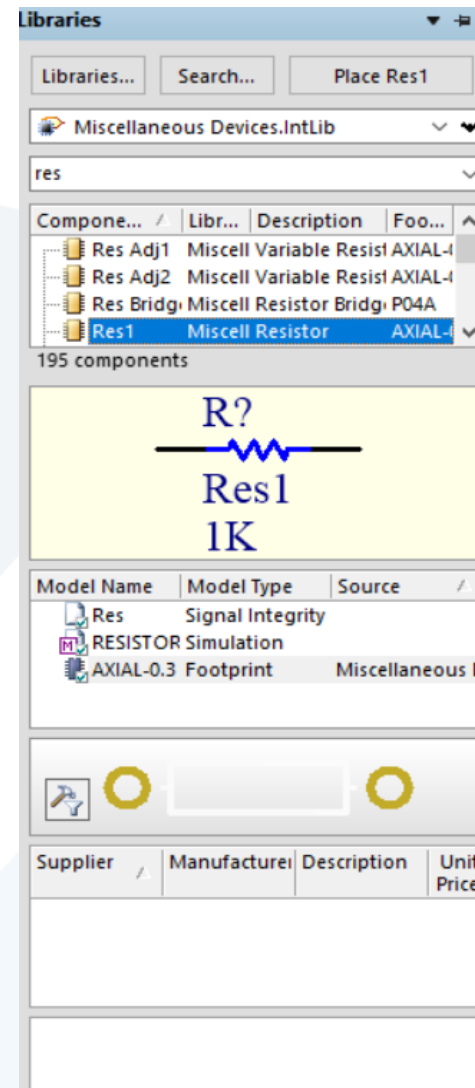
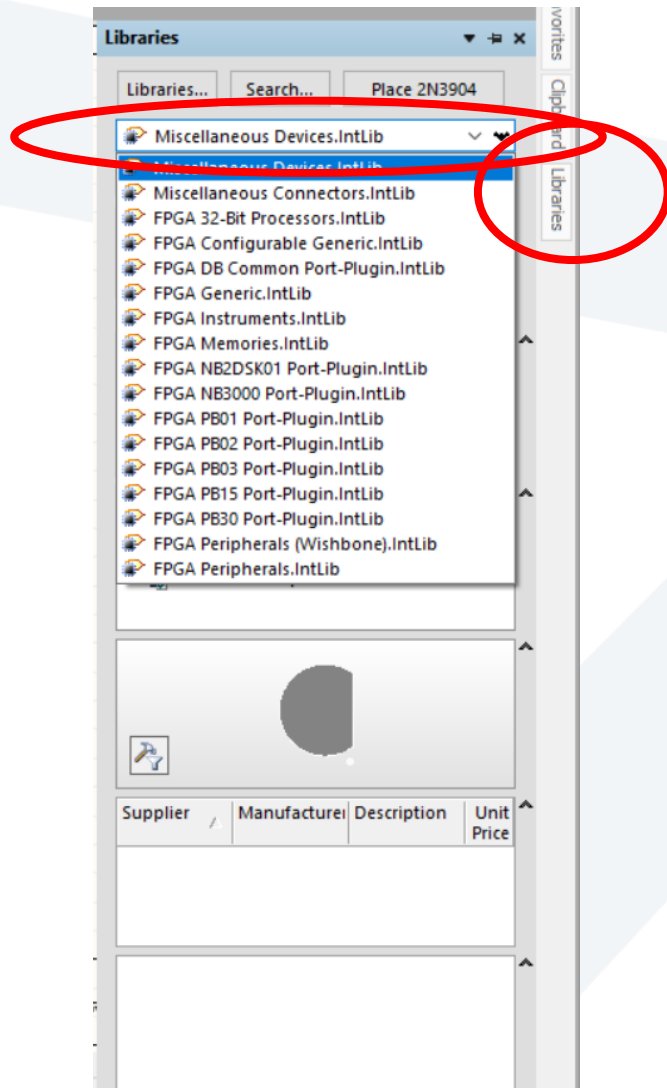


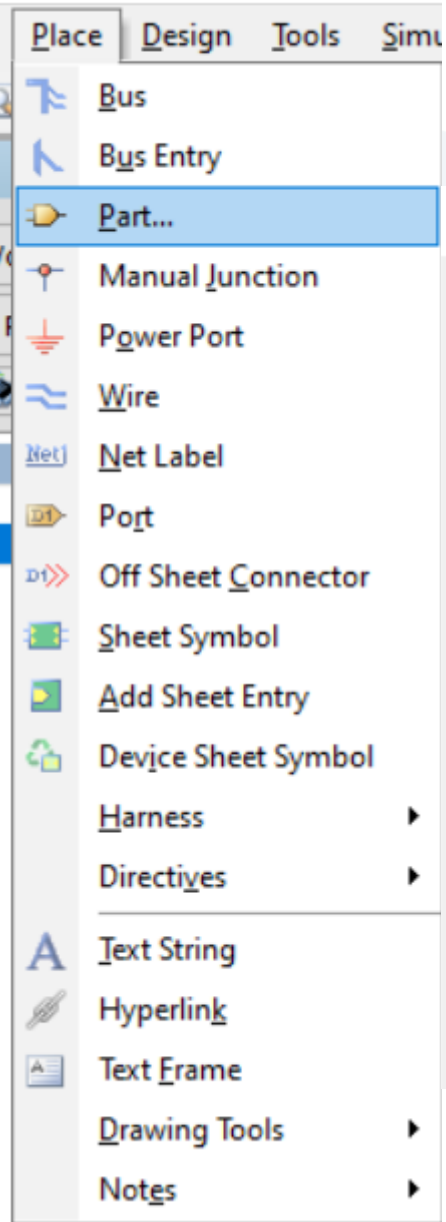












Place Part

Part Details

Physical Component:

Logical Symbol:

Designator:

Comment:

Footprint:

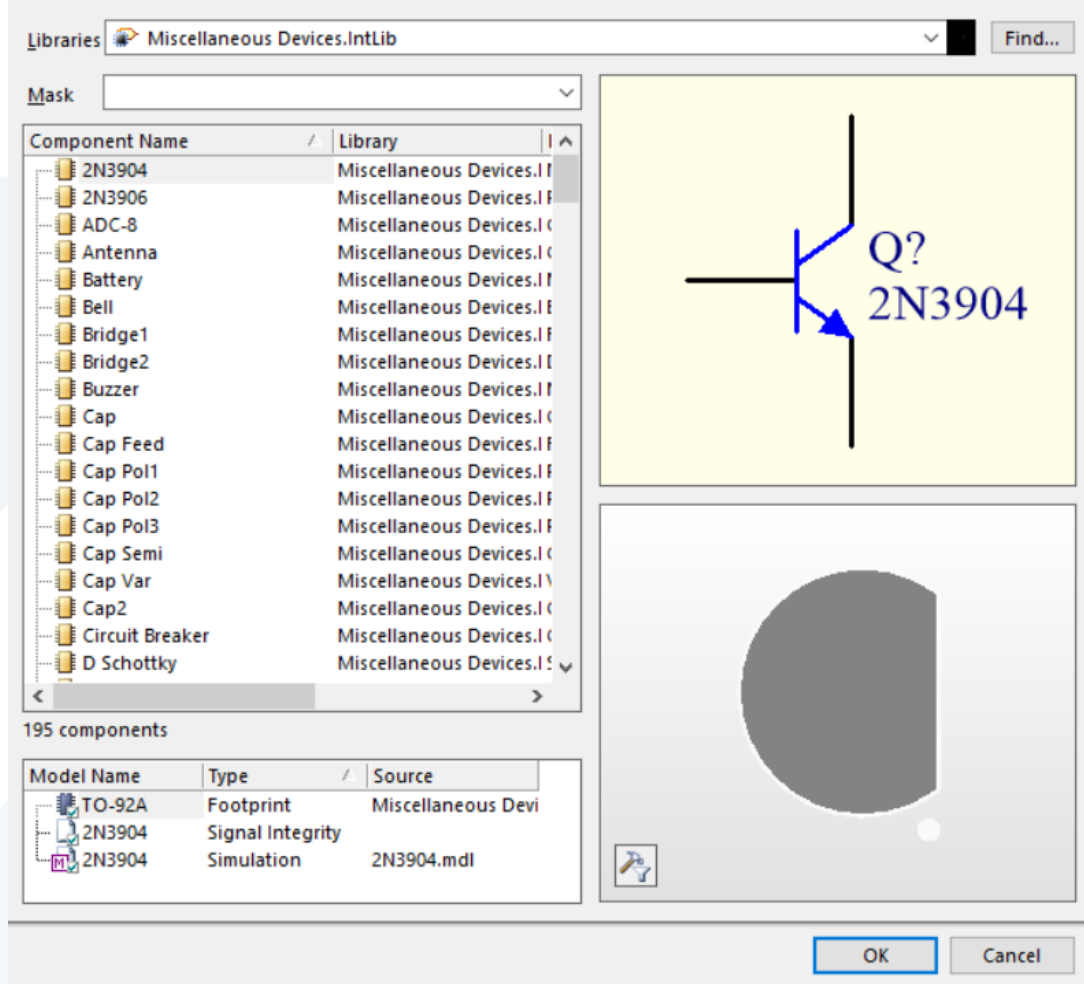
Part ID:

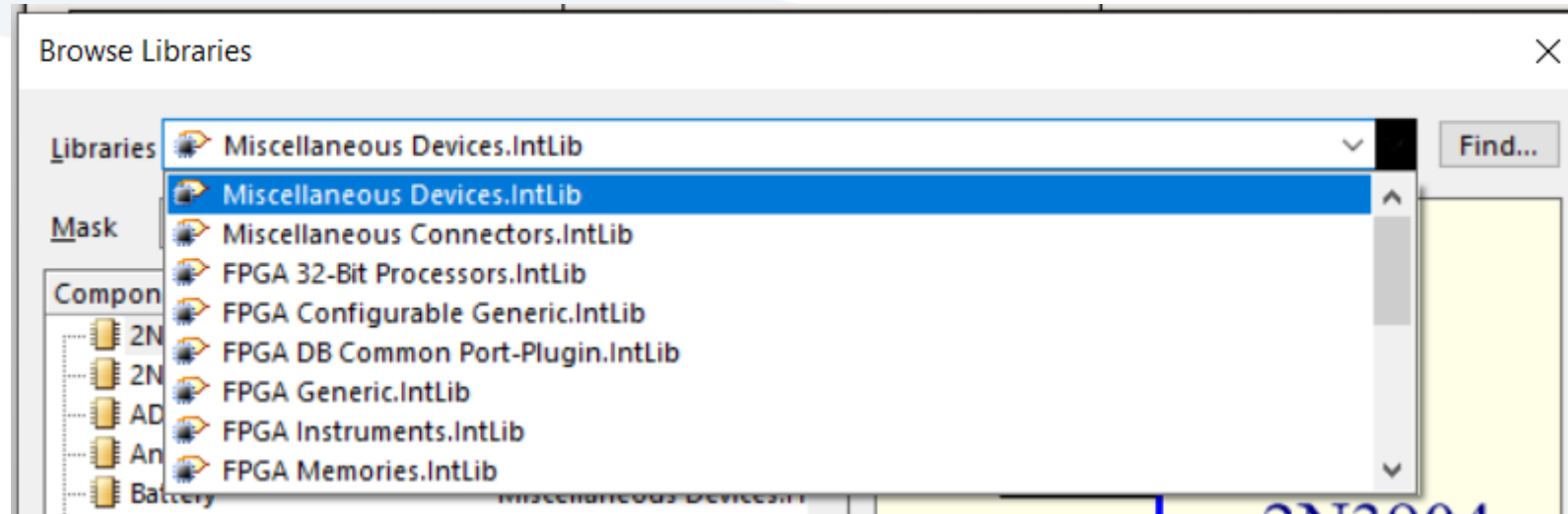
Library:

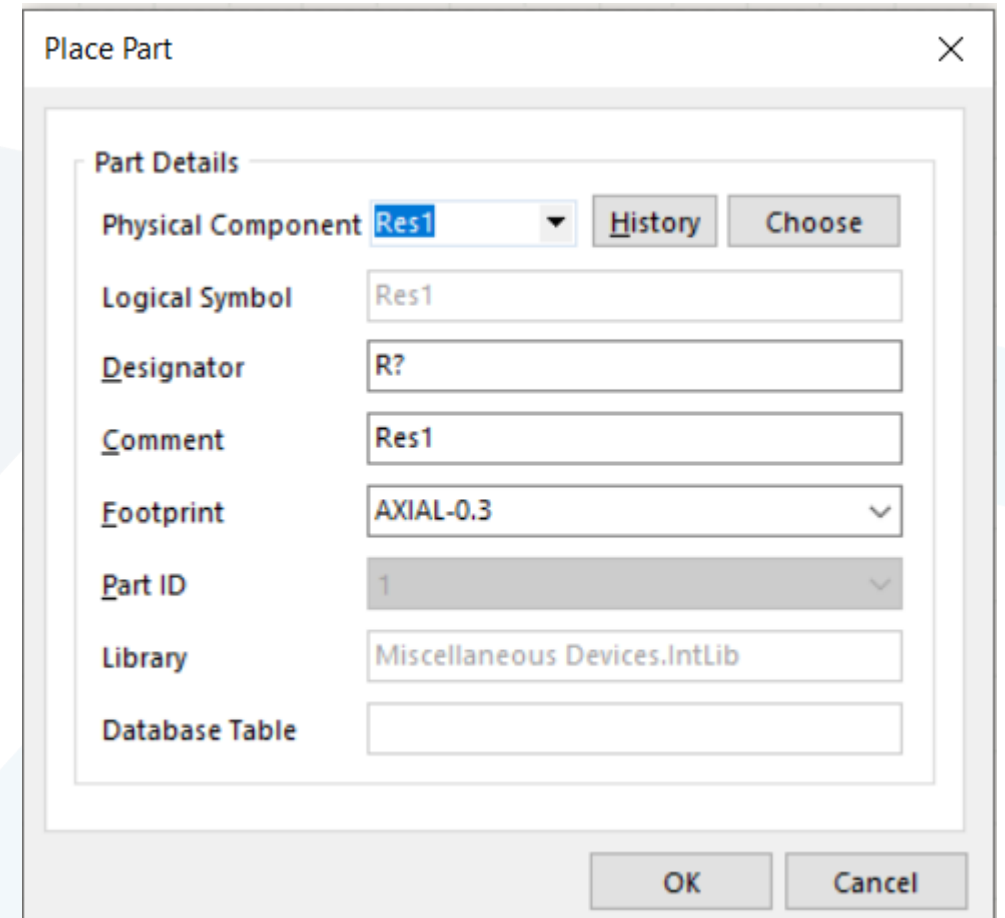
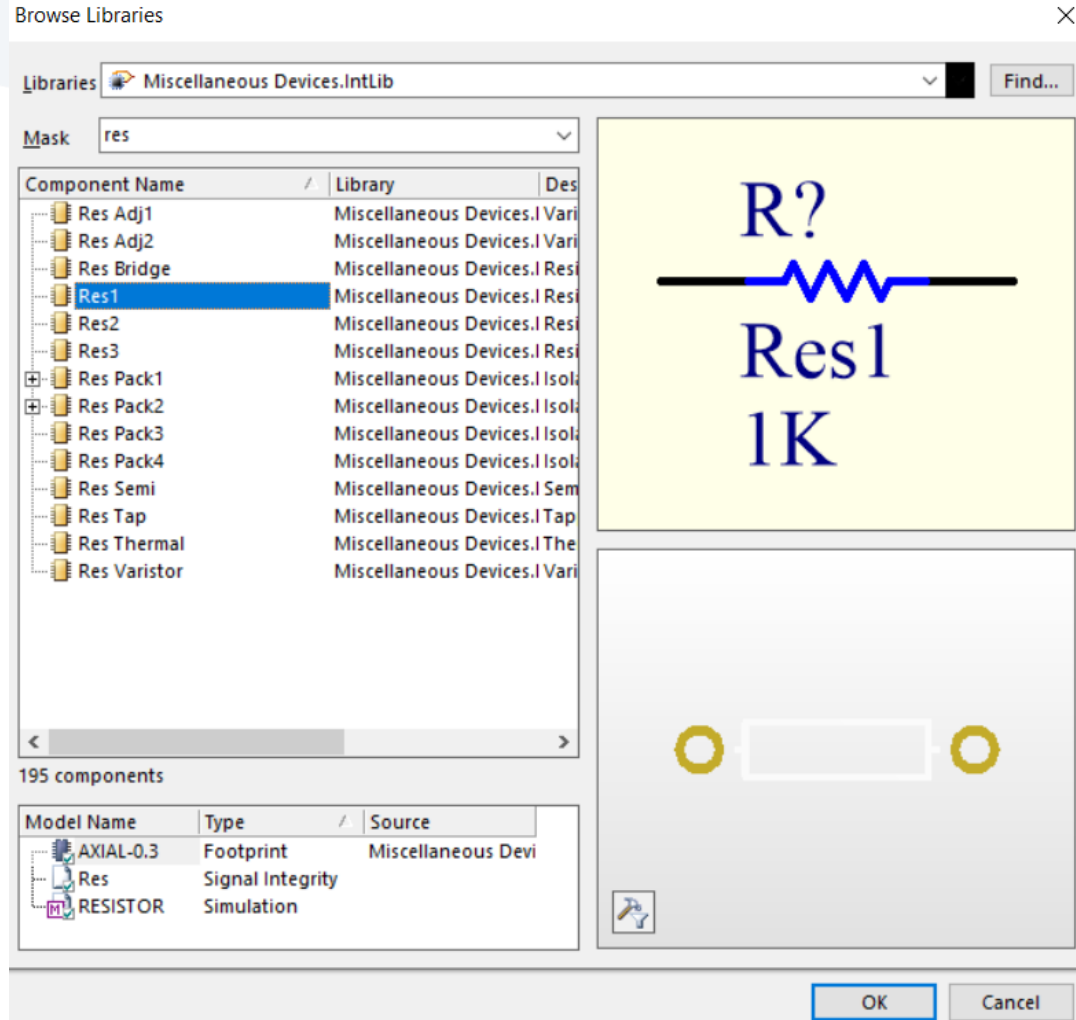
Database Table:

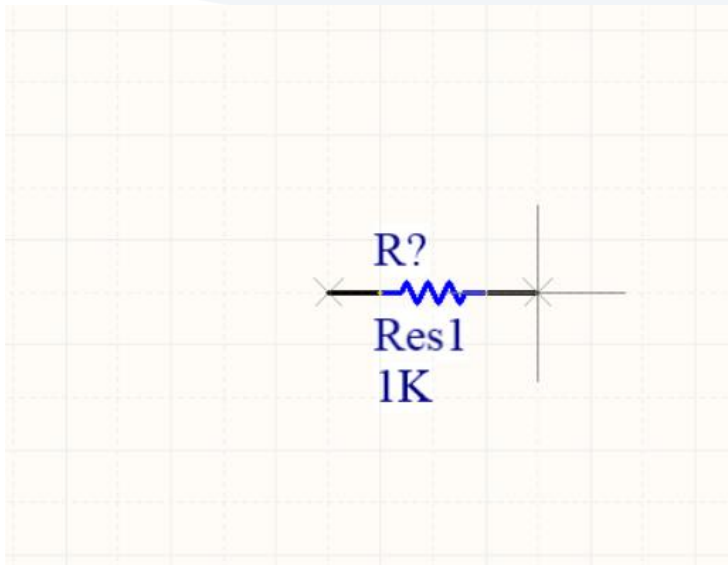
p+p

Browse Libraries









Zoom: click and hold wheel and move mouse

Rotate: space

left click to place

**Esc to end and show the place part window
again**

Browse Libraries

Libraries: Miscellaneous Devices.IntLib Find...

Mask: cap

Component Name	Library	Description
Cap	Miscellaneous Devices.IntLib	Capacitor
Cap Feed	Miscellaneous Devices.IntLib	Capacitor Feed
Cap2	Miscellaneous Devices.IntLib	Capacitor
Cap Pol1	Miscellaneous Devices.IntLib	Capacitor Polarized
Cap Pol2	Miscellaneous Devices.IntLib	Capacitor Polarized
Cap Pol3	Miscellaneous Devices.IntLib	Capacitor Polarized
Cap Semi	Miscellaneous Devices.IntLib	Capacitor Semi
Cap Var	Miscellaneous Devices.IntLib	Capacitor Variable

195 components

Model Name	Type	Source
RAD-0.3	Footprint	Miscellaneous Devi
Cap	Signal Integrity	
CAP	Simulation	

Part Details

Physical Component: Cap

Logical Symbol: Cap

Designator: C?

Comment: Cap

Footprint: RAD-0.3

Part ID: 1

Library: Miscellaneous Devices.IntLib

Database Table:

OK Cancel

Place Part

Part Details

Physical Component: Cap

Logical Symbol: Cap

Designator: C?

Comment: Cap

Footprint: RAD-0.3

Part ID: 1

Library: Miscellaneous Devices.IntLib

Database Table:

OK Cancel

- Before place it by left click
Click tab to show this
window

Properties for Schematic Component in Sheet [Sheet1.SchDoc]

Properties

Designator: ☒ Visible ☐ Locked

Comment: Cap ☒ Visible

Description: Capacitor

Unique Id: RPRJHMKA

Type: Standard

Link to Library Component ☐ Use Vault Component

Design Item ID: Cap

☒ Library Name: Miscellaneous Devices.IntLib

☐ Table Name:

Graphical

Location X: 350 Y: 420

Orientation: 90 Degrees ☐ Locked

Mode: Normal ☒ Lock Pins ☐ Mirrored

☐ Show All Pins On Sheet (Even if Hidden)

☐ Local Colors

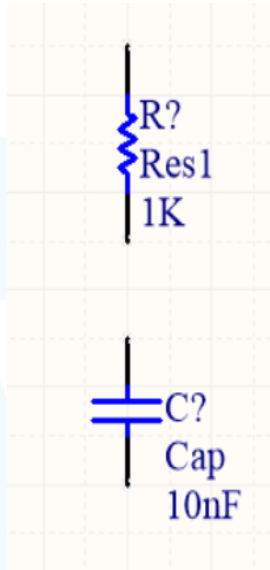
Parameters

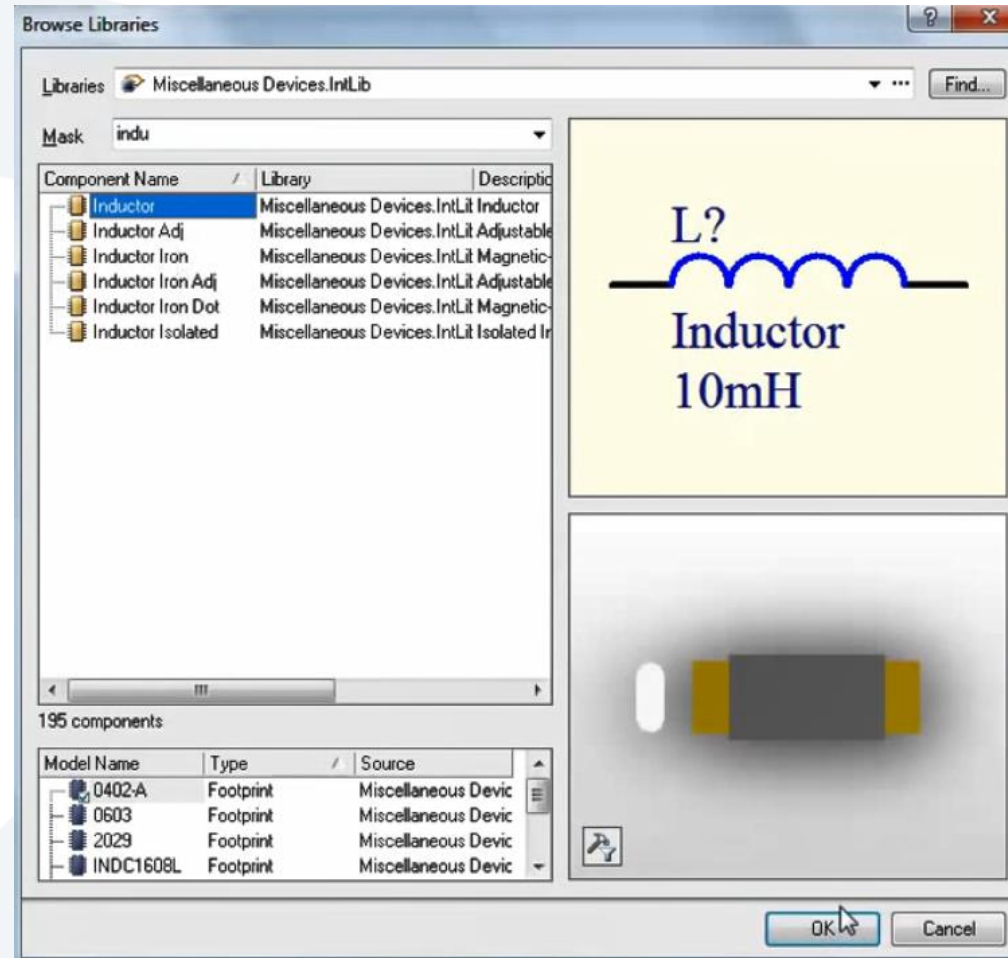
Visible	Name	Value	Type
<input type="checkbox"/>	LatestRevisionDate	17-Jul-2002	STRING
<input type="checkbox"/>	LatestRevisionNote	Re-released for DXP Platform.	STRING
<input type="checkbox"/>	PackageDescription	Radial Cap, Thru-Hole; 2 Leads; 0.3	STRING
<input type="checkbox"/>	PackageReference	RAD-0.3	STRING
<input type="checkbox"/>	Published	8-Jun-2000	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING
<input checked="" type="checkbox"/>	Value	100pF	STRING

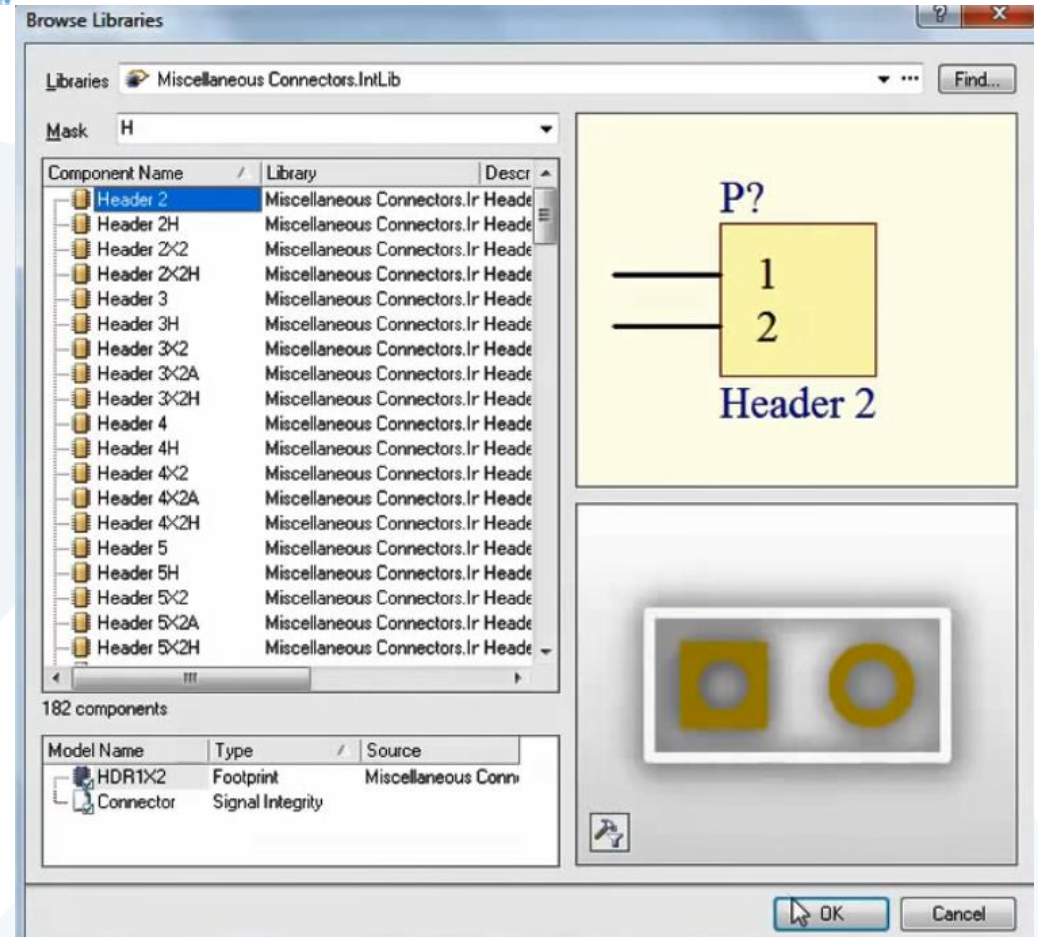
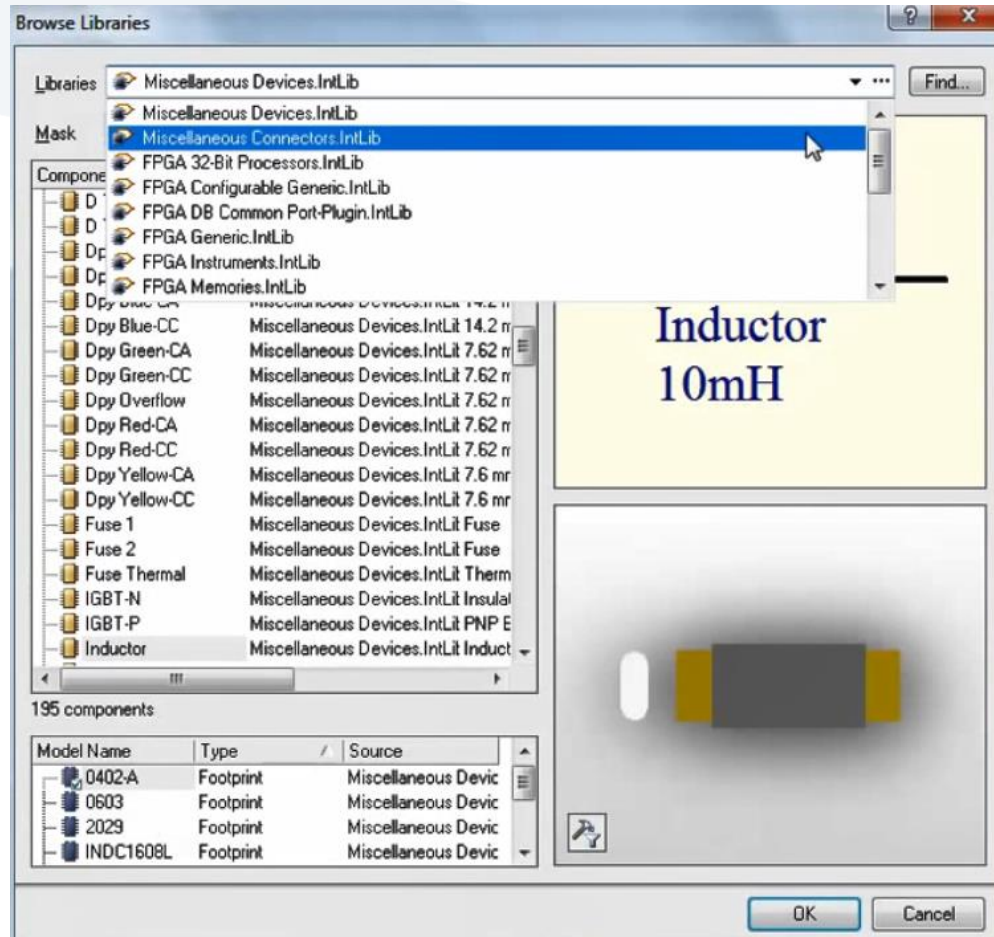
Models

Name	Type	Description	Vault	Item Rev...	Revision...
RAD-0.3	Footprint	Radial Cap, Thru-Hole; 2 Leads; 0			[Unknown]
Cap	Signal Inte				[Unknown]
CAP	Simulation	Capacitor			[Unknown]

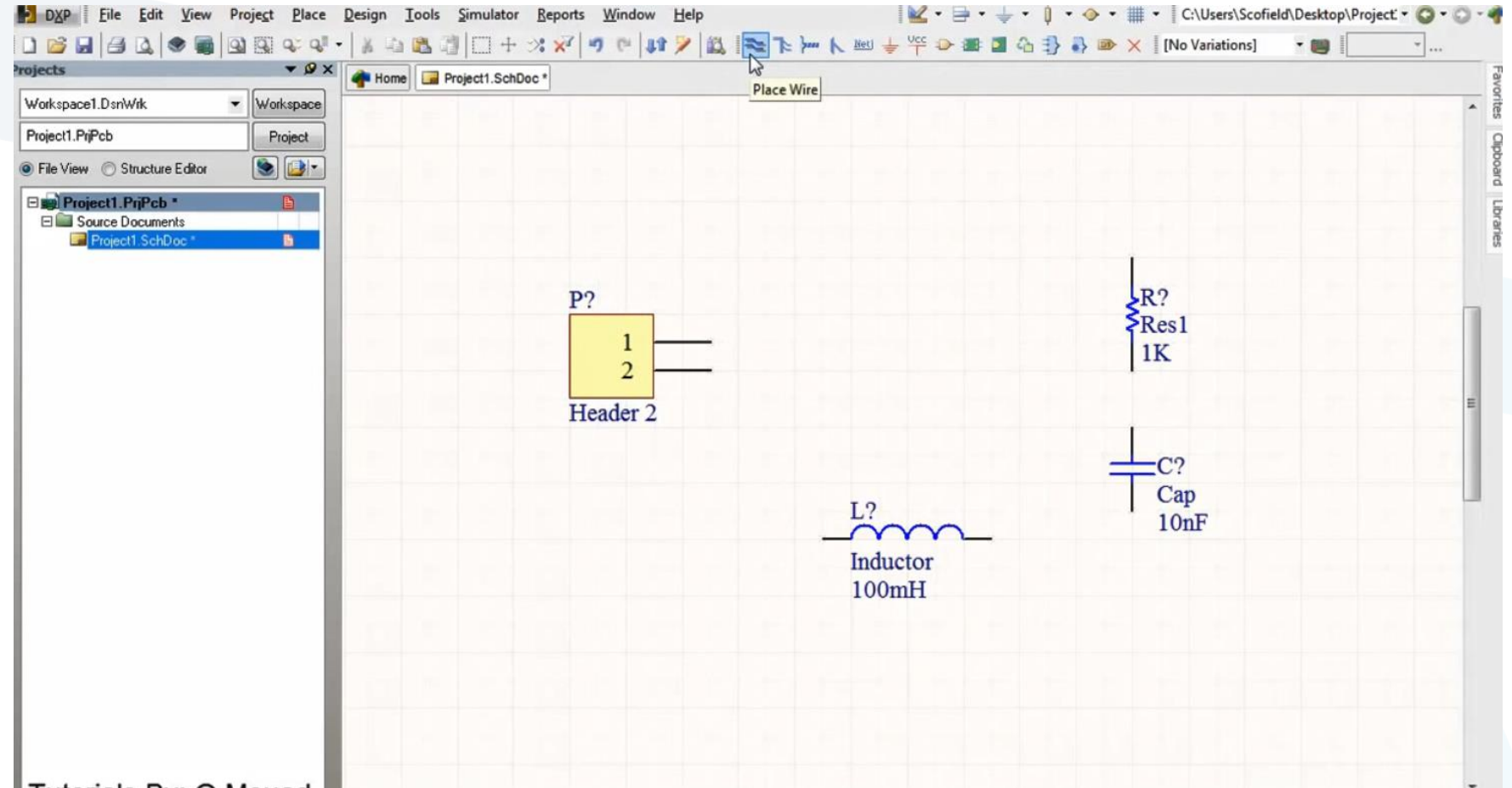
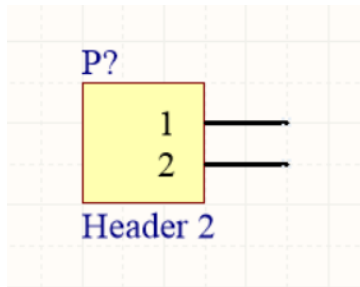
Visible	Name	Value	Type
<input checked="" type="checkbox"/>	Value	10nF	STRING

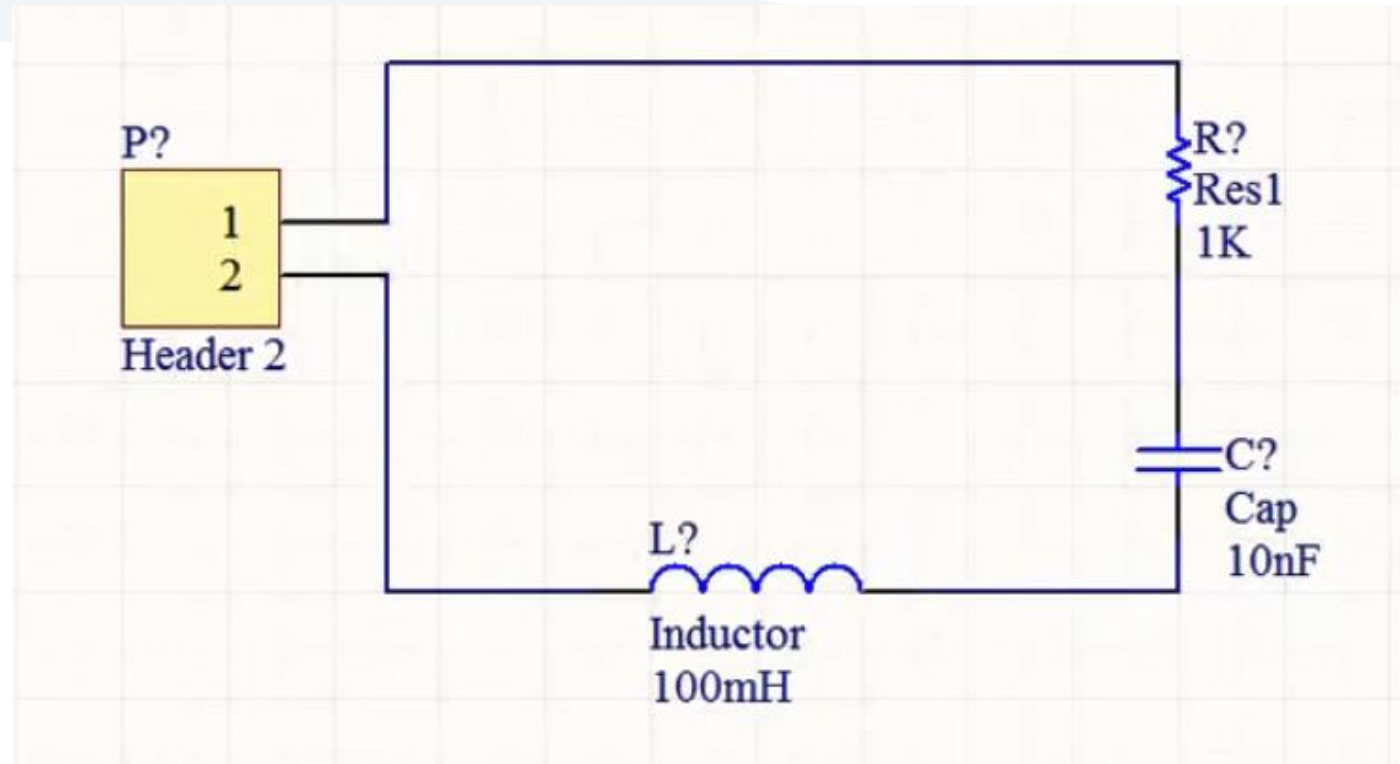


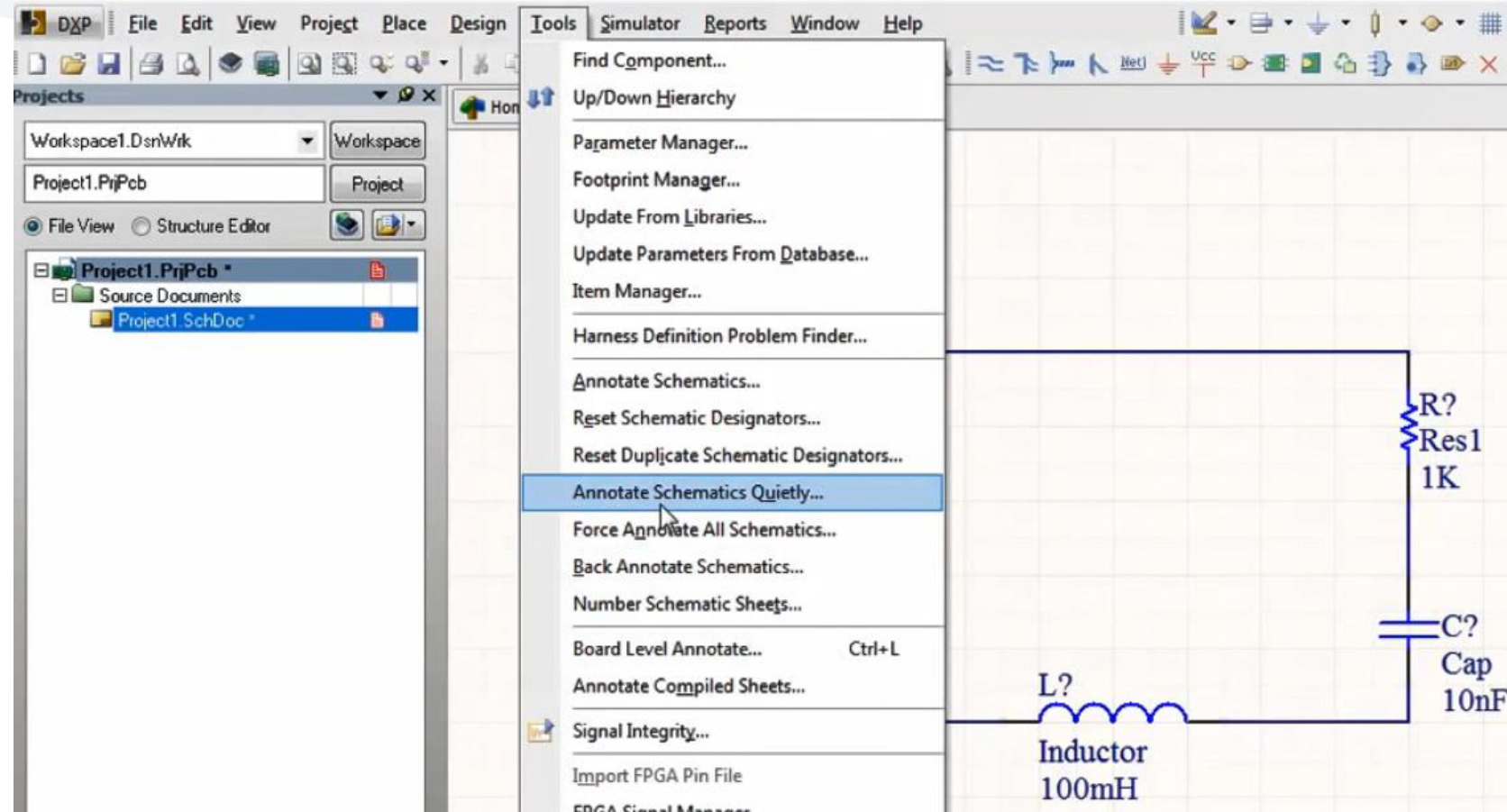


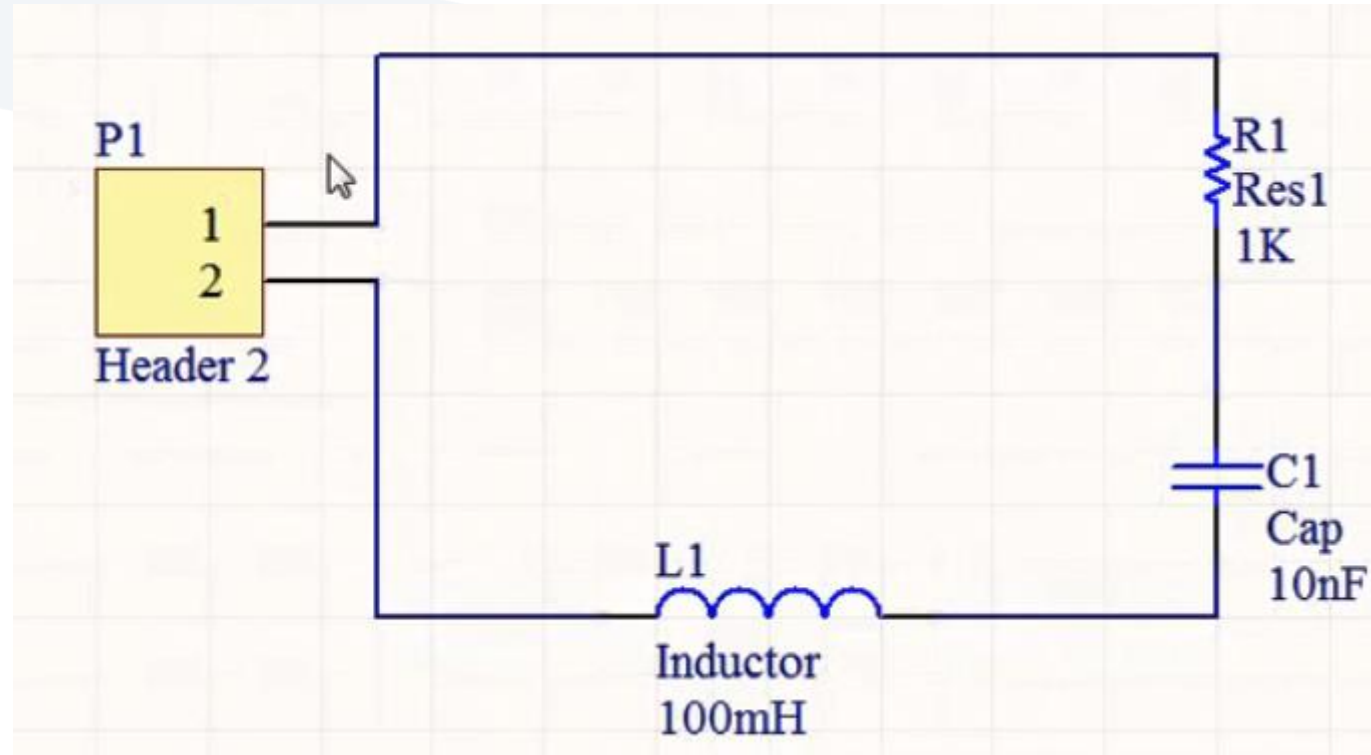


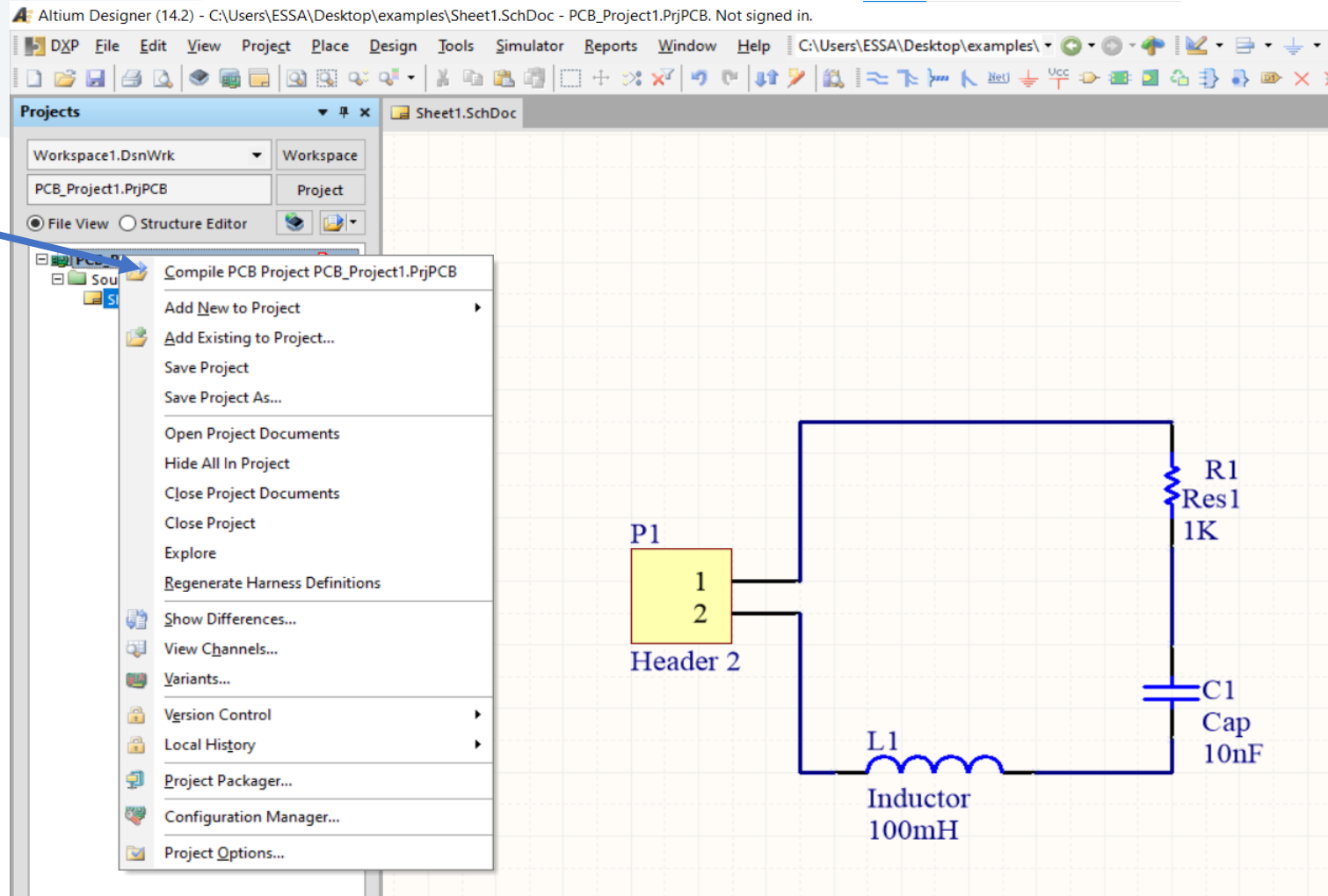
Click x to flip the component

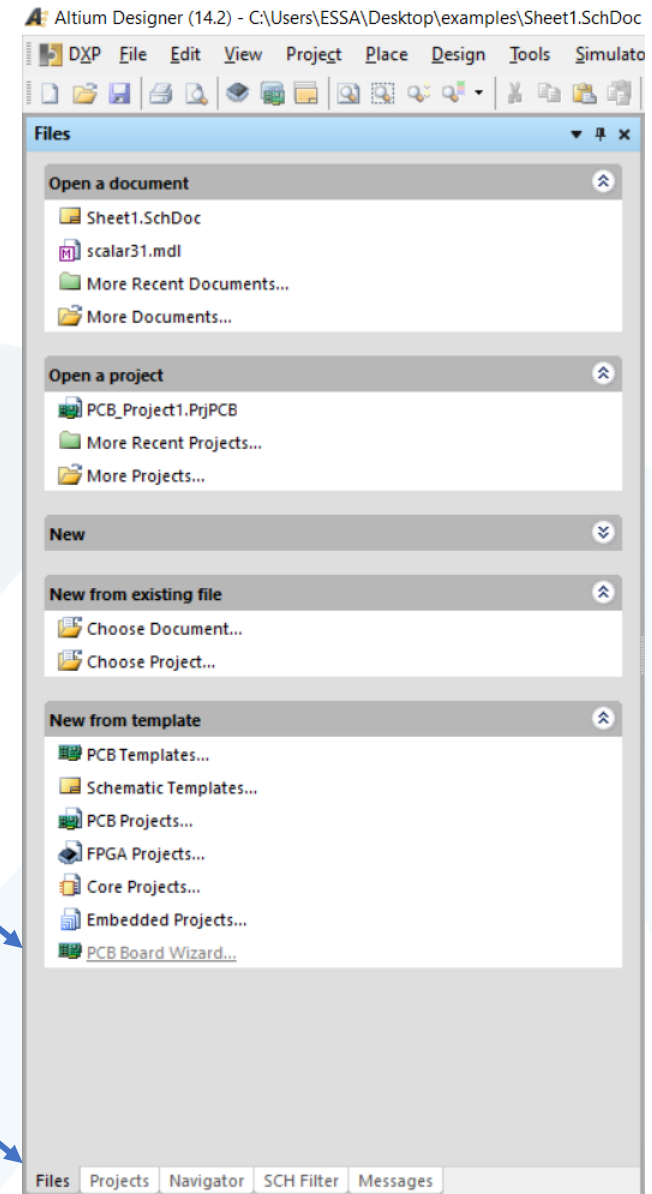
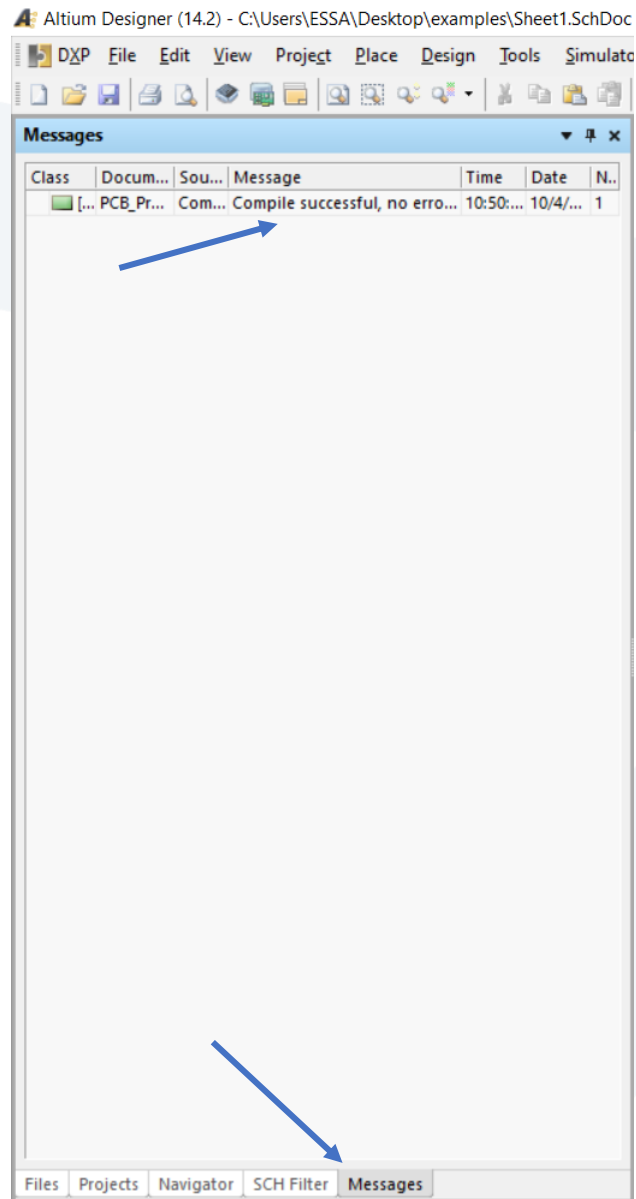












PCB Board Wizard



Altium Designer 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.

Cancel

< Back

Next >

Finish

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.

- ☐ Imperial
☒ Metric

Cancel

< Back

Next >

Finish

PCB Board Wizard

Choose Board Profiles

Select a specific board type from 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 4.8 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 4.5 inches)
- AT short bus with break-away tab (7 x 4.8 inches)
- B
- C

Cancel

< Back

Next >

Finish

PCB Board Wizard

Choose Board Details

Choose Board Details

Outline Shape:

☒ Rectangular

☐ Circular

☐ Custom

Board Size:

Width: 40.0 mm

Height: 40.0 mm

Dimension Layer: Mechanical Layer 1

Boundary Track Width: 0.3 mm

Dimension Line Width: 0.3 mm

Keep Out Distance From Board Edge: 1.3 mm

☒ Title Block and Scale

☒ Legend String

☒ Dimension Lines

☐ Corner Cutoff

☐ Inner CutOff

Cancel < Back Next > Finish

PCB Board Wizard

Choose Board Layers

Set the number of signal layers and power planes suitable for your design.

Signal Layers:

2

Power Planes:

0

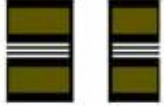
Cancel < Back Next > Finish

PCB Board Wizard

Choose Via Style

Choose the routing via style that is suitable for your design.

☒ Thruhole Vias only
☐ Blind and Buried Vias only



Cancel < Back Next > Finish

PCB Board Wizard

Choose Component and Routing Technologies

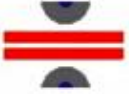
Choose the component and routing style that you intend to use

The board has mostly:

☐ Surface-mount components.
☒ Through-hole components.

Number of tracks between adjacent pads





☐ One Track
☒ Two Track
☐ Three Track




Cancel < Back Next > Finish

PCB Board Wizard

Choose Default Track and Via sizes
Choose the minimum track size, via size and the copper to copper clearances to use on the new board.

Minimum Track Size	<u>0.2 mm</u>	
Minimum Via Width	<u>1.0 mm</u>	
Minimum Via Hole Size	<u>0.6 mm</u>	
Minimum Clearance	<u>0.2 mm</u>	



Cancel < Back Next > Finish

PCB Board Wizard

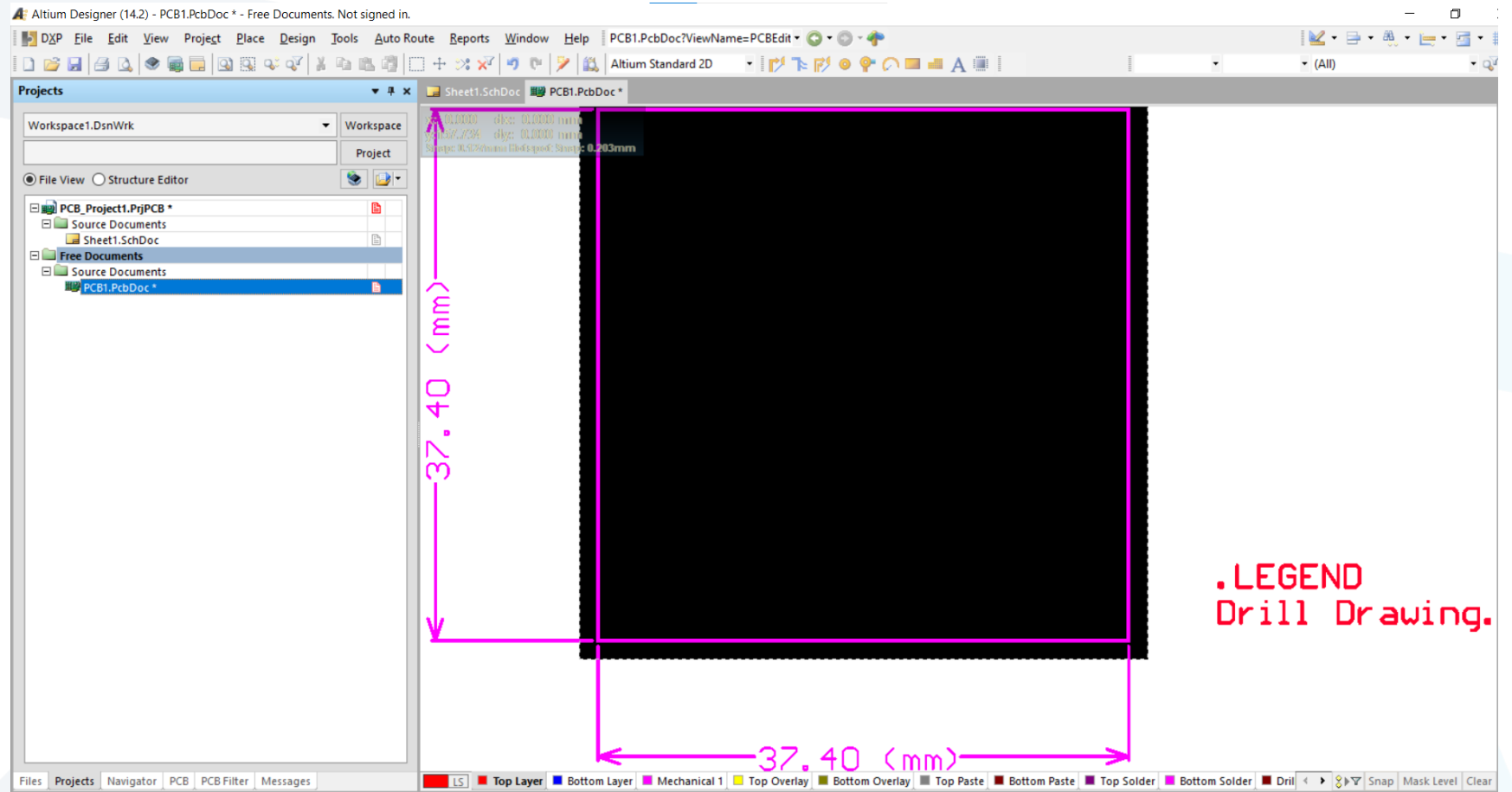
**Altium Designer
Board Wizard is complete**

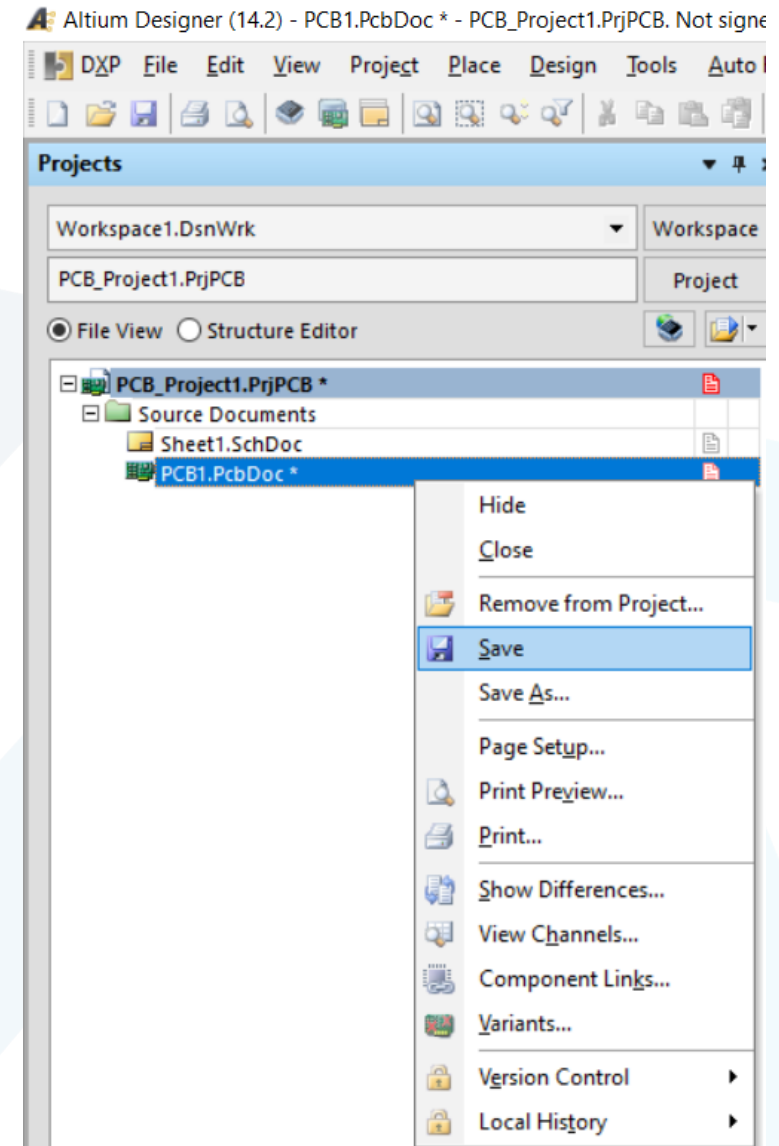
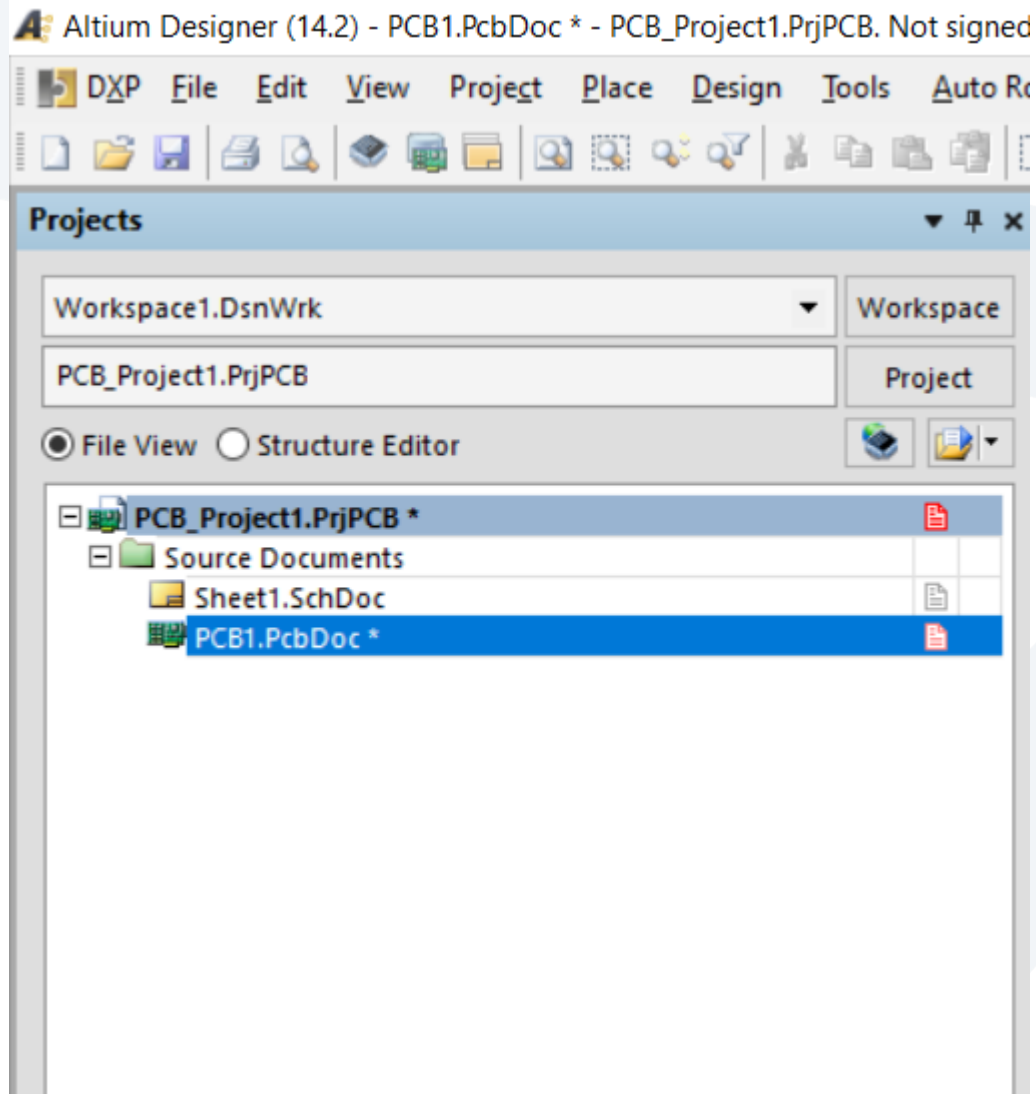
You have successfully completed the Board Wizard.
To close this wizard and create the board click Finish.

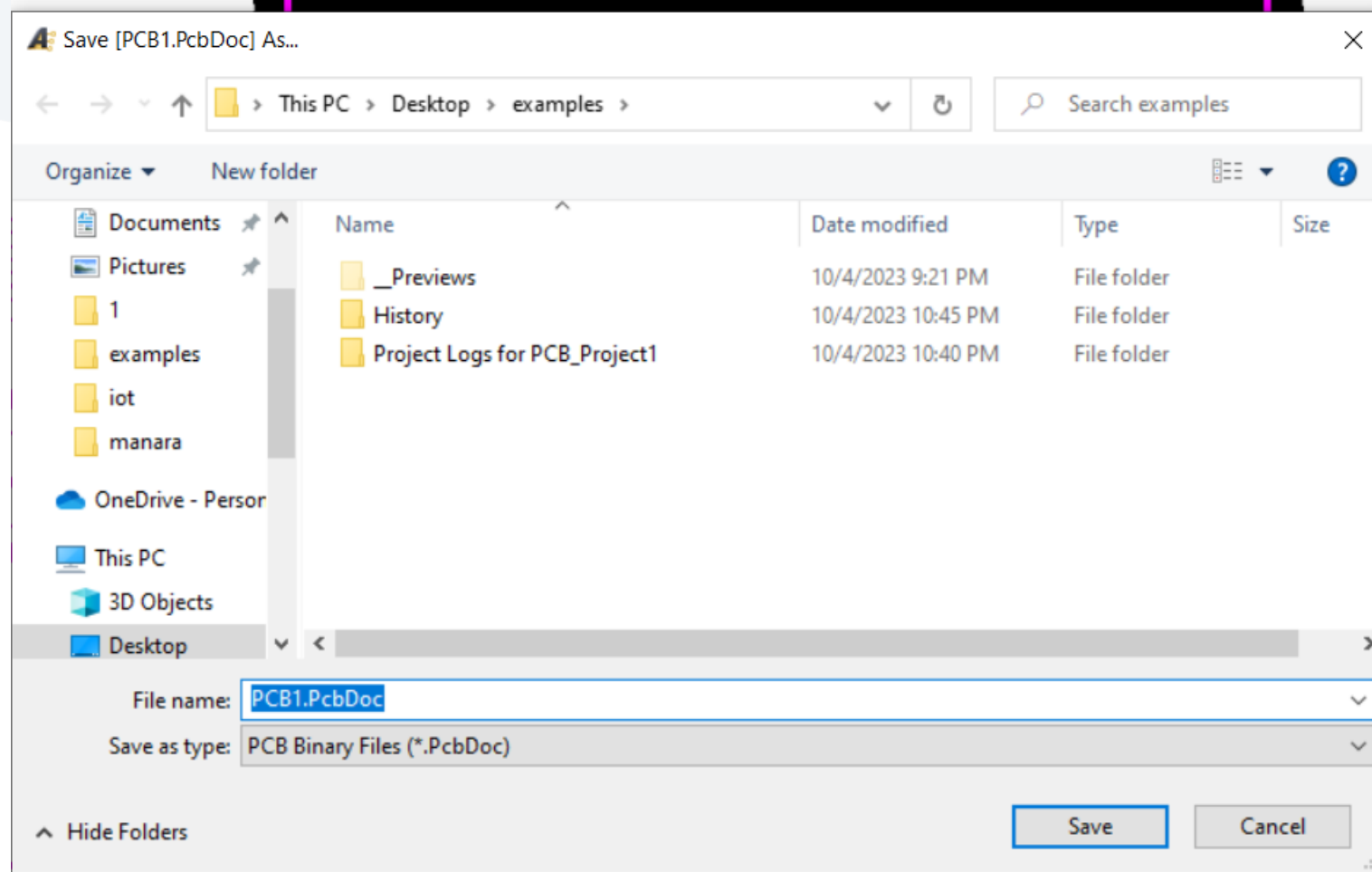


Cancel < Back Next > Finish

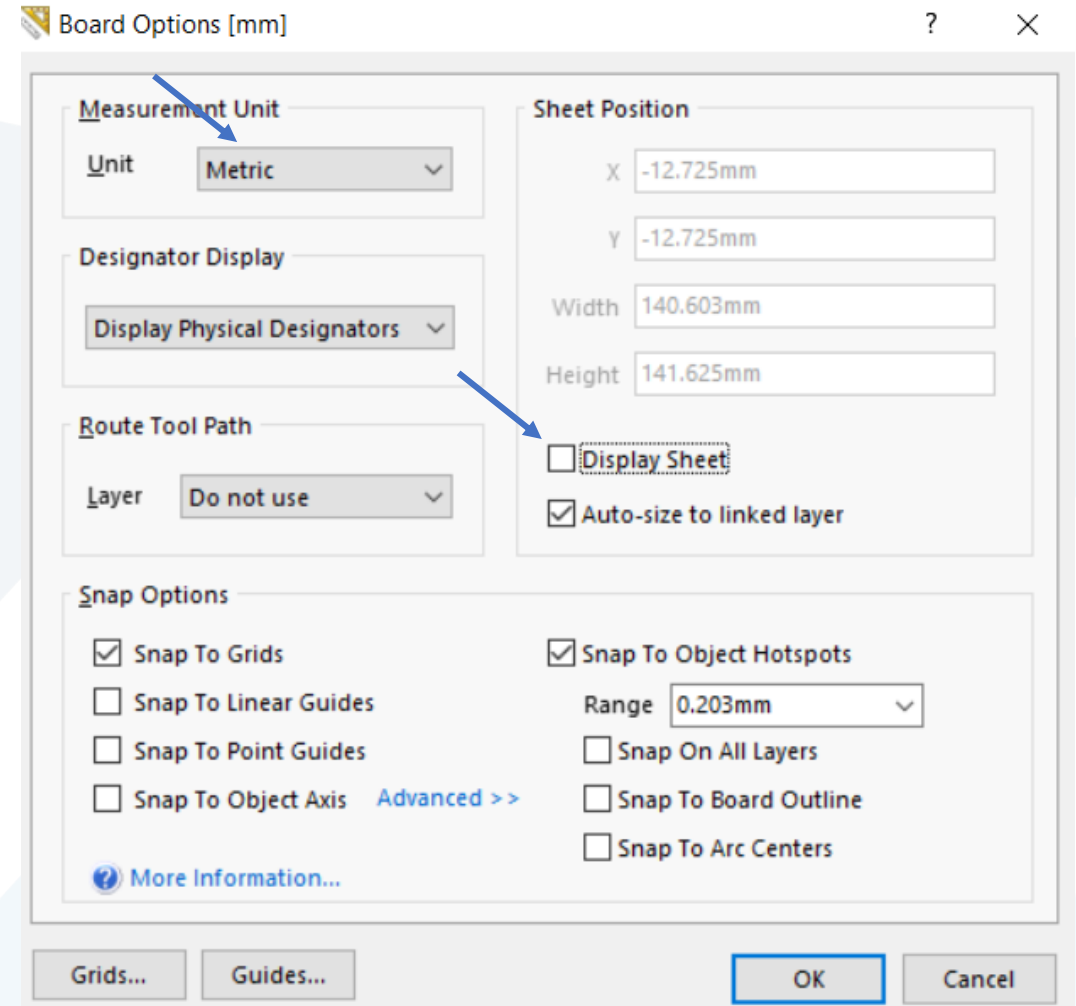
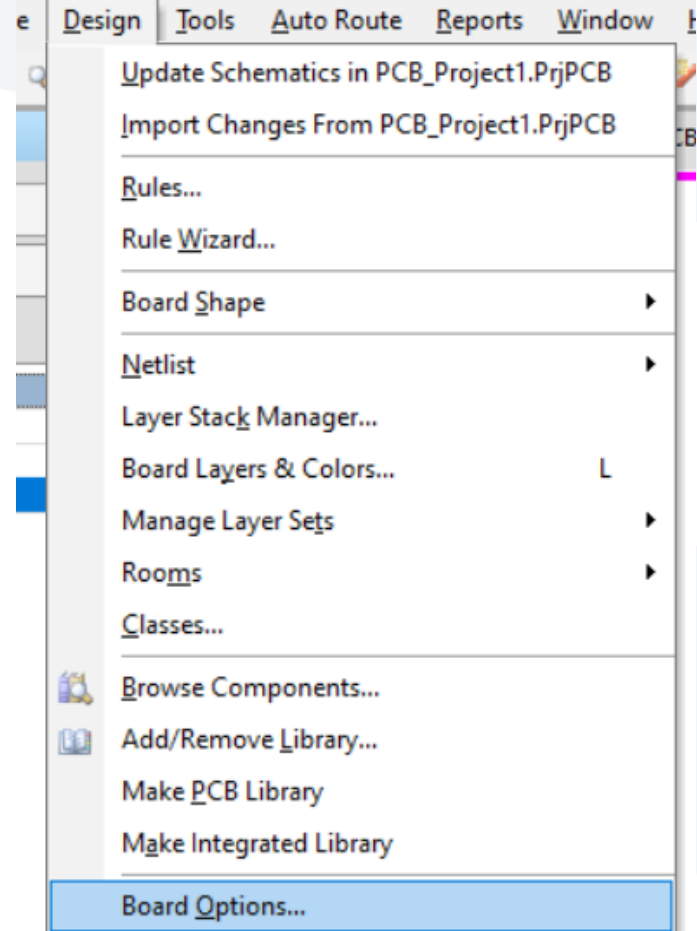
Click the file and hold then
move it up to our project
folder

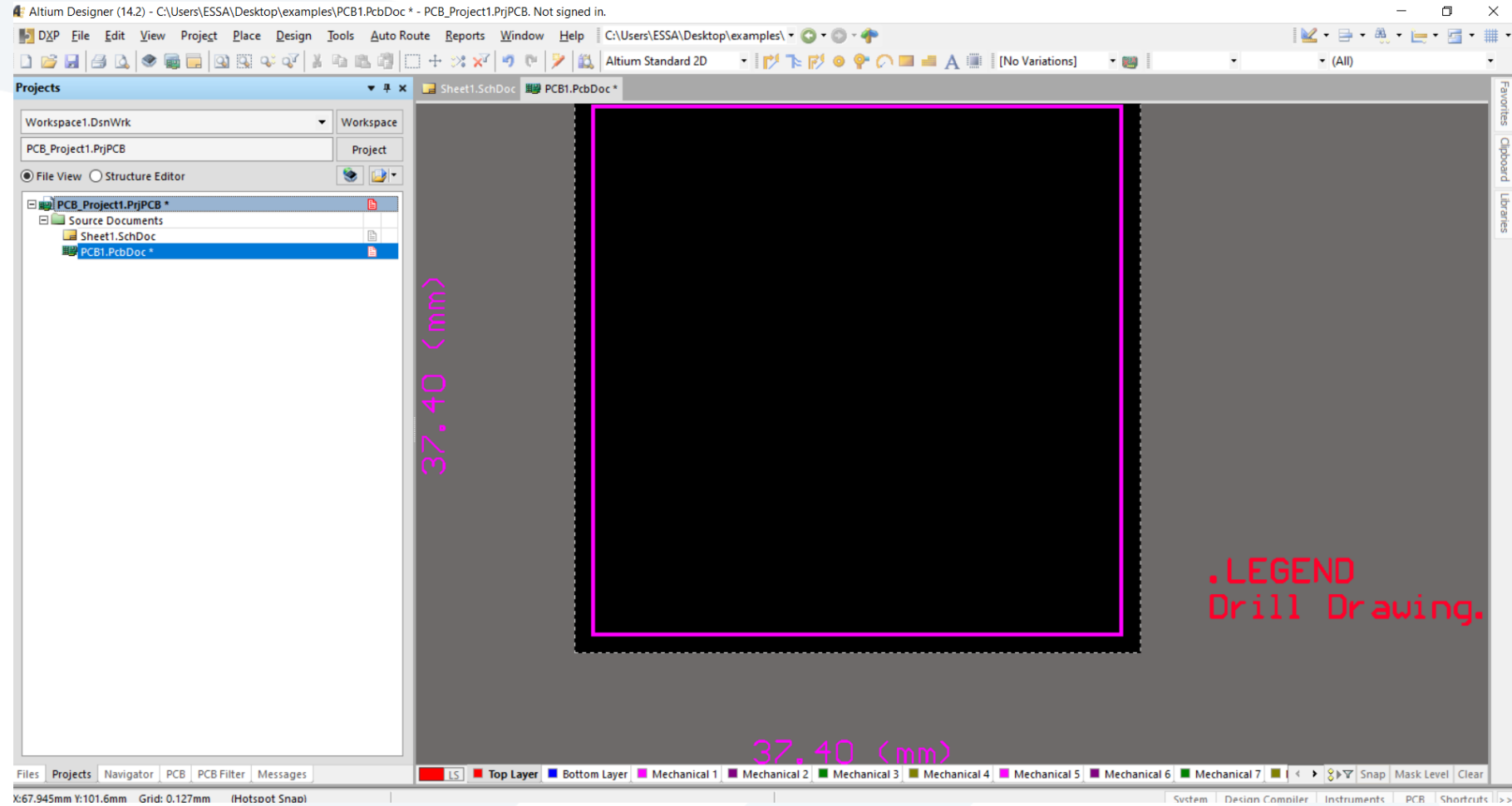






ktop\examples\PCB1.PcbDoc - PCB_Project1.PrjPCB. Not sig





ktop\examples\PCB1.PcbDoc * - PCB_Project1.PrjPCB. Not si

Design Tools Auto Route Reports Window

Update Schematics in PCB_Project1.PrjPCB

Import Changes From PCB_Project1.PrjPCB

Rules...

Rule Wizard...

Board Shape

Netlist

Layer Stack Manager...

Board Layers & Colors...

Manage Layer Sets

Rooms

Classes...

Browse Components...

Add/Remove Library...

Make PCB Library

Make Integrated Library









































Board Options...

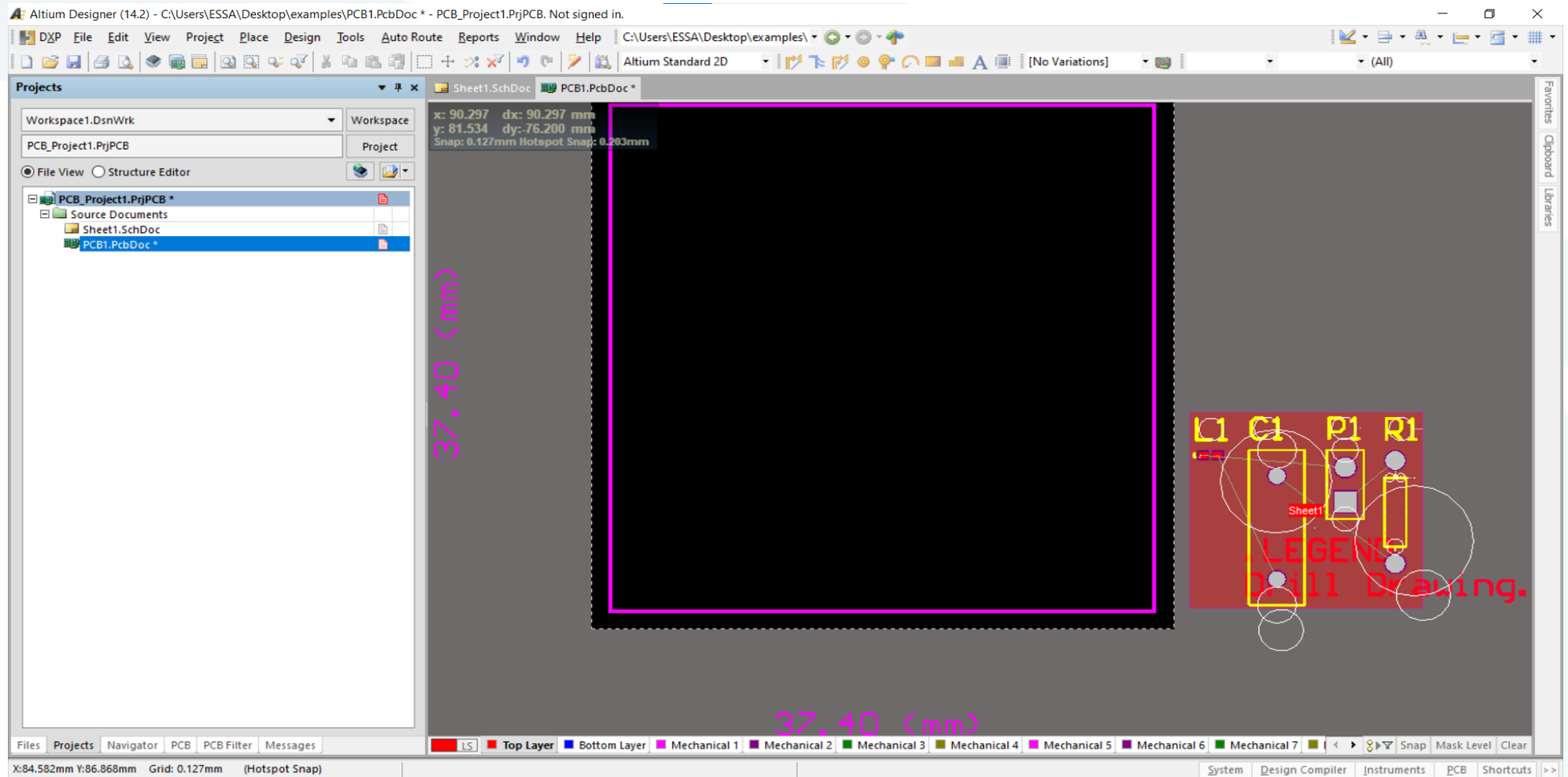
Engineering Change Order

Modifications					Status		
Enable	Action	Affected Object		Affected Document	Check	Done	Message
Add Components(4)							
<input checked="" type="checkbox"/>	Add	C1	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	L1	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	P1	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	R1	To	PCB1.PcbDoc			
Add Nets(4)							
<input checked="" type="checkbox"/>	Add	NetC1_1	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	NetC1_2	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	NetL1_1	To	PCB1.PcbDoc			
<input checked="" type="checkbox"/>	Add	NetP1_1	To	PCB1.PcbDoc			
Add Component Classes(1)							
<input checked="" type="checkbox"/>	Add	Sheet1	To	PCB1.PcbDoc			
Add Rooms(1)							
<input checked="" type="checkbox"/>	Add	Room Sheet1 (Scope=InComponentCI To		PCB1.PcbDoc			

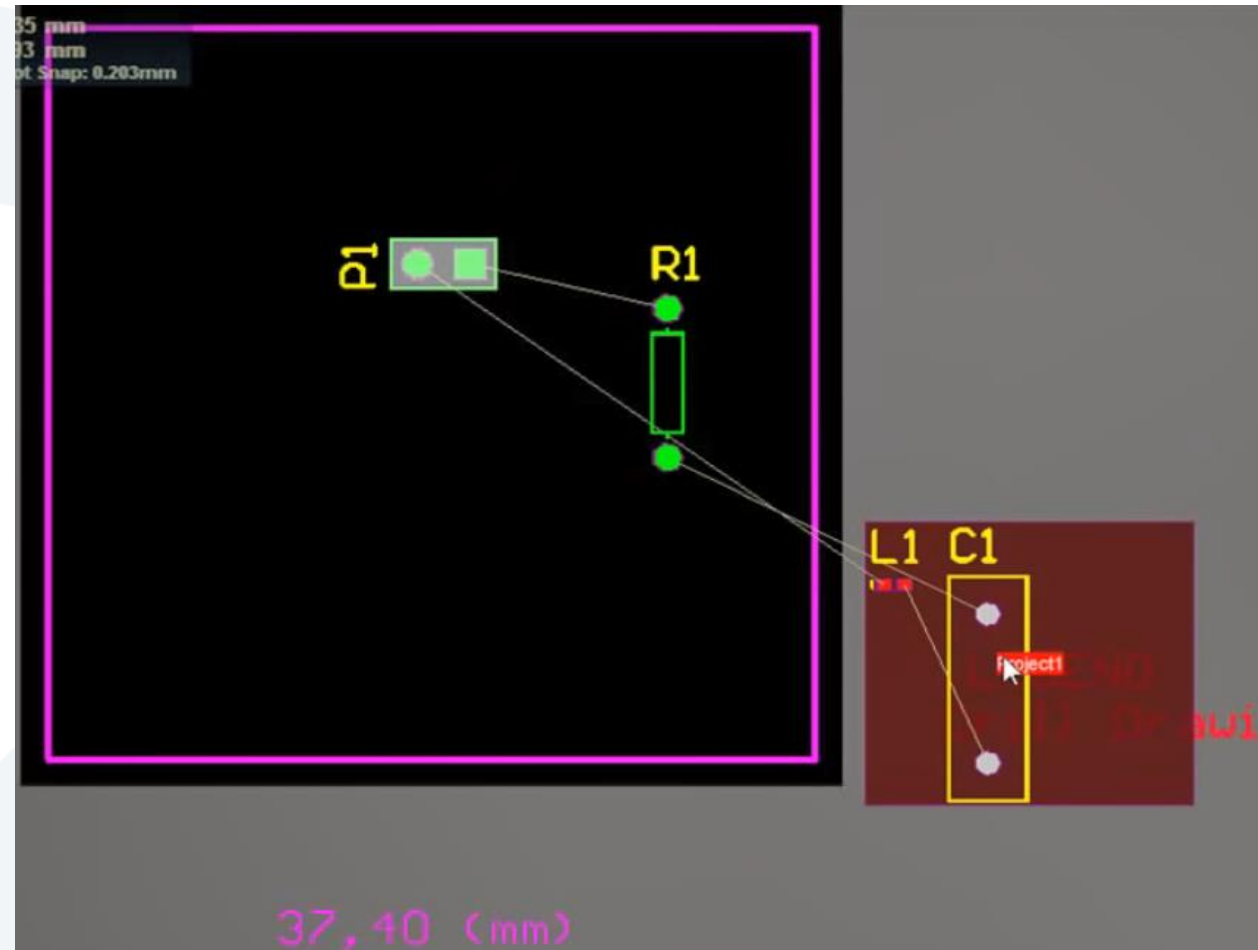
Validate Changes Execute Changes Report Changes... ☐ Only Show Errors

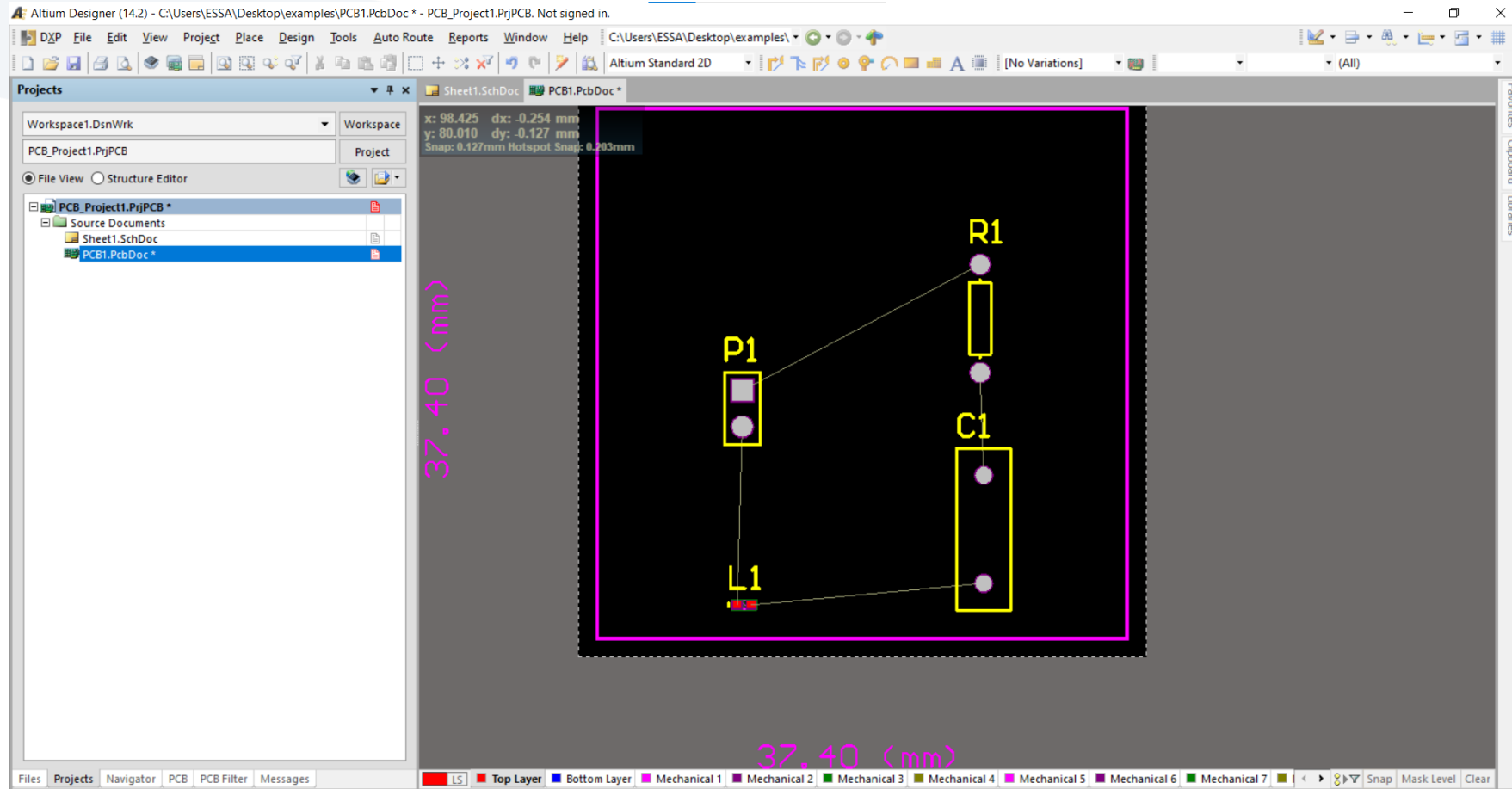
Close

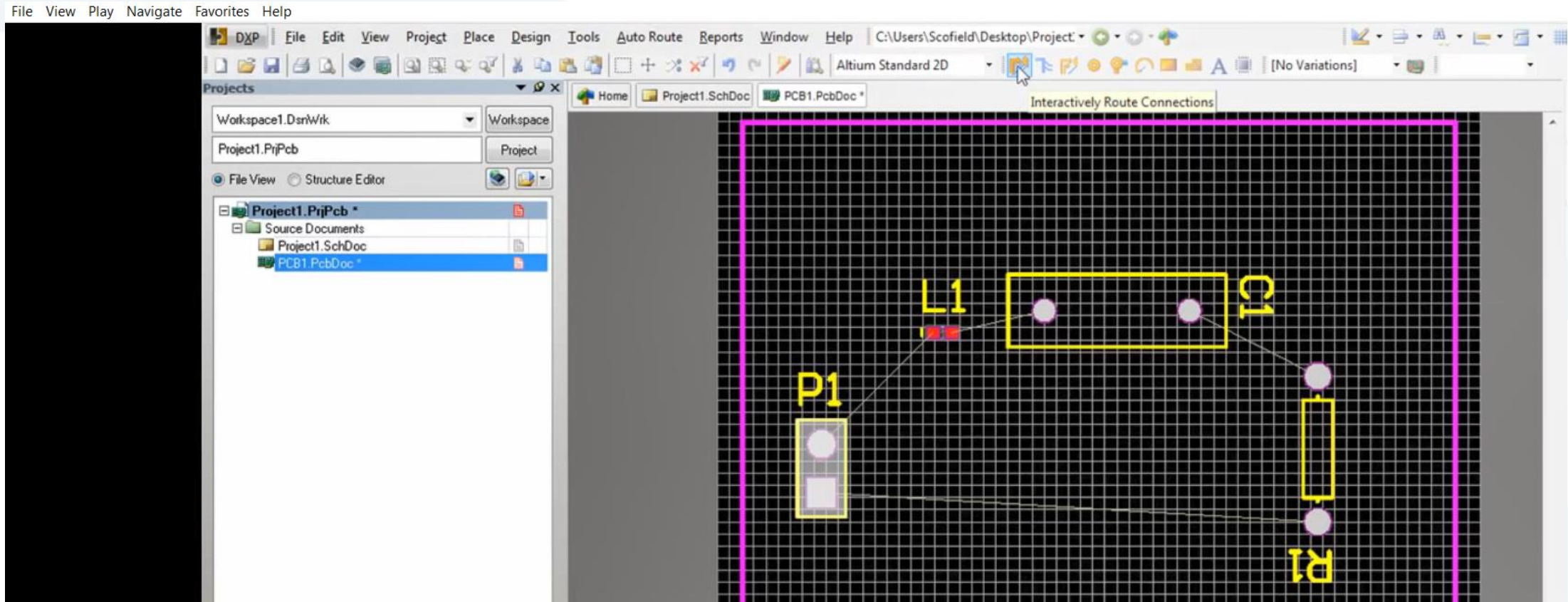
Engineering Change Order								
Modifications						Status		
	Enable	Action	Affected Object		Affected Document	Check	Done	Message
[-]		Add Components(4)						
	<input checked="" type="checkbox"/>	Add	 C1	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 L1	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 P1	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 R1	To	 PCB1.PcbDoc			
[-]		Add Nets(4)						
	<input checked="" type="checkbox"/>	Add	 NetC1_1	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 NetC1_2	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 NetL1_1	To	 PCB1.PcbDoc			
	<input checked="" type="checkbox"/>	Add	 NetP1_1	To	 PCB1.PcbDoc			
[-]		Add Component Classes(1)						
	<input checked="" type="checkbox"/>	Add	 Project1	To	 PCB1.PcbDoc			
[-]		Add Rooms(1)						
	<input checked="" type="checkbox"/>	Add	 Room Project1 (Scope=InComponentCla:	To	 PCB1.PcbDoc			
<div>Validate Changes Execute Changes Report Changes... <input type="checkbox"/> Only Show Errors</div> <div>Close</div>								

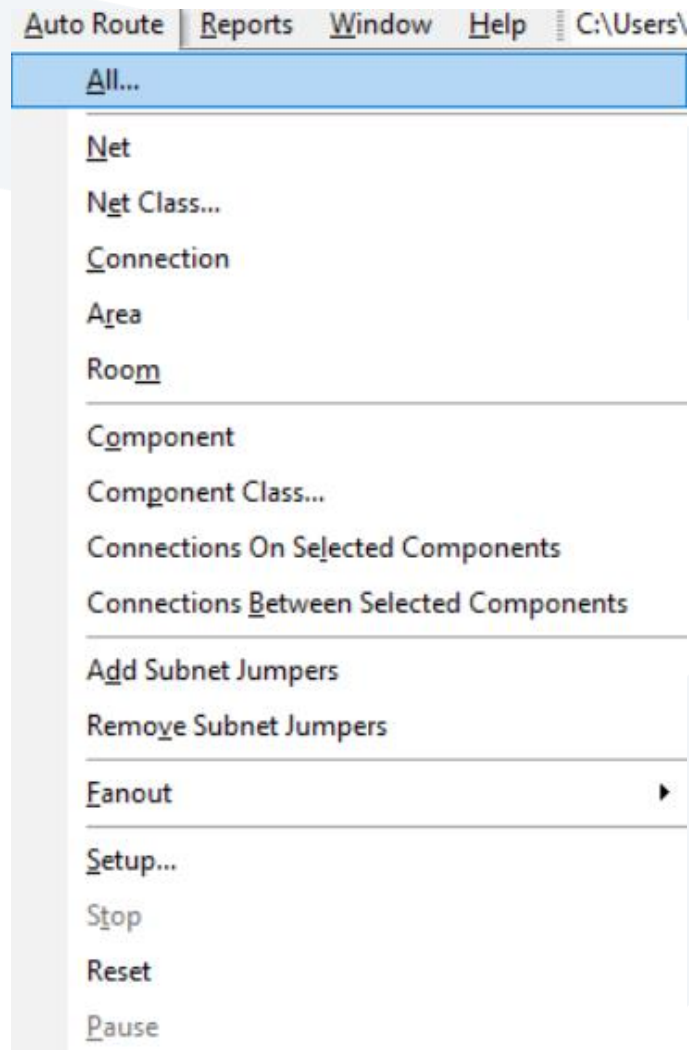


- You can rotate by space
- Delete the red square and the words

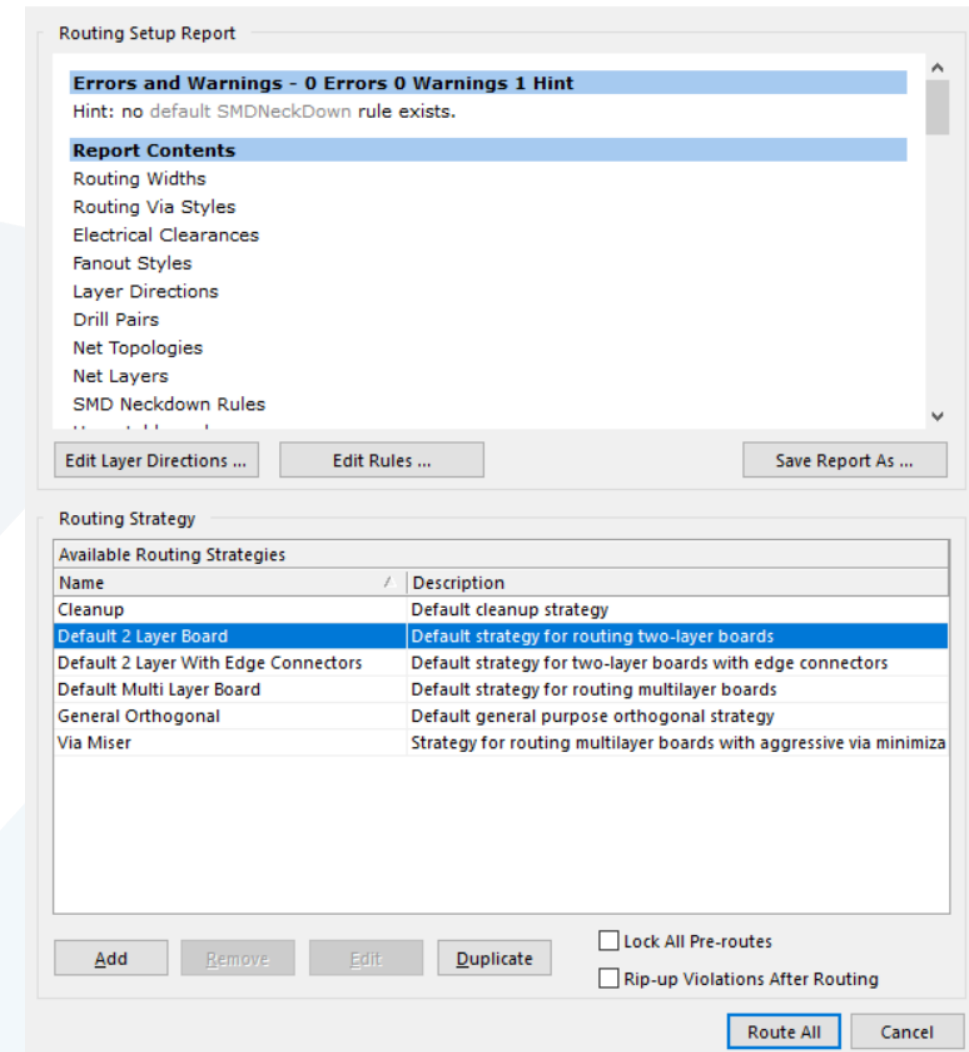


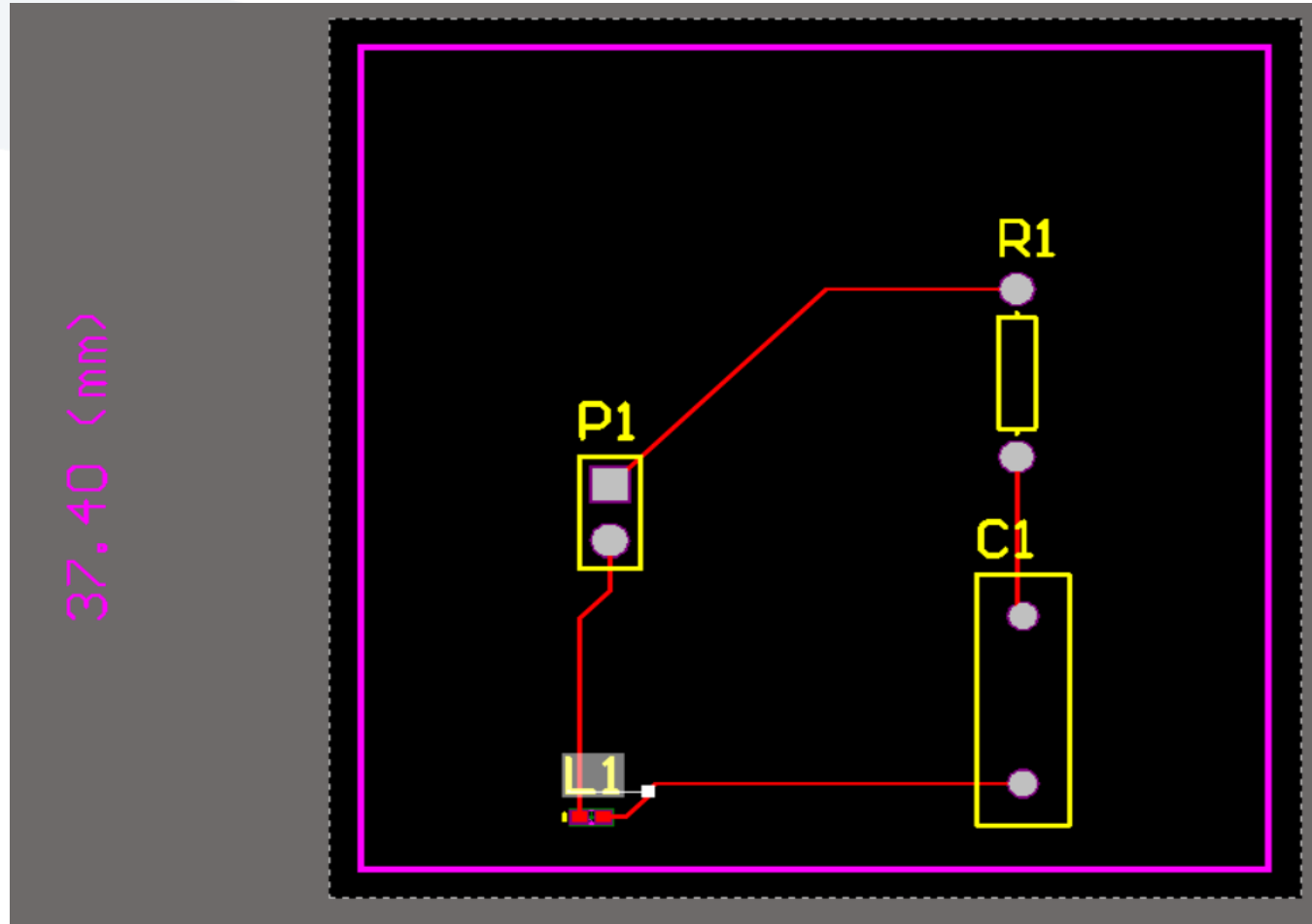


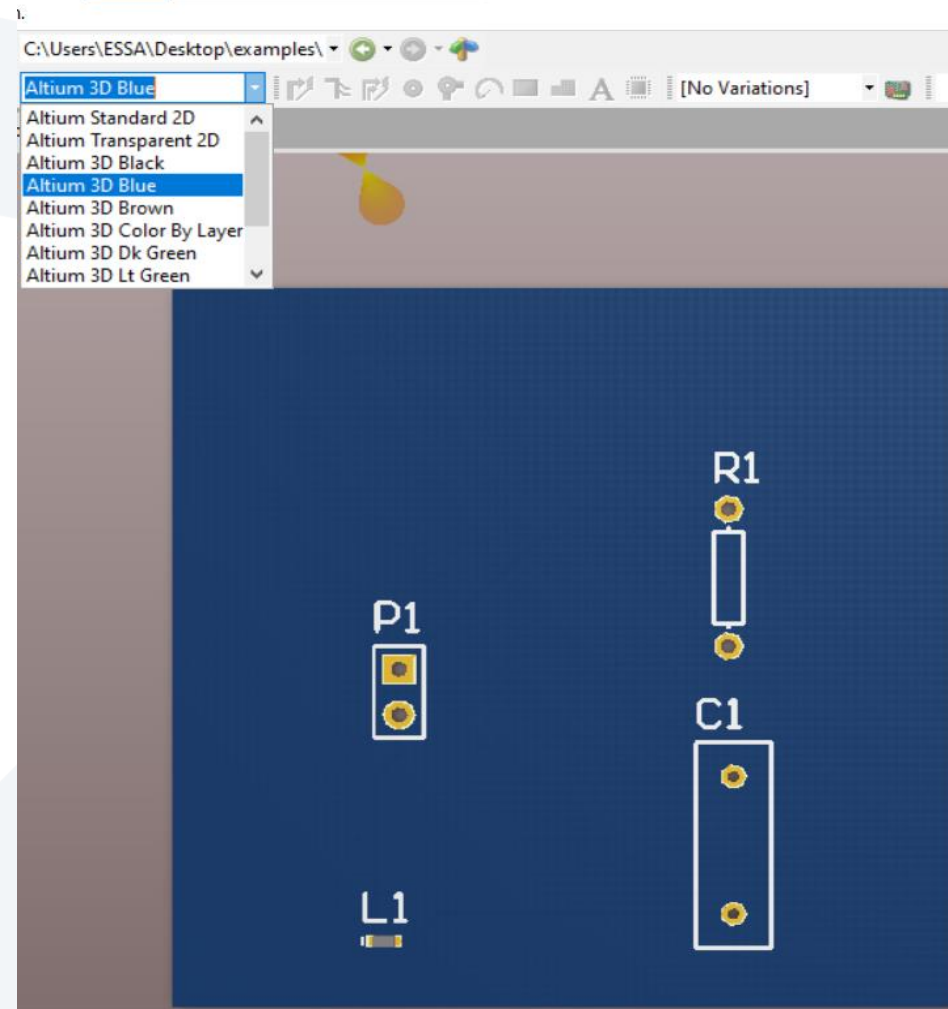




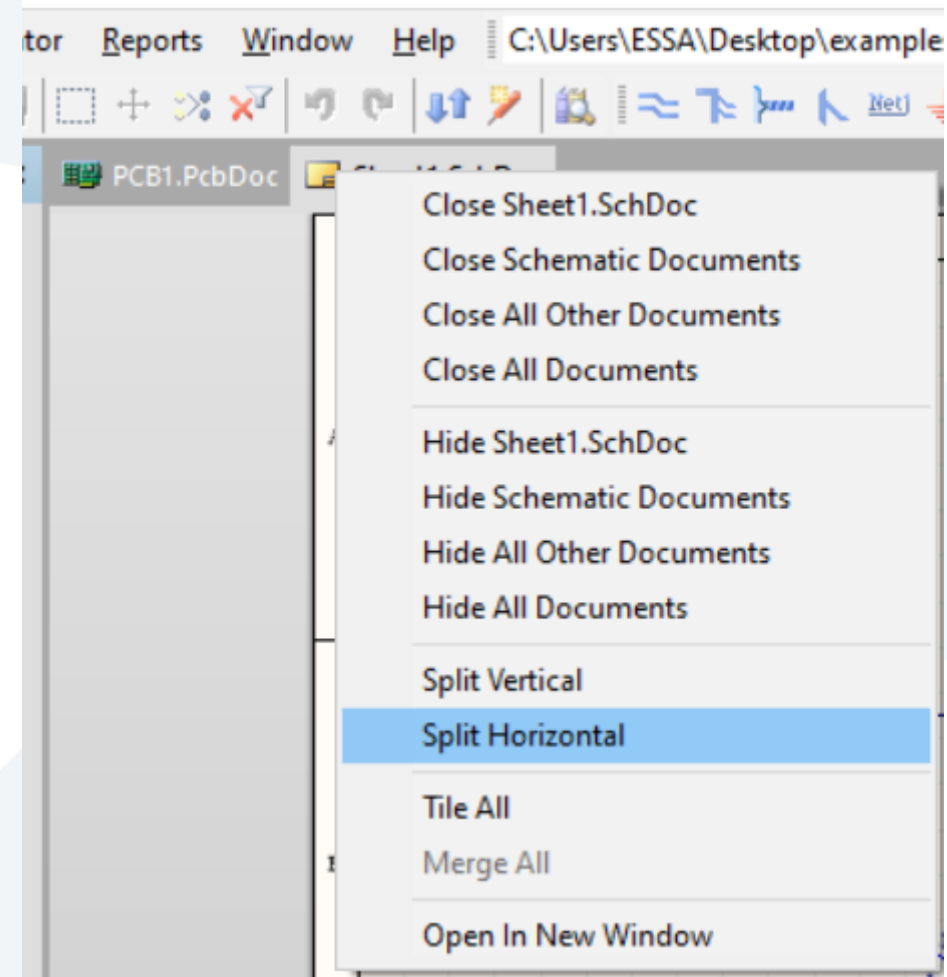
Situs Routing Strategies

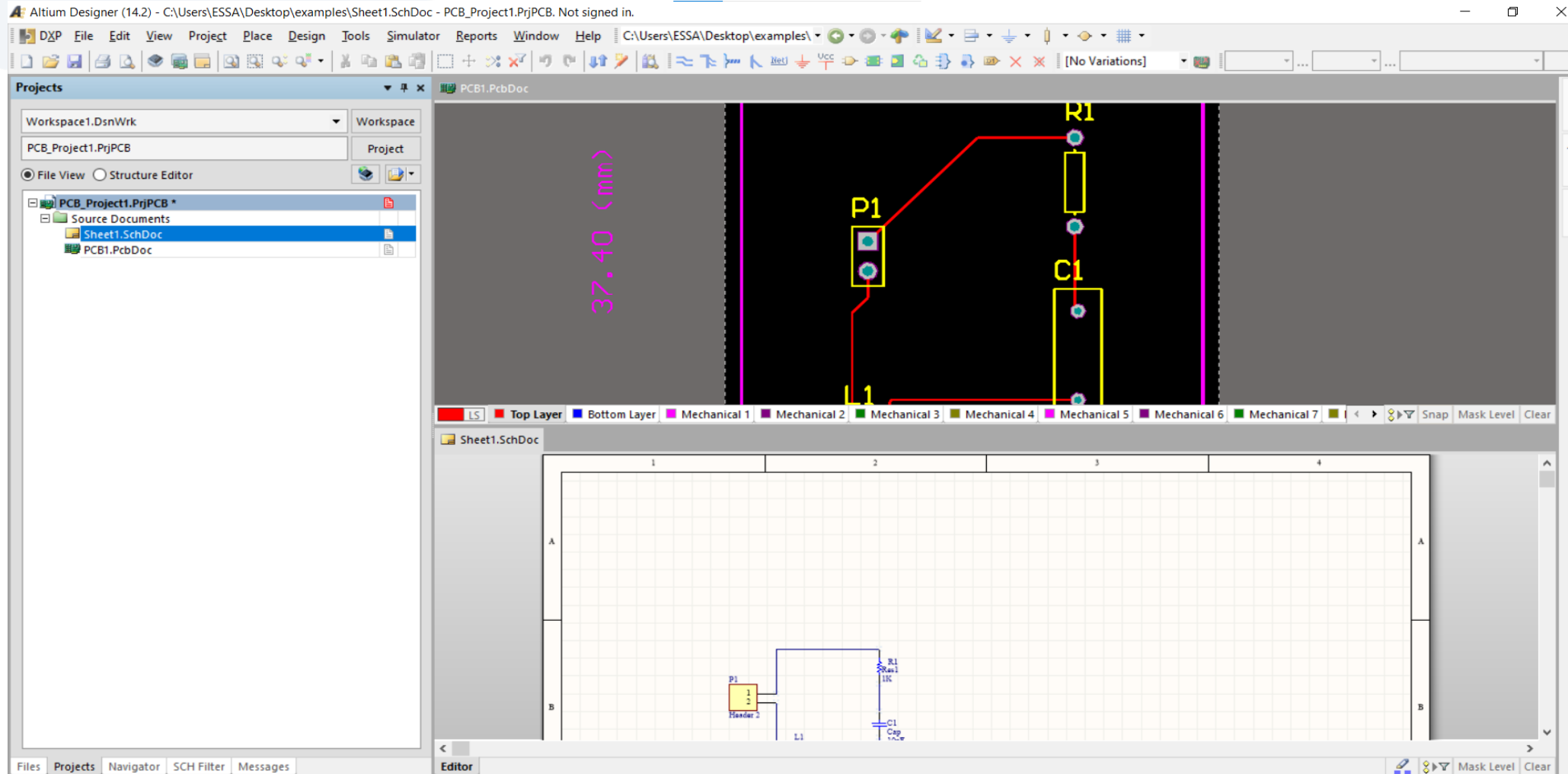


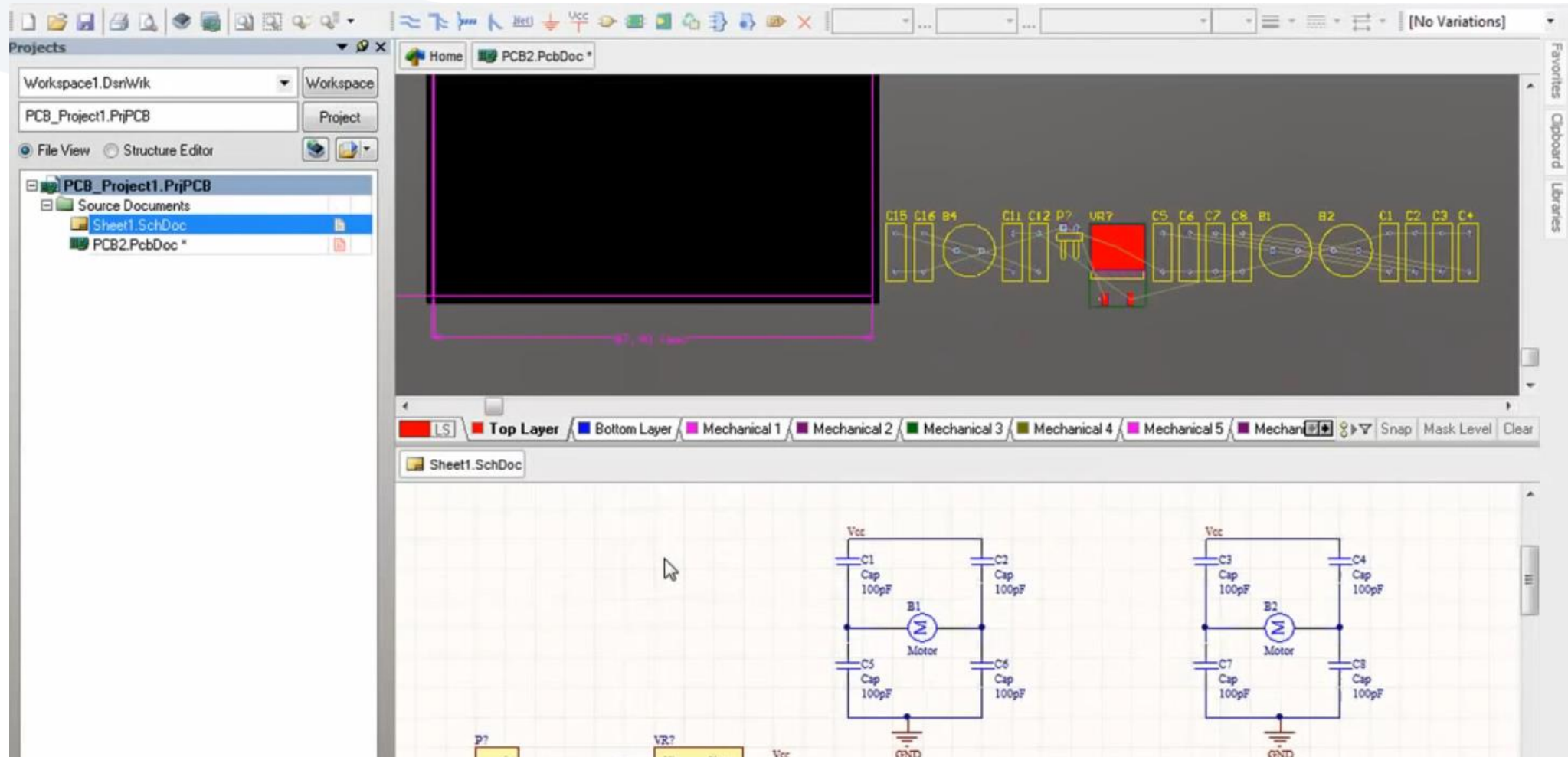


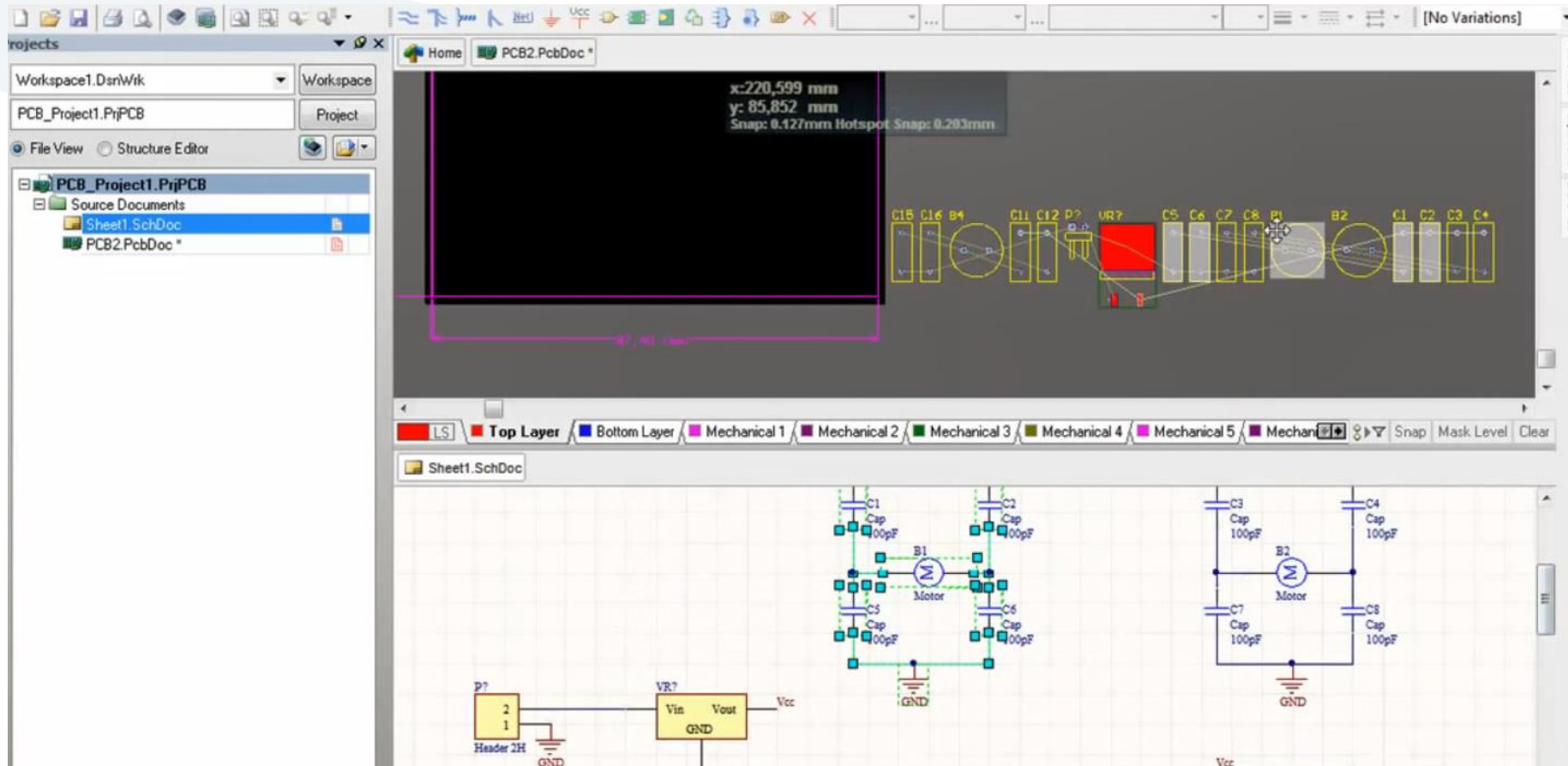


doc - PCB_Project1.PrjPCB. Not signed in.

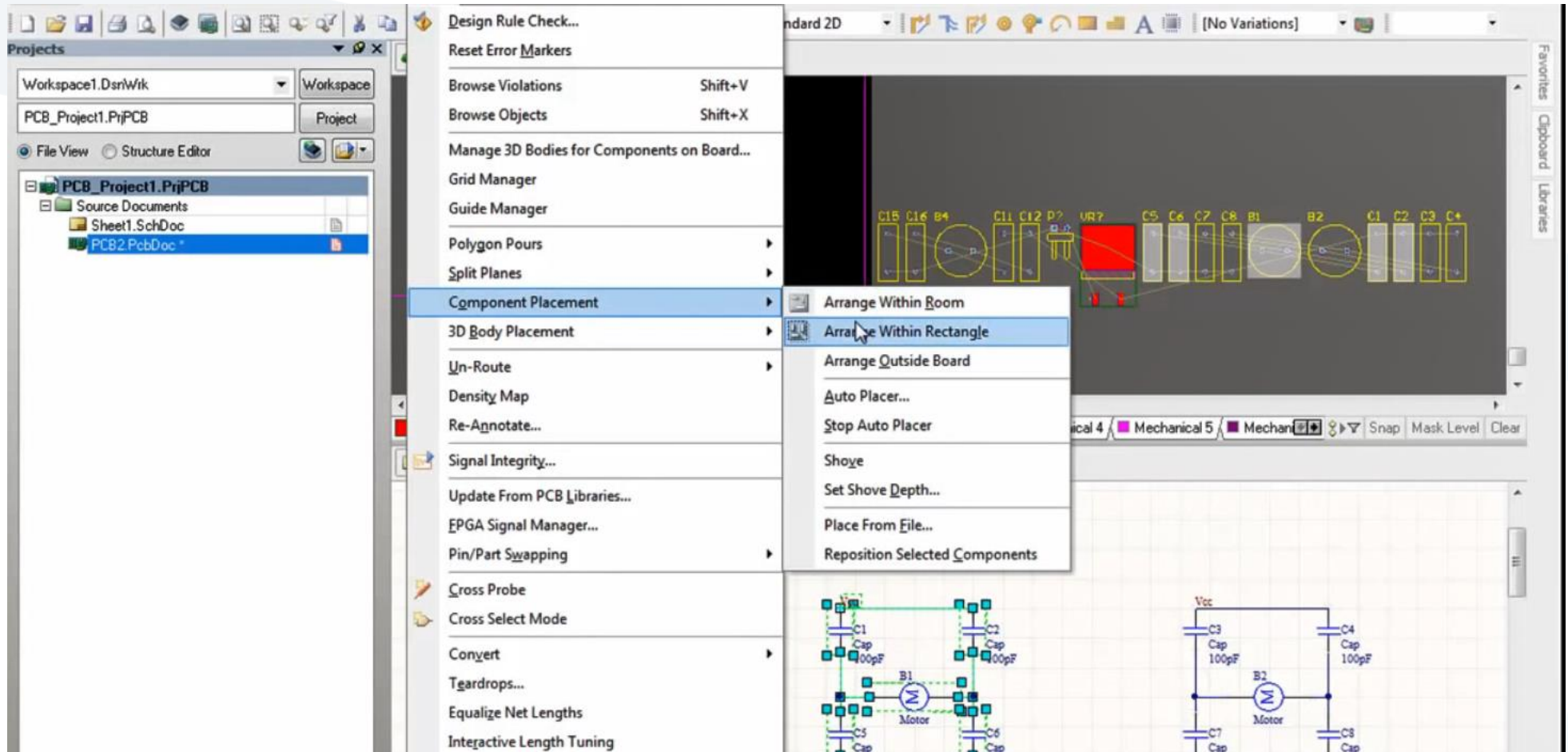


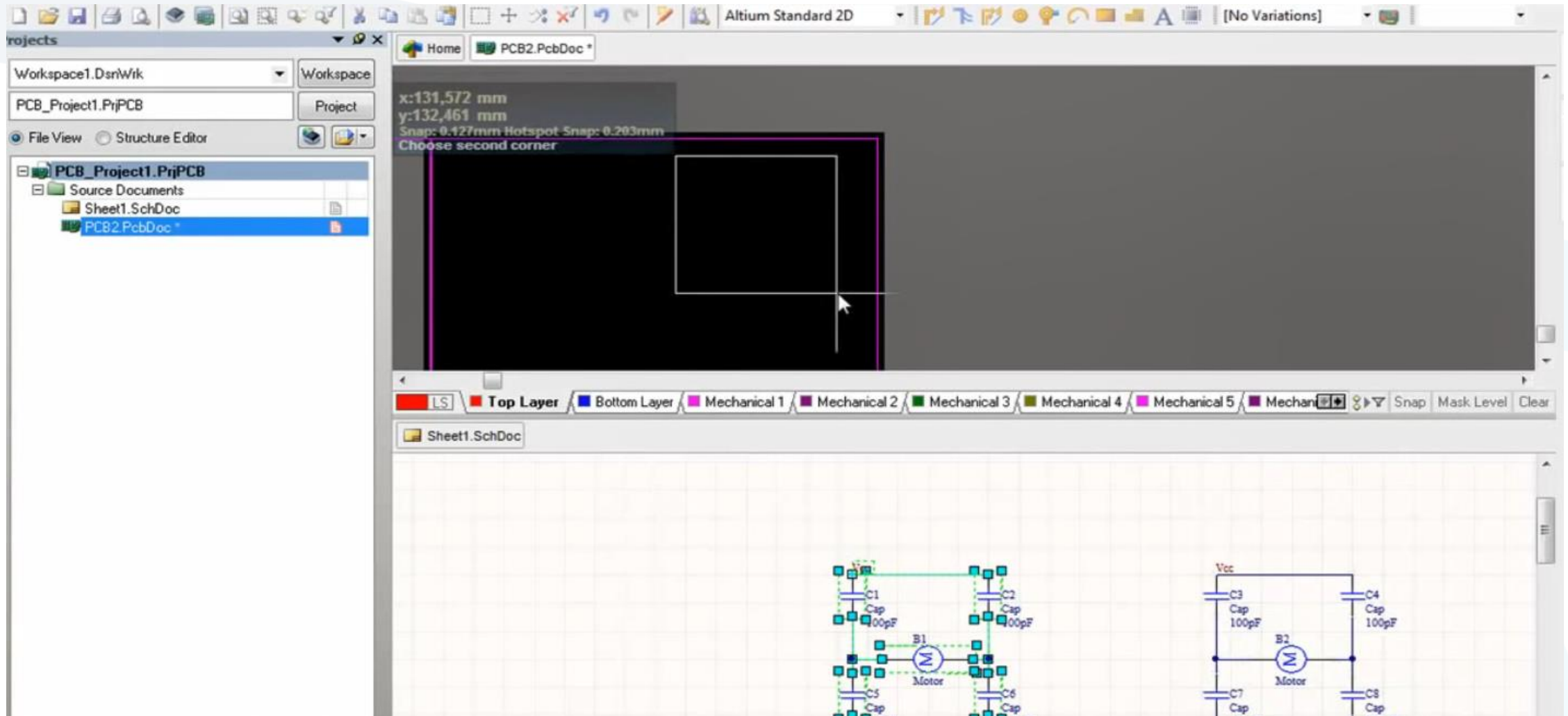


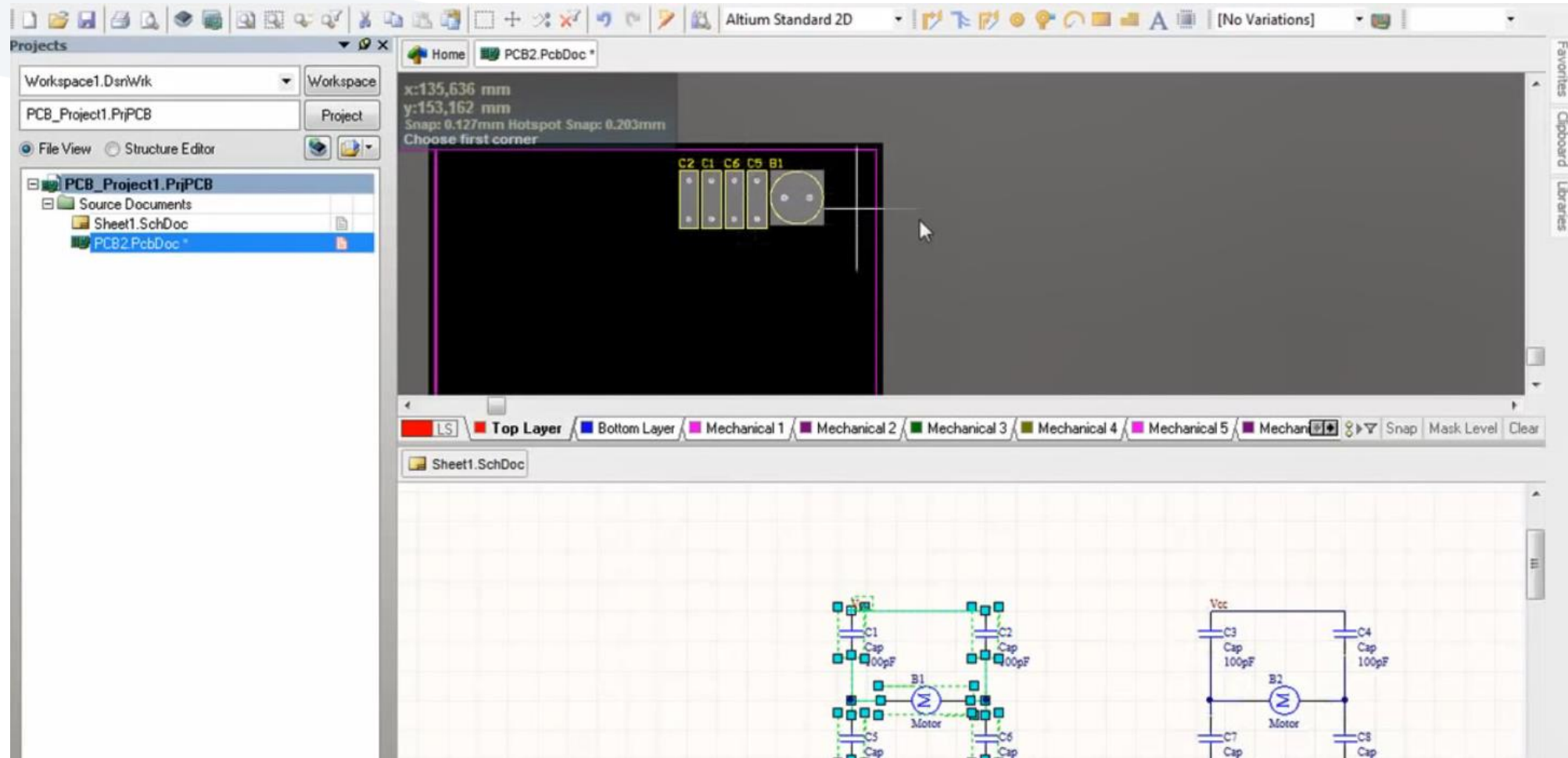


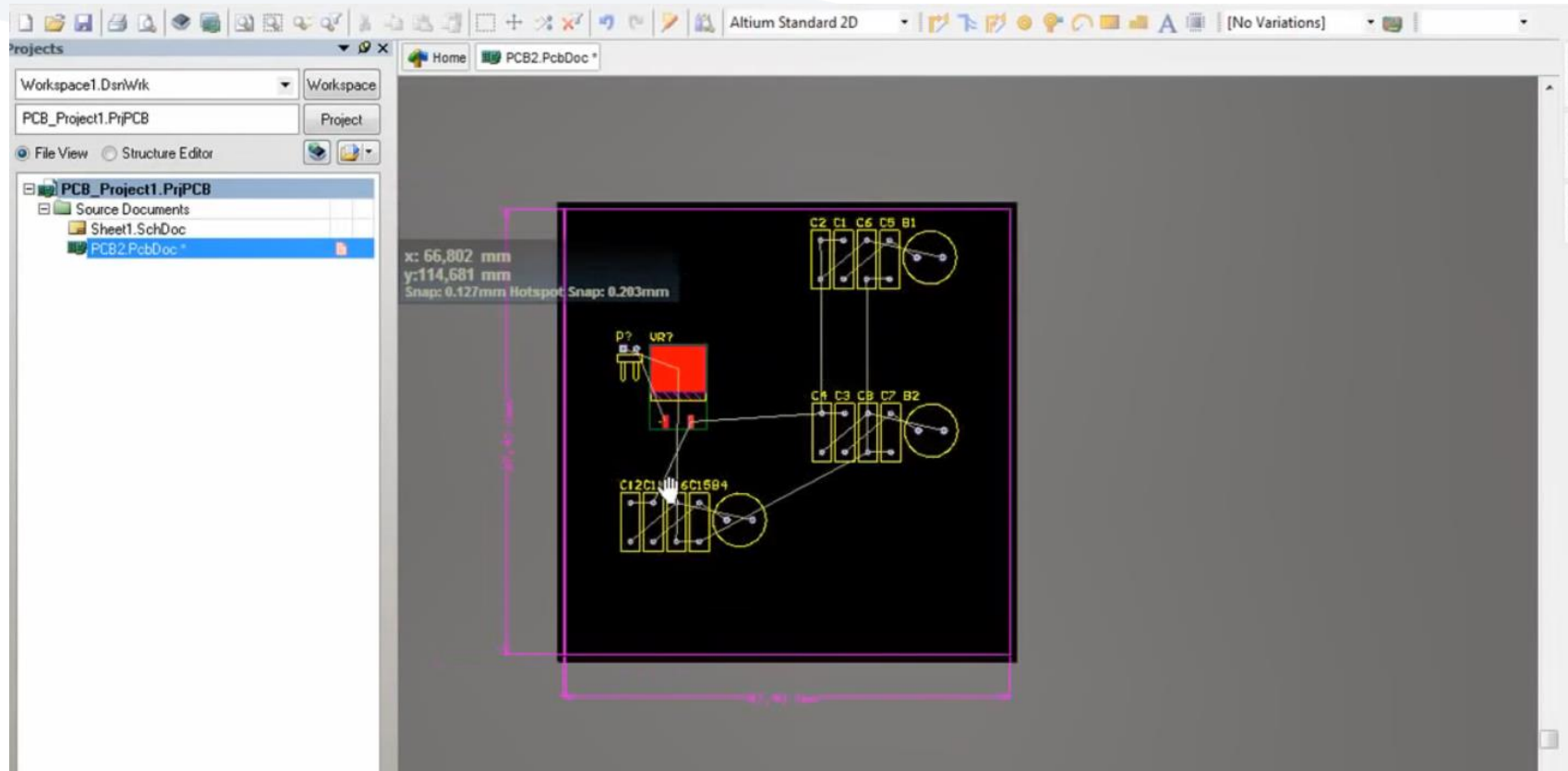


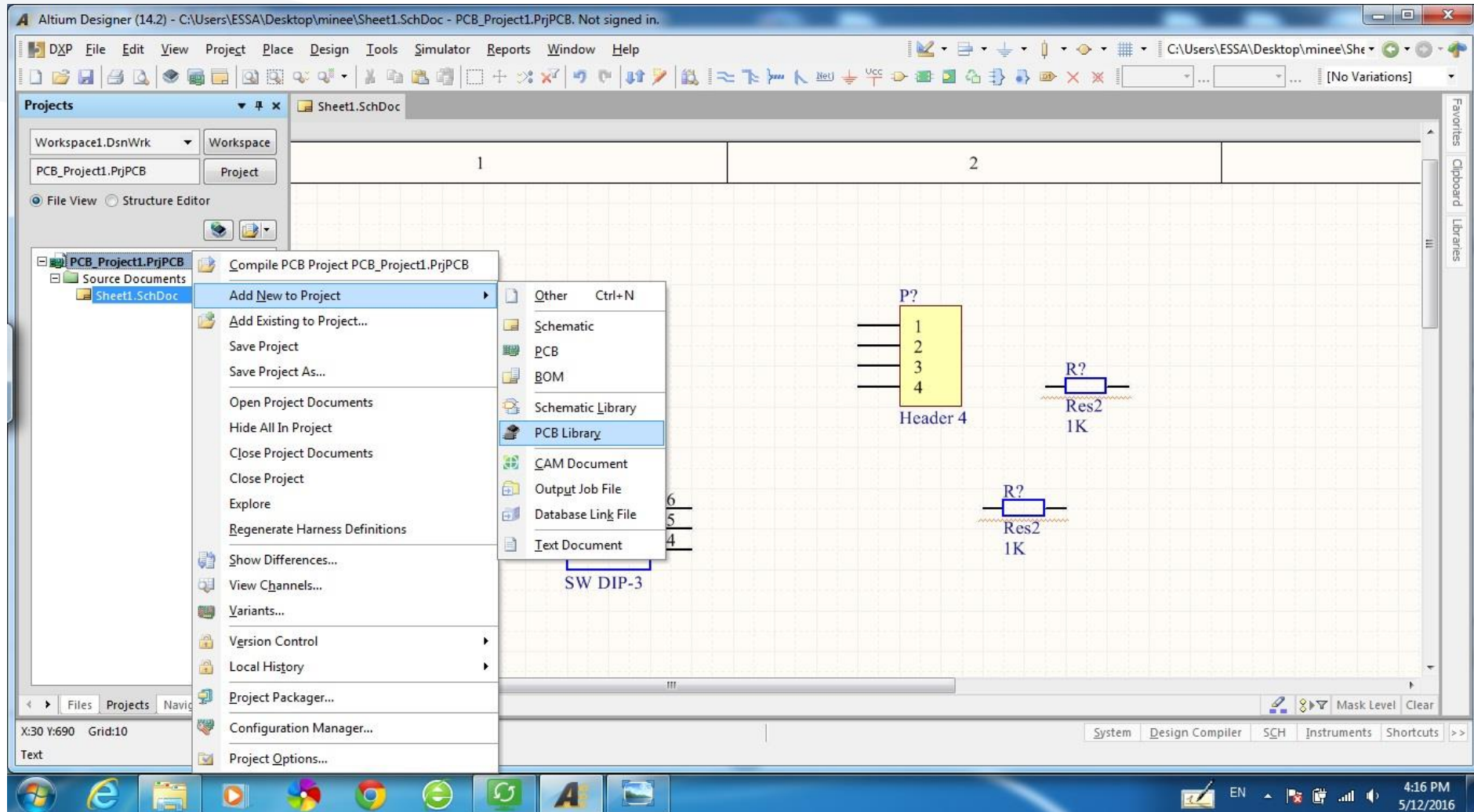
- PCB file
- Tools then

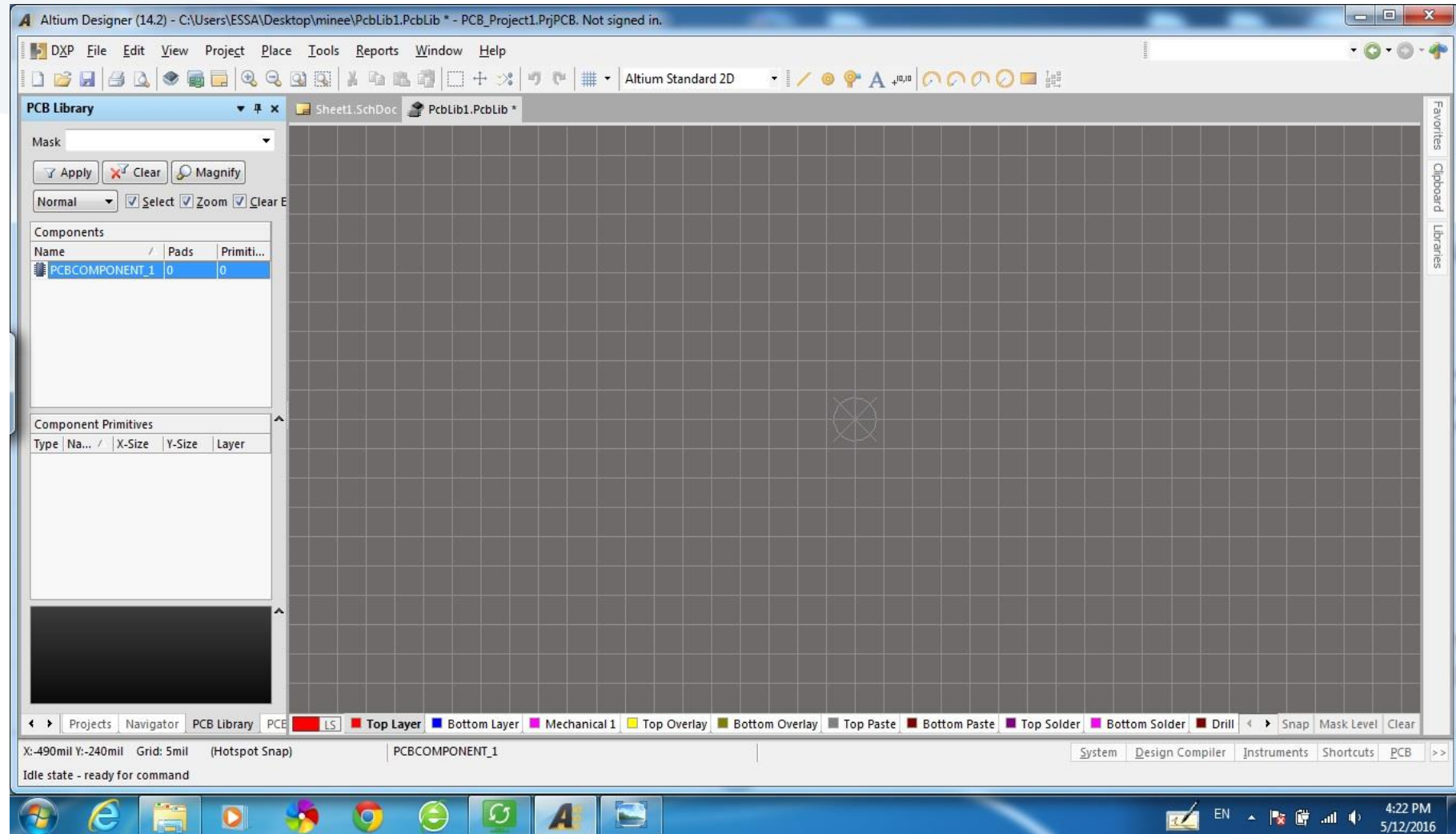


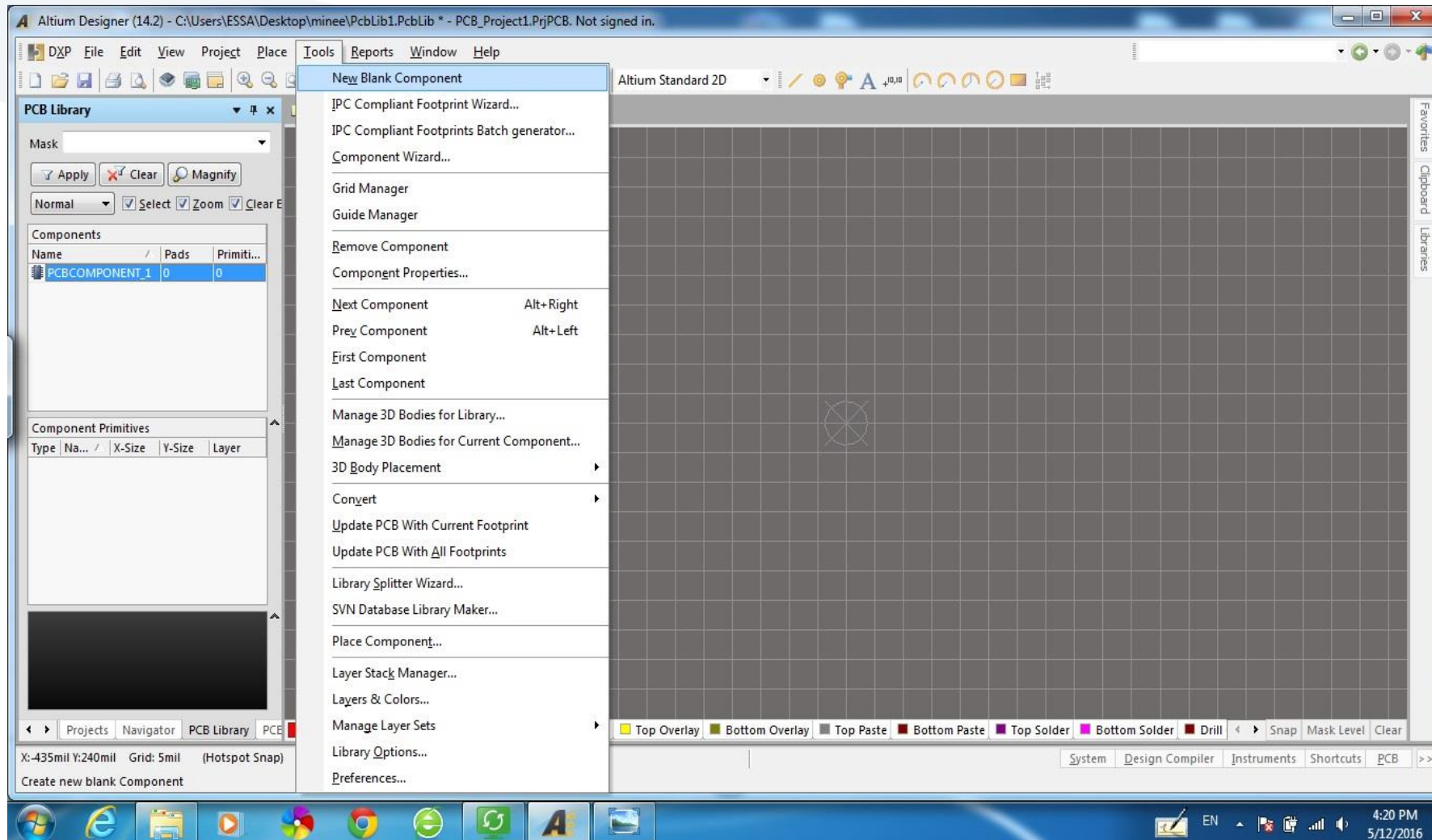


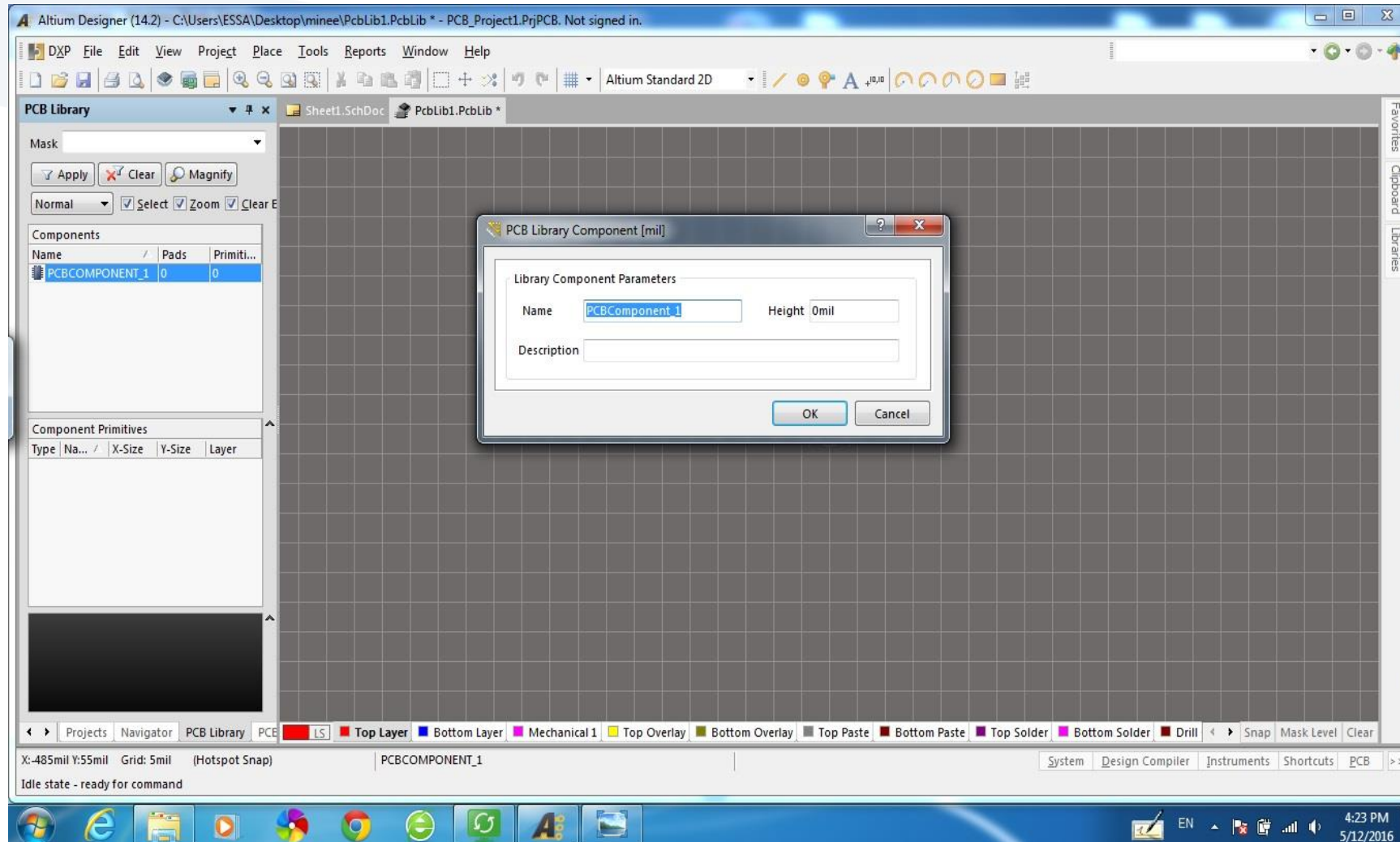


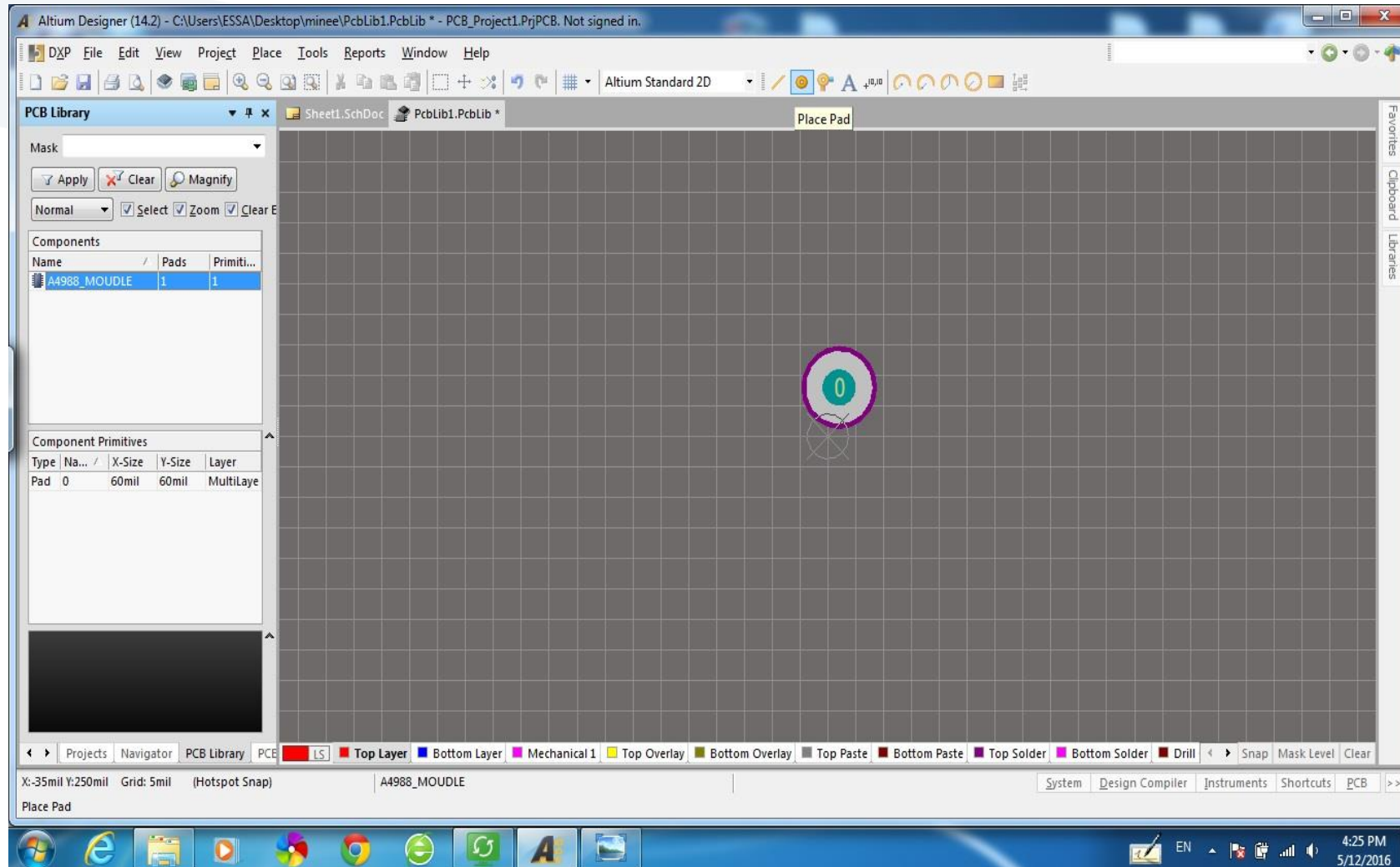


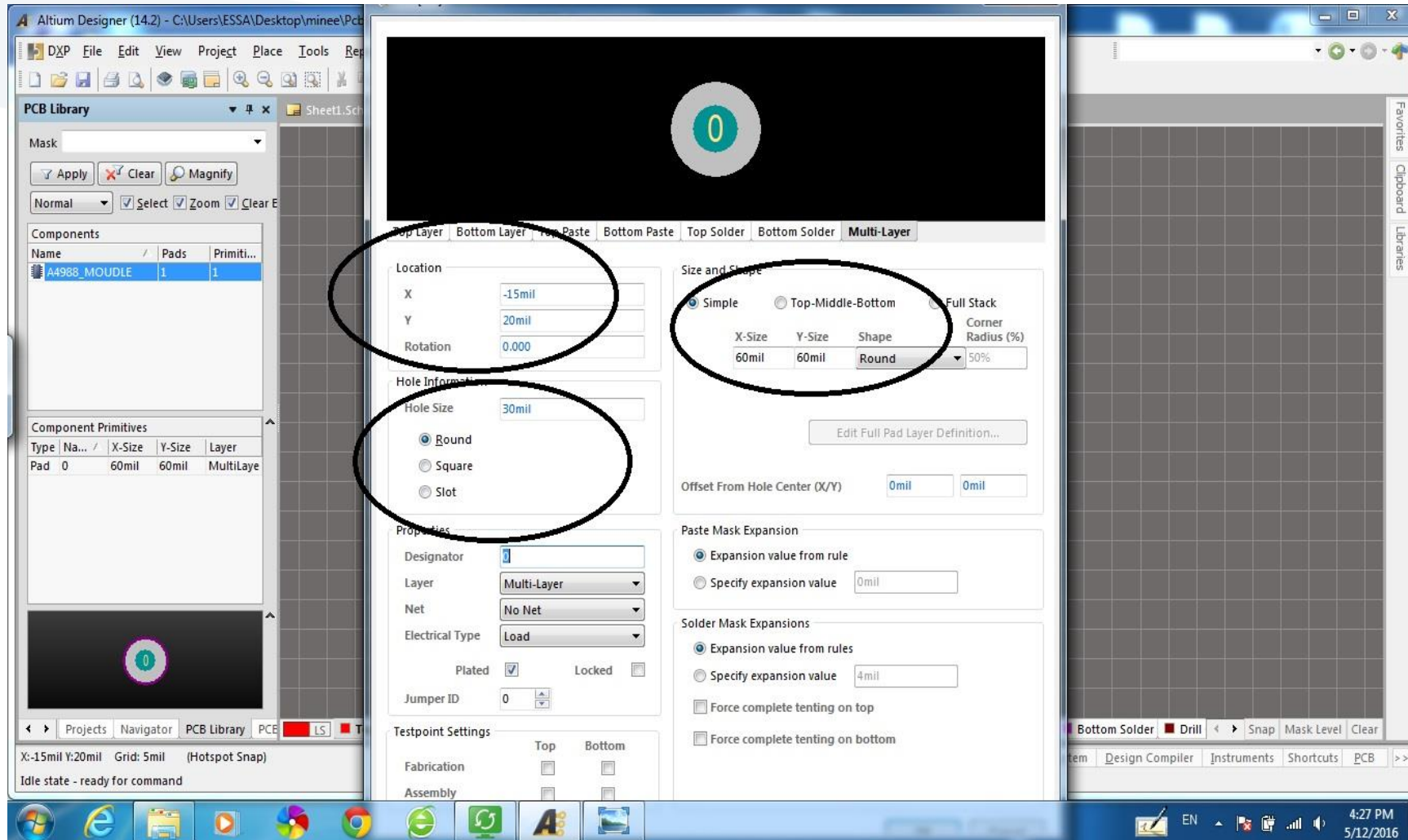








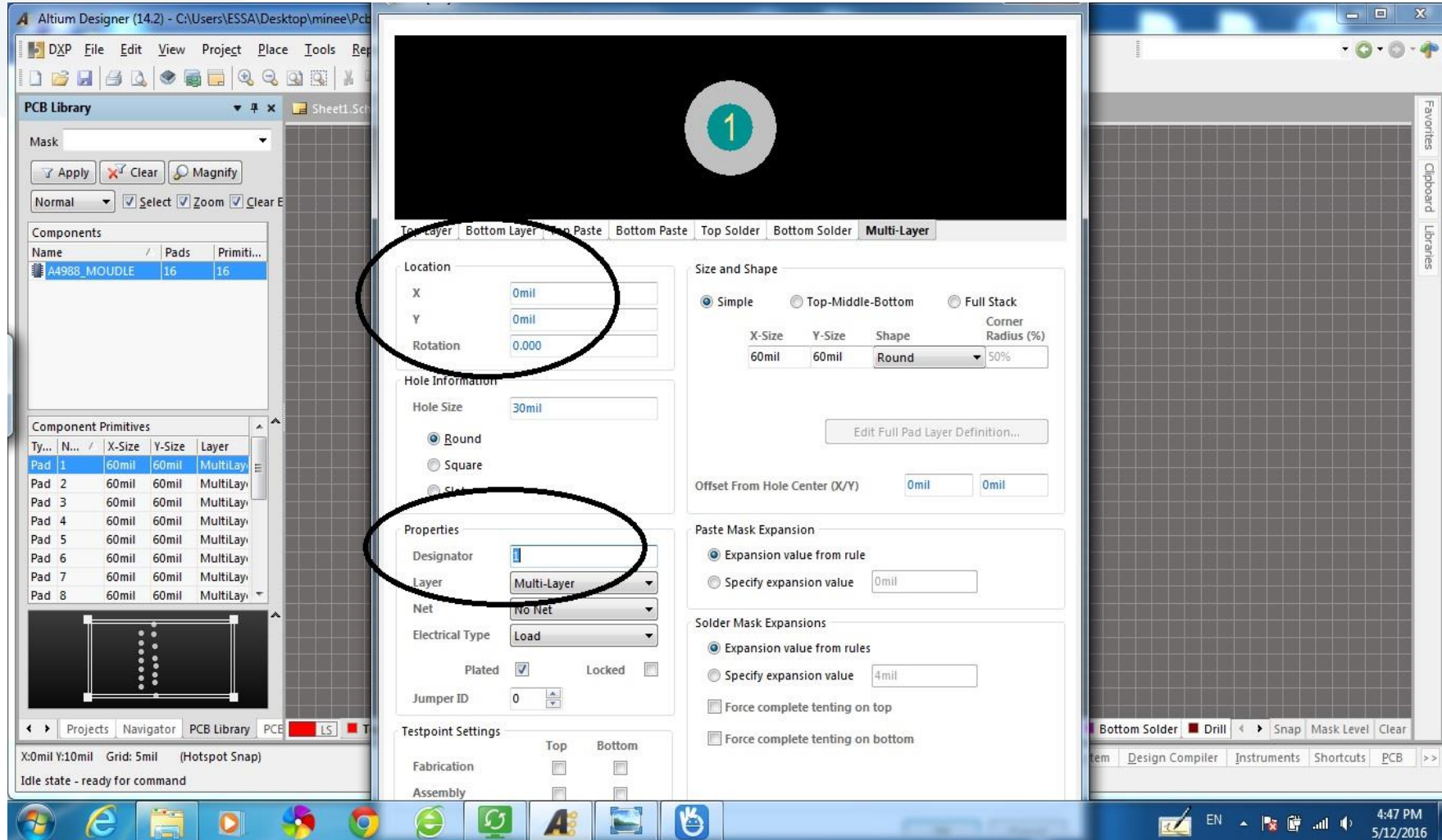


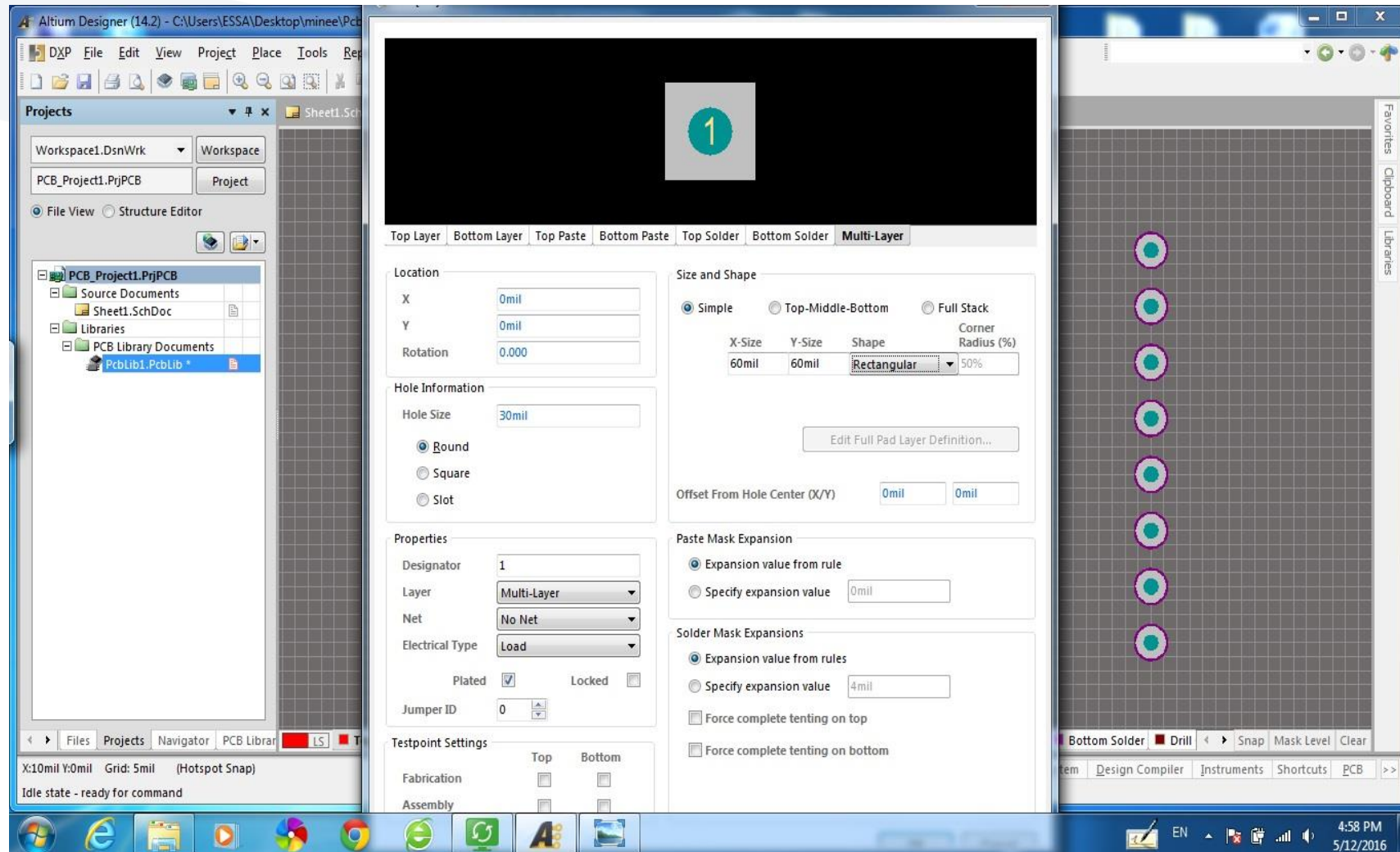


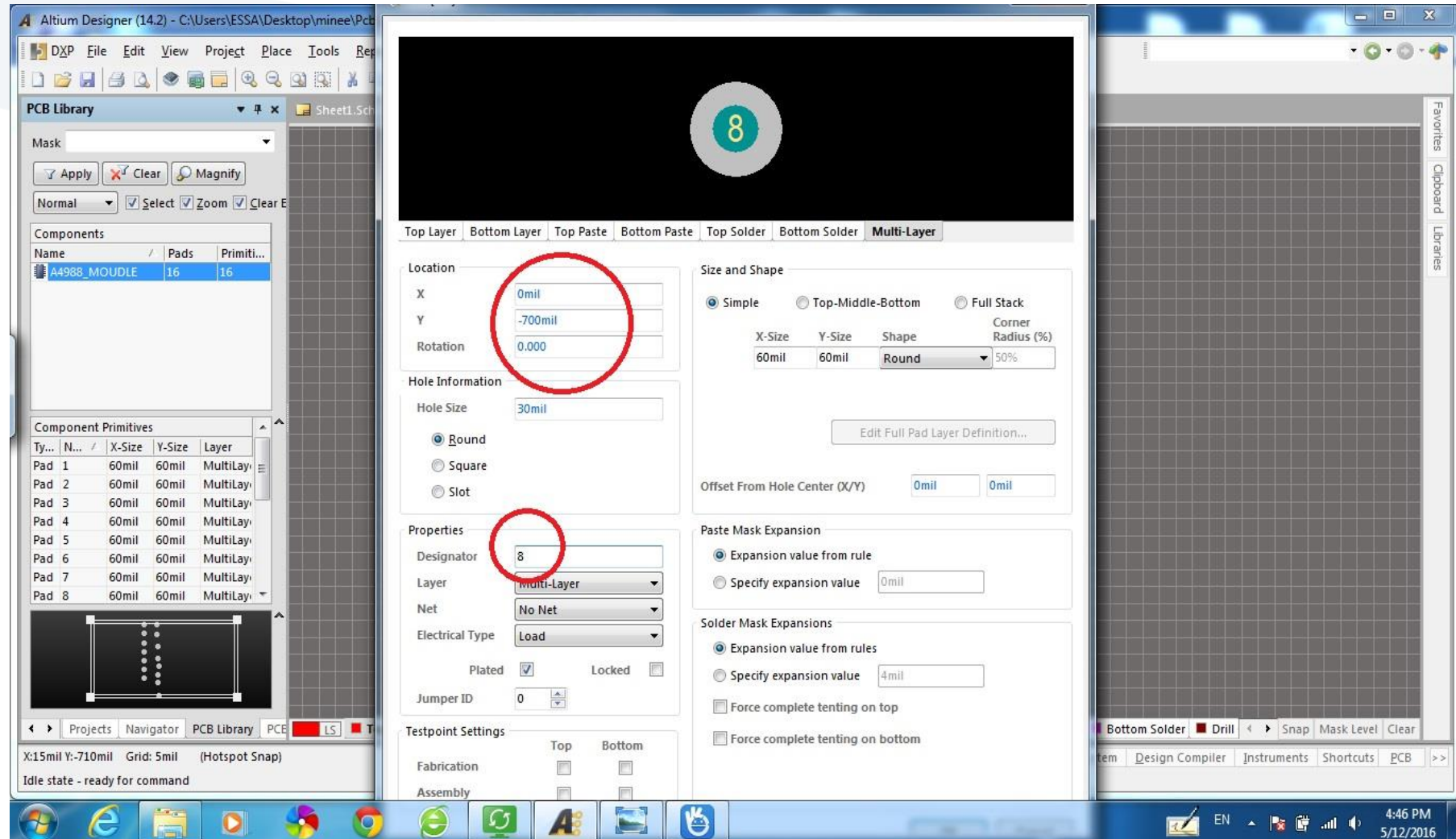
In the Schematic Editor - Change units with the *View --> Toggle Units* menu option •

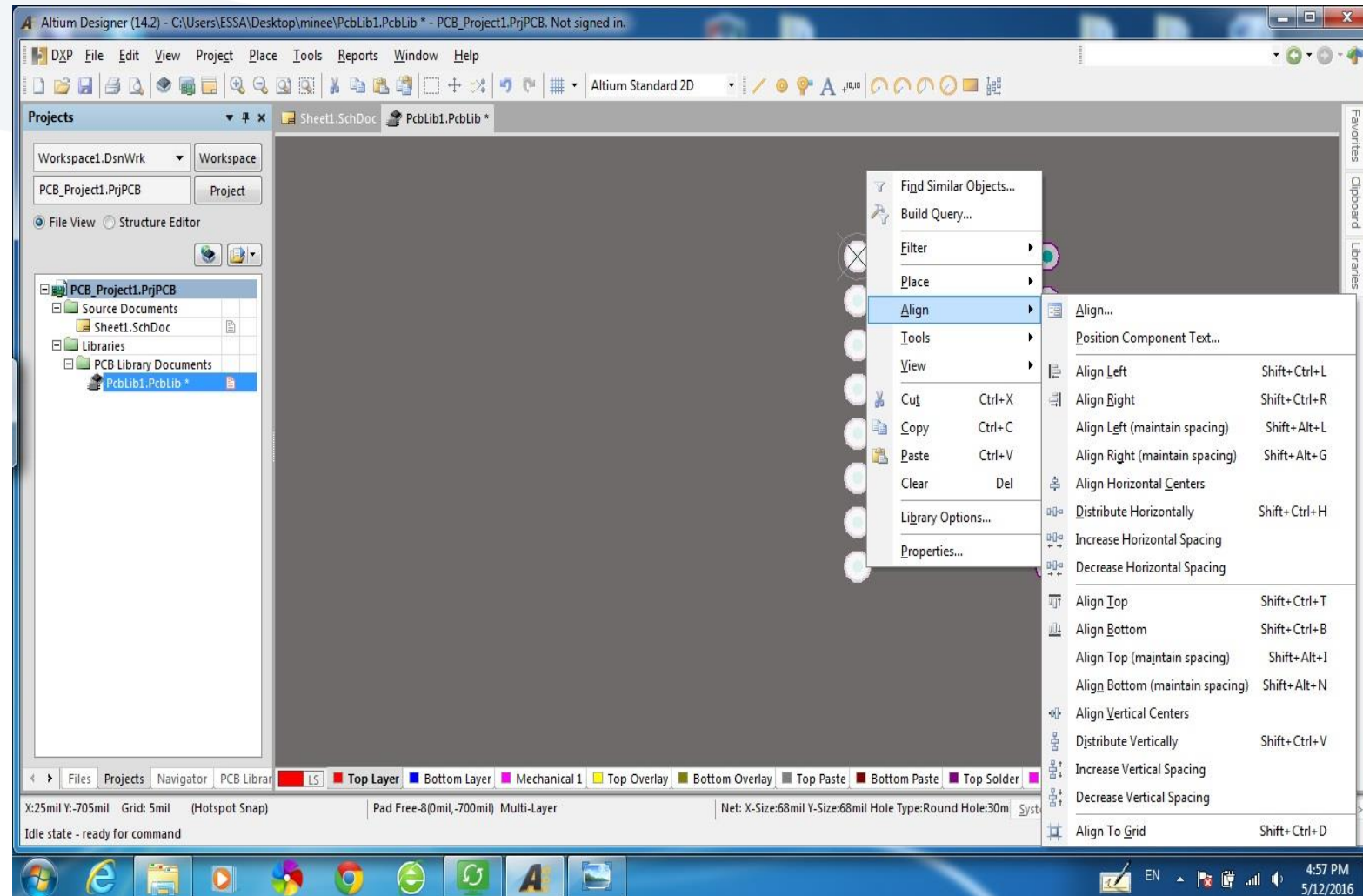
• **In the PCB Editor** - There are three options:

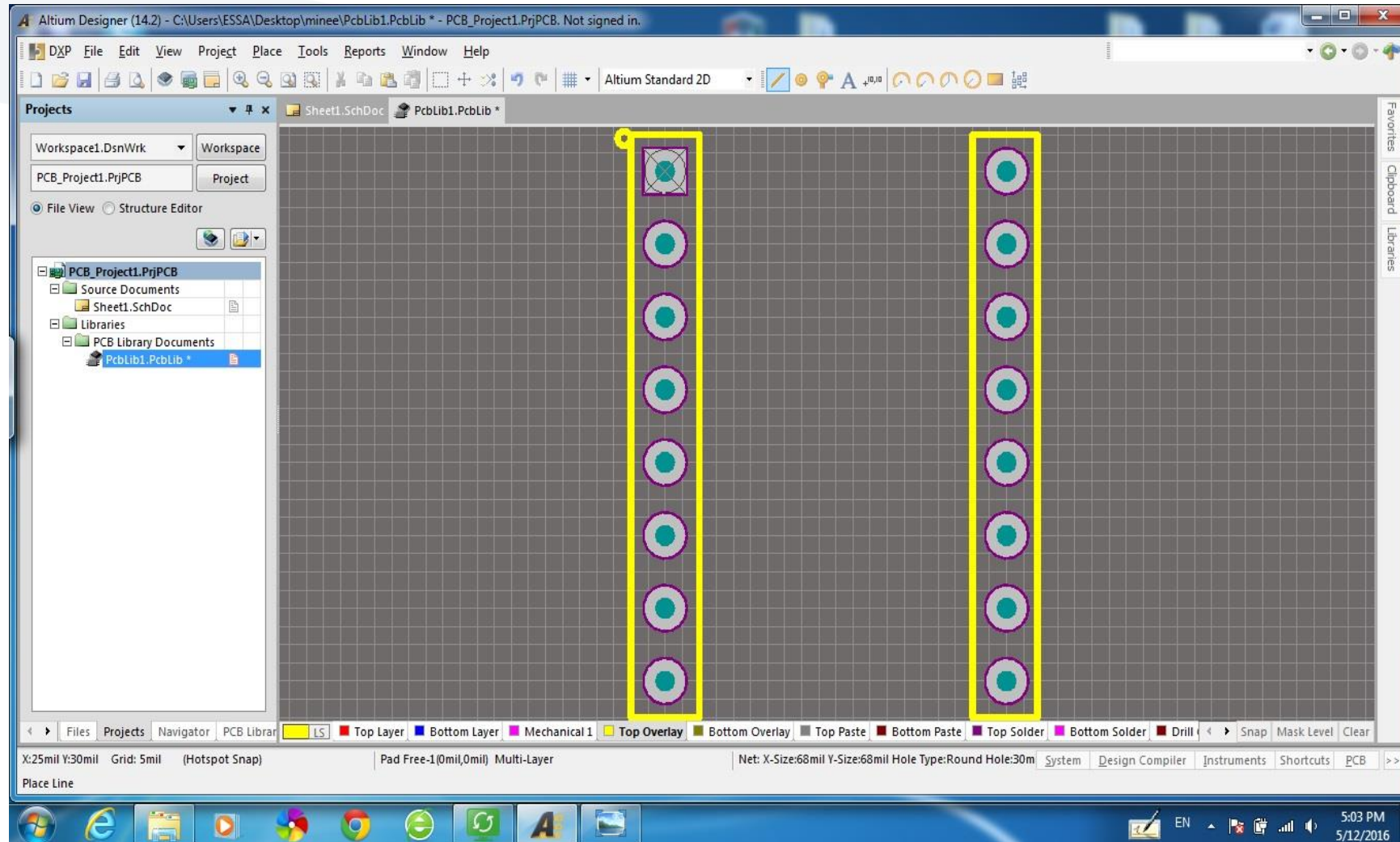
- Press "Q" on the keyboard
- Press "V" followed by "U" on the keyboard
- Use the *View --> Toggle Units* menu option

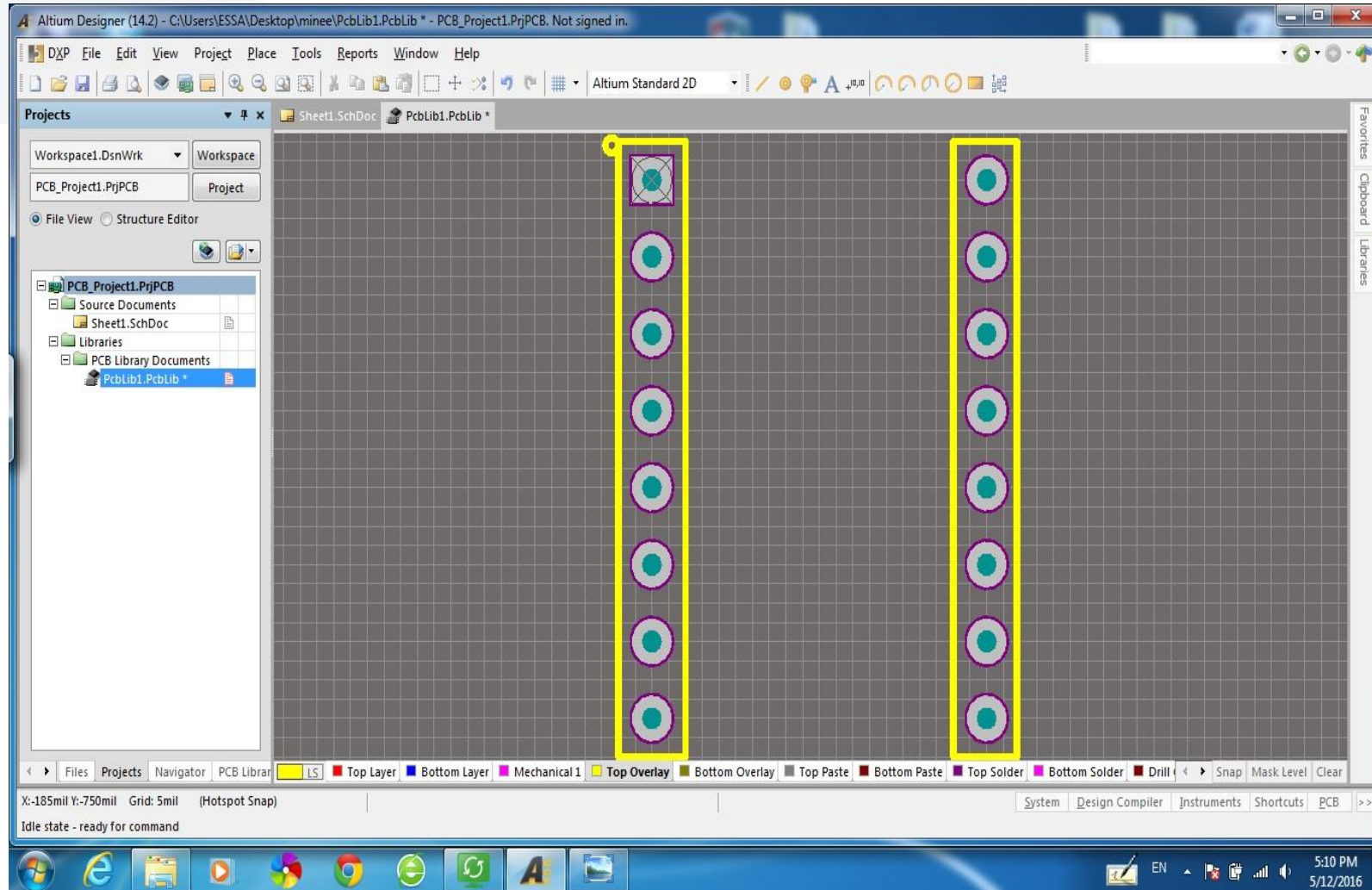


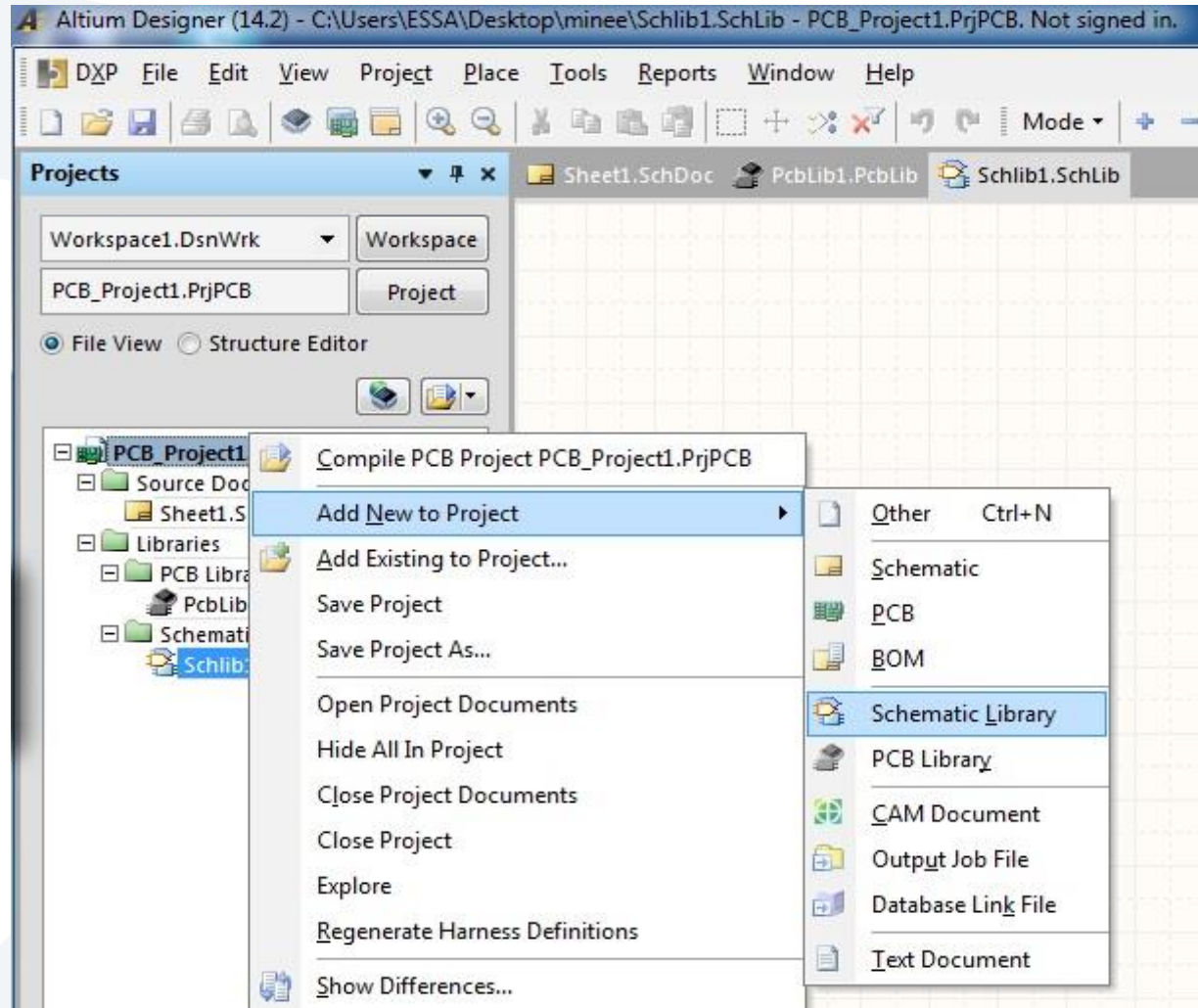


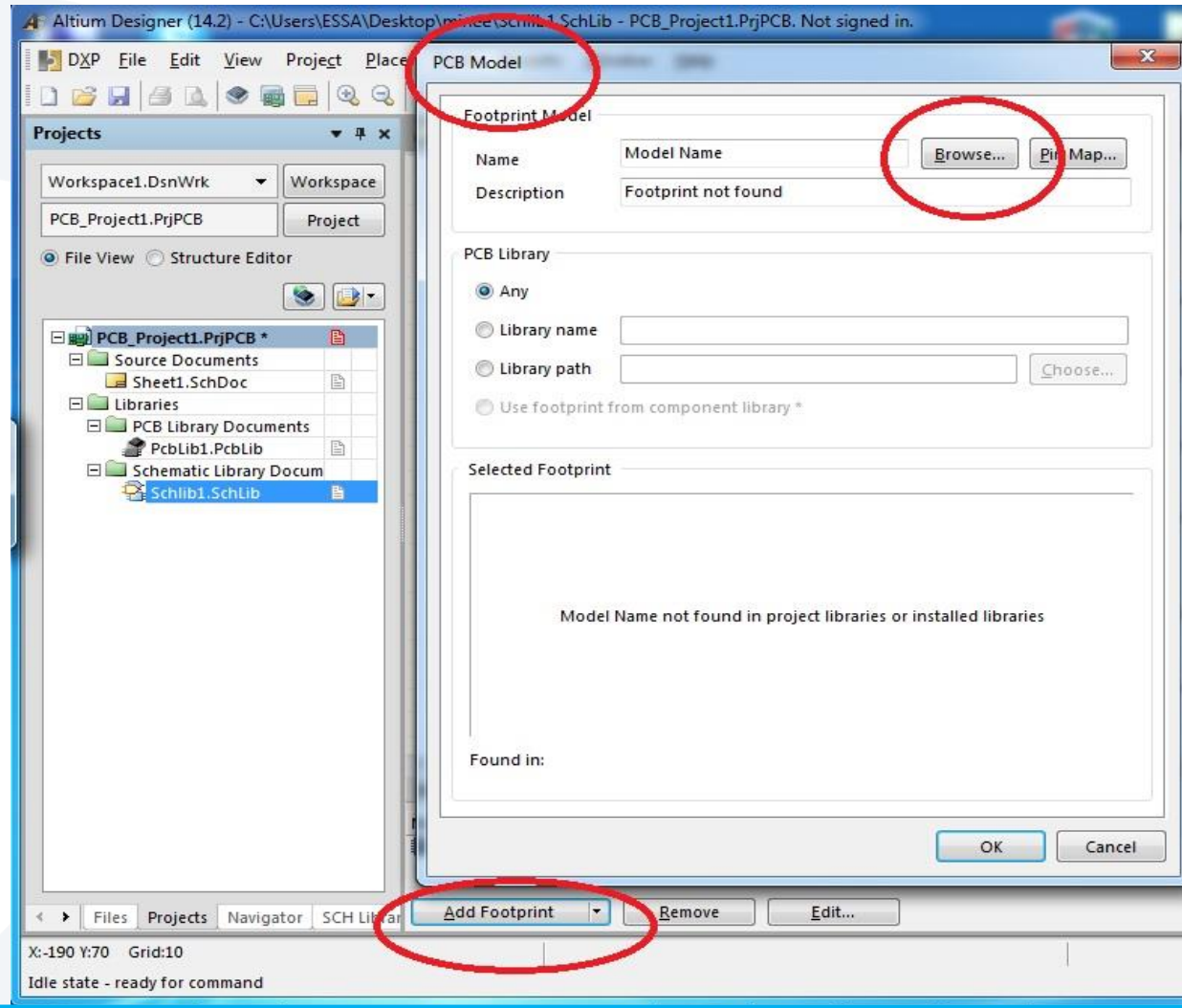


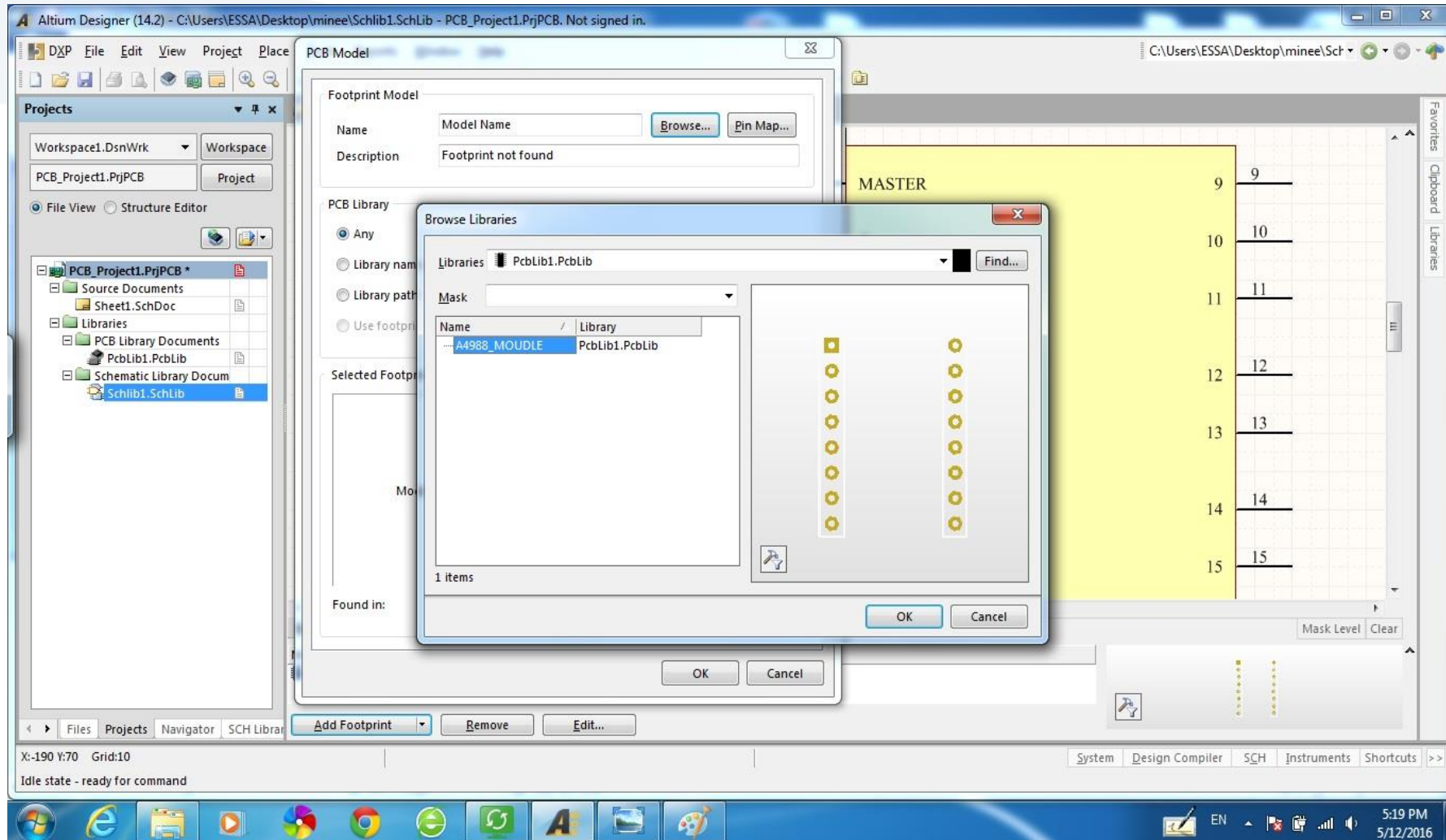


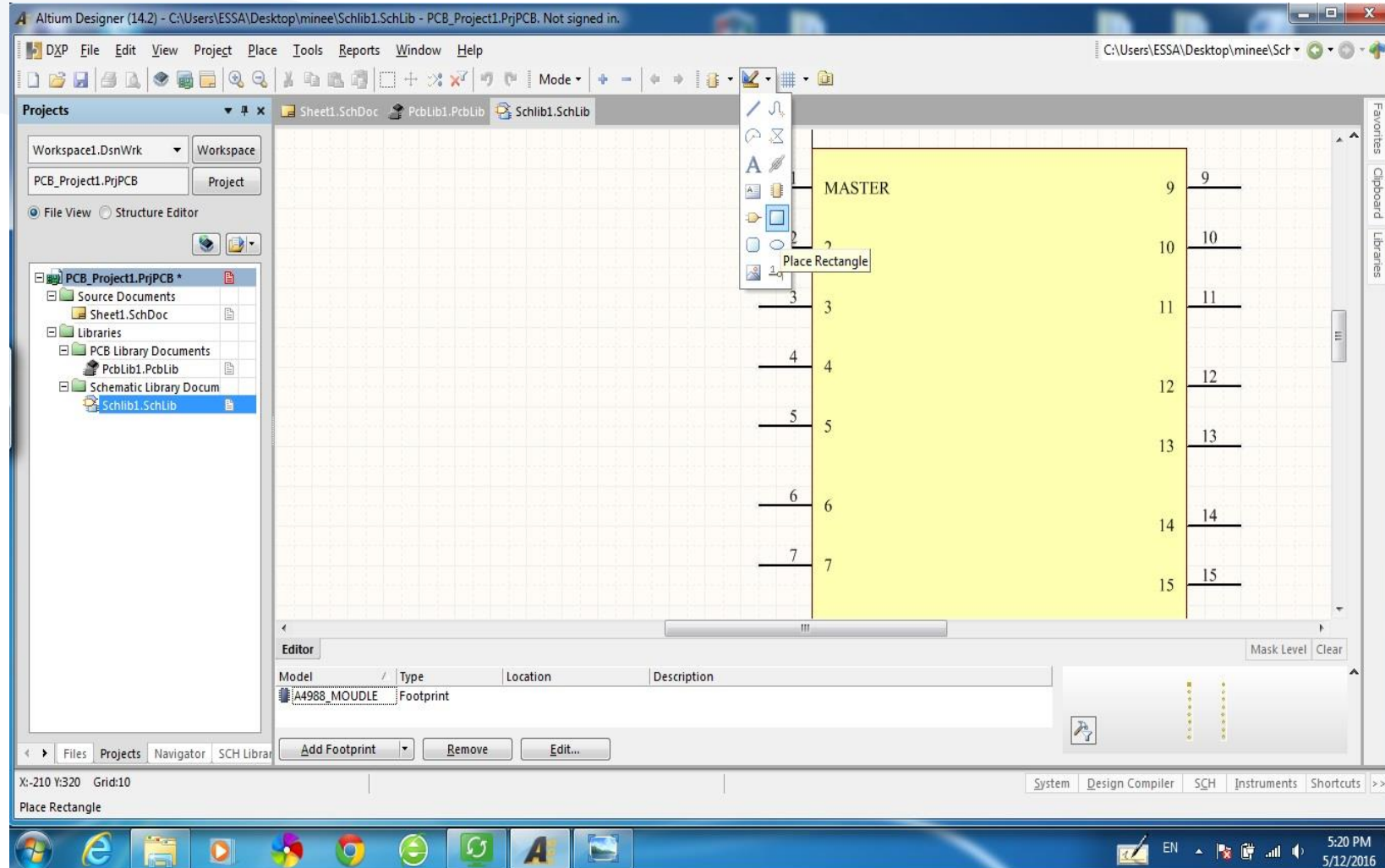


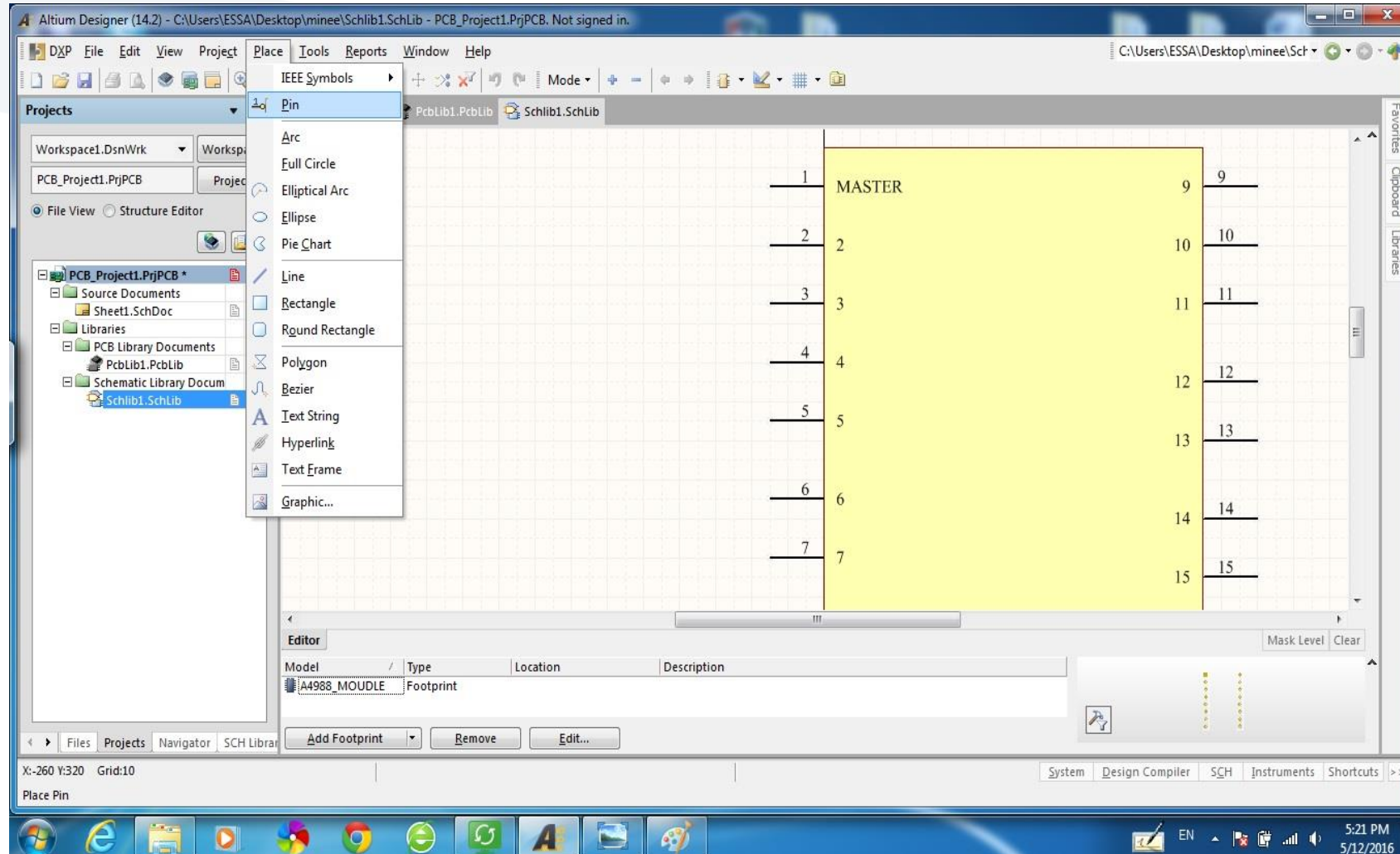


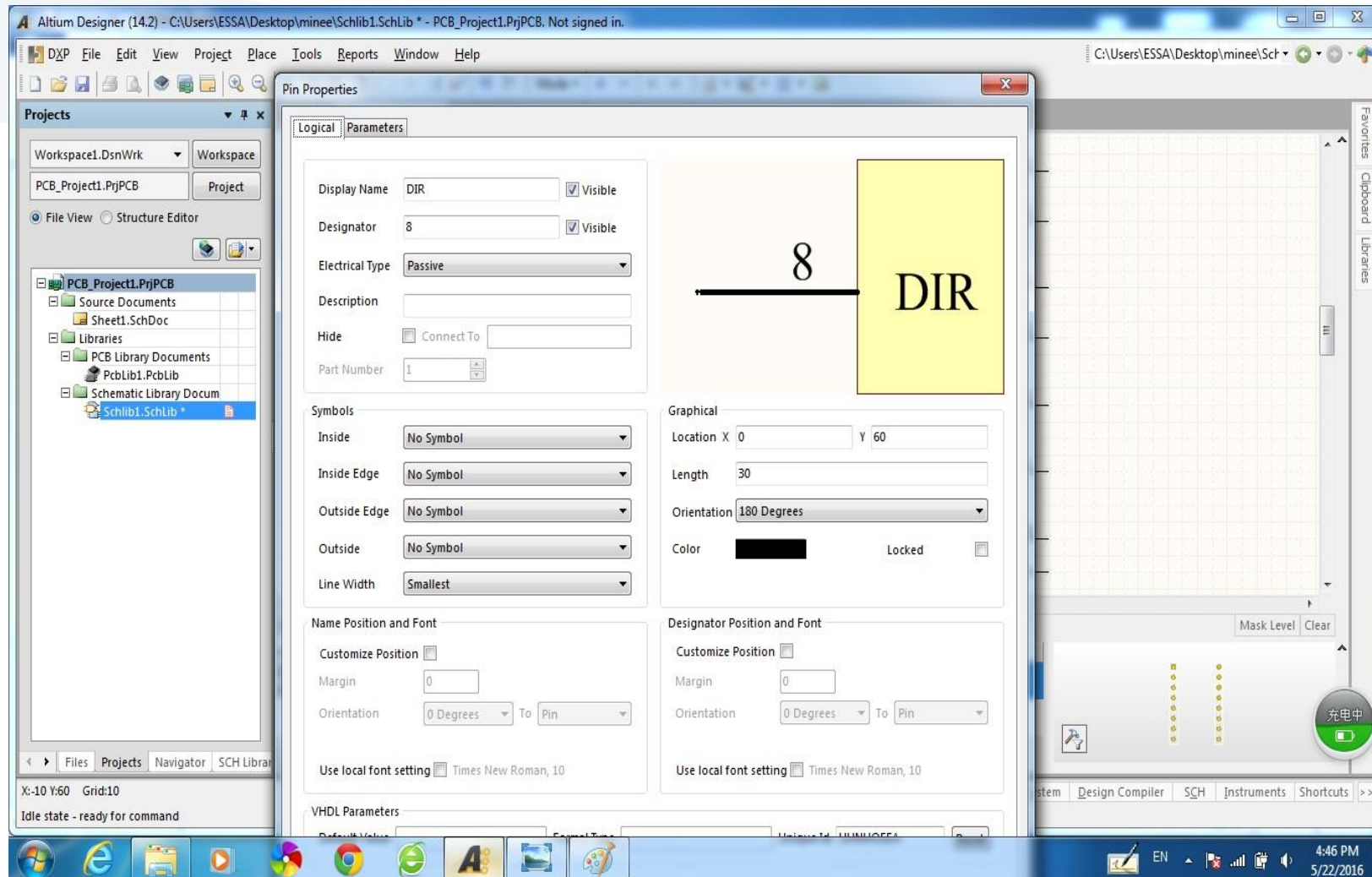


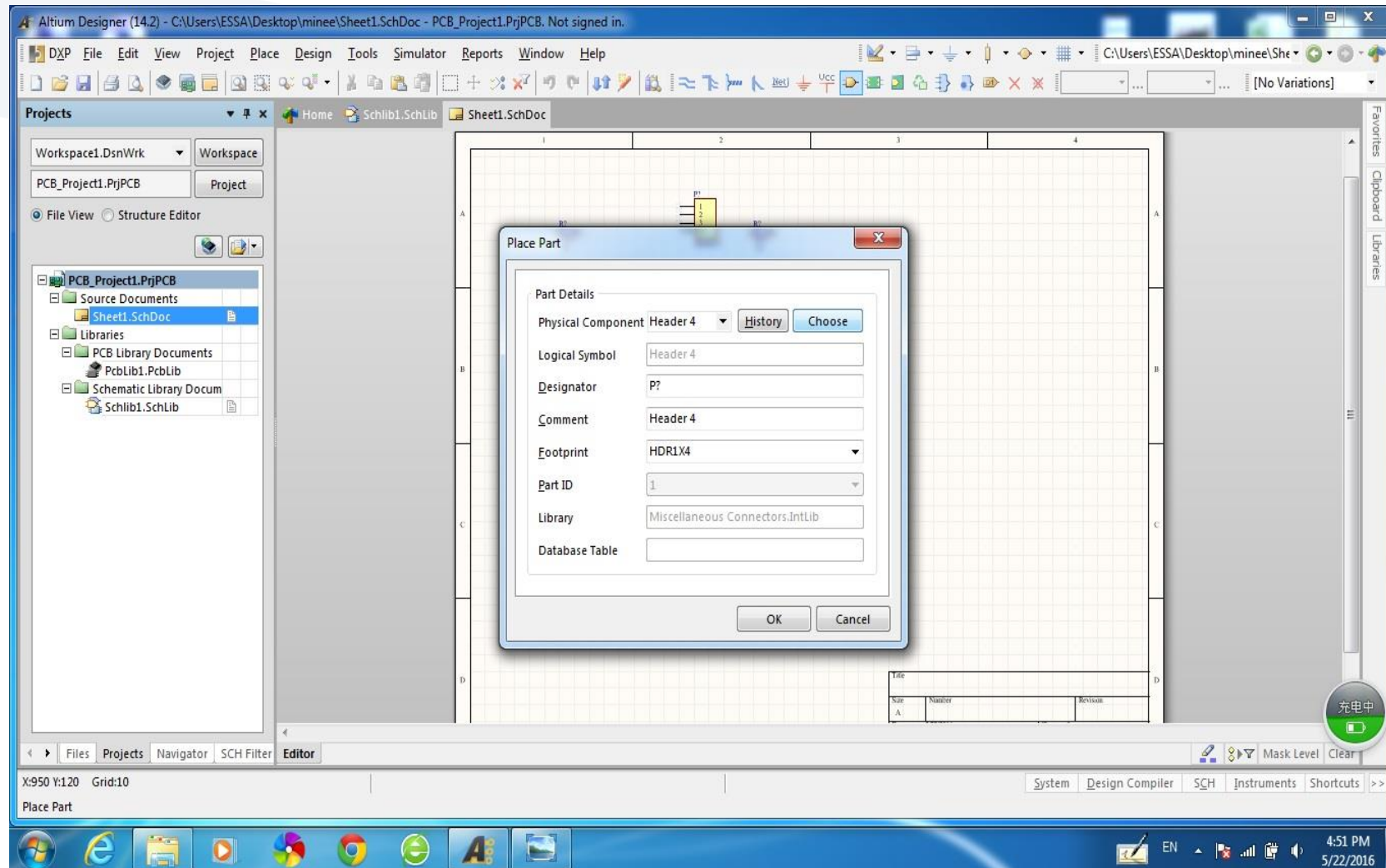


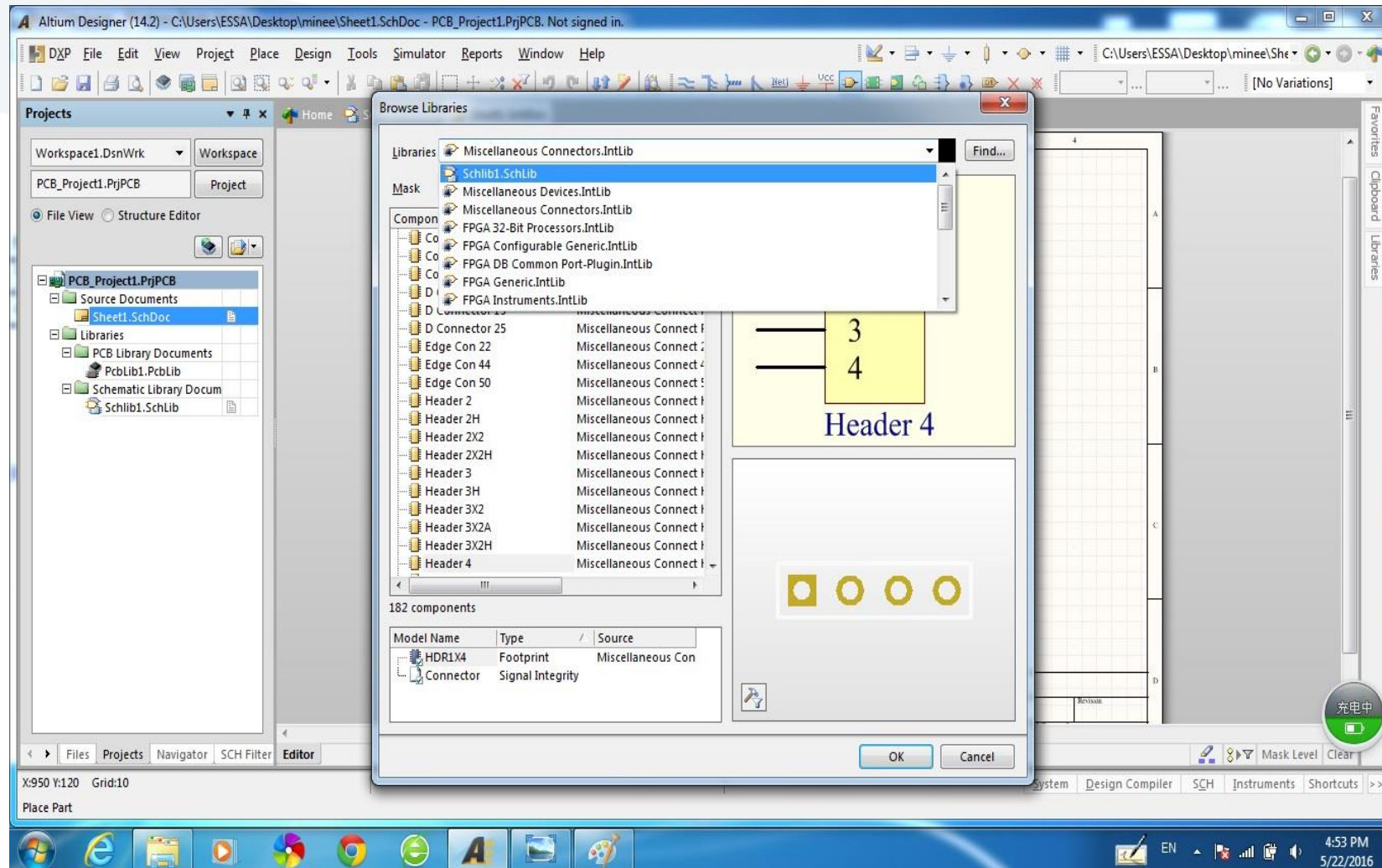


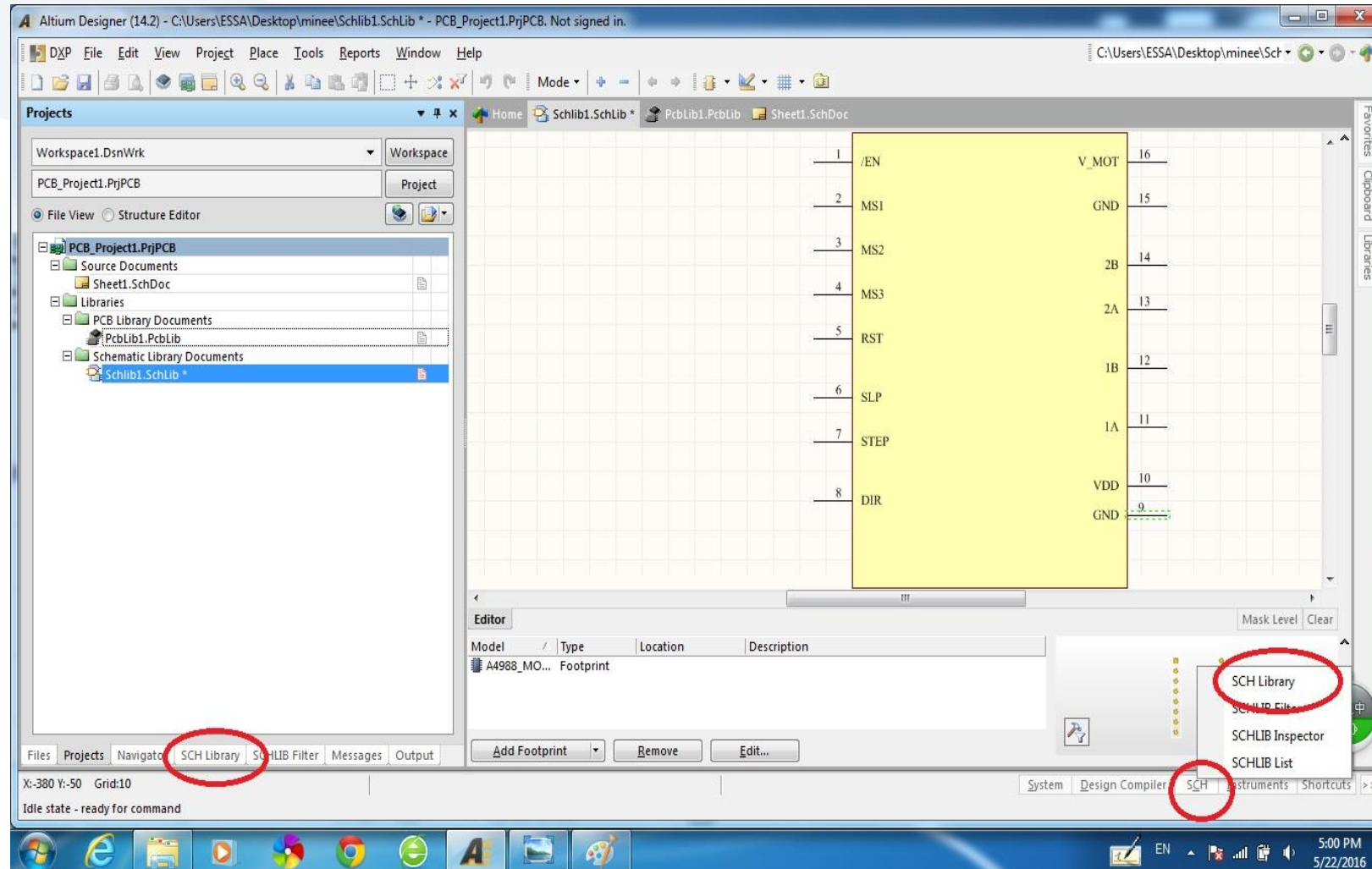


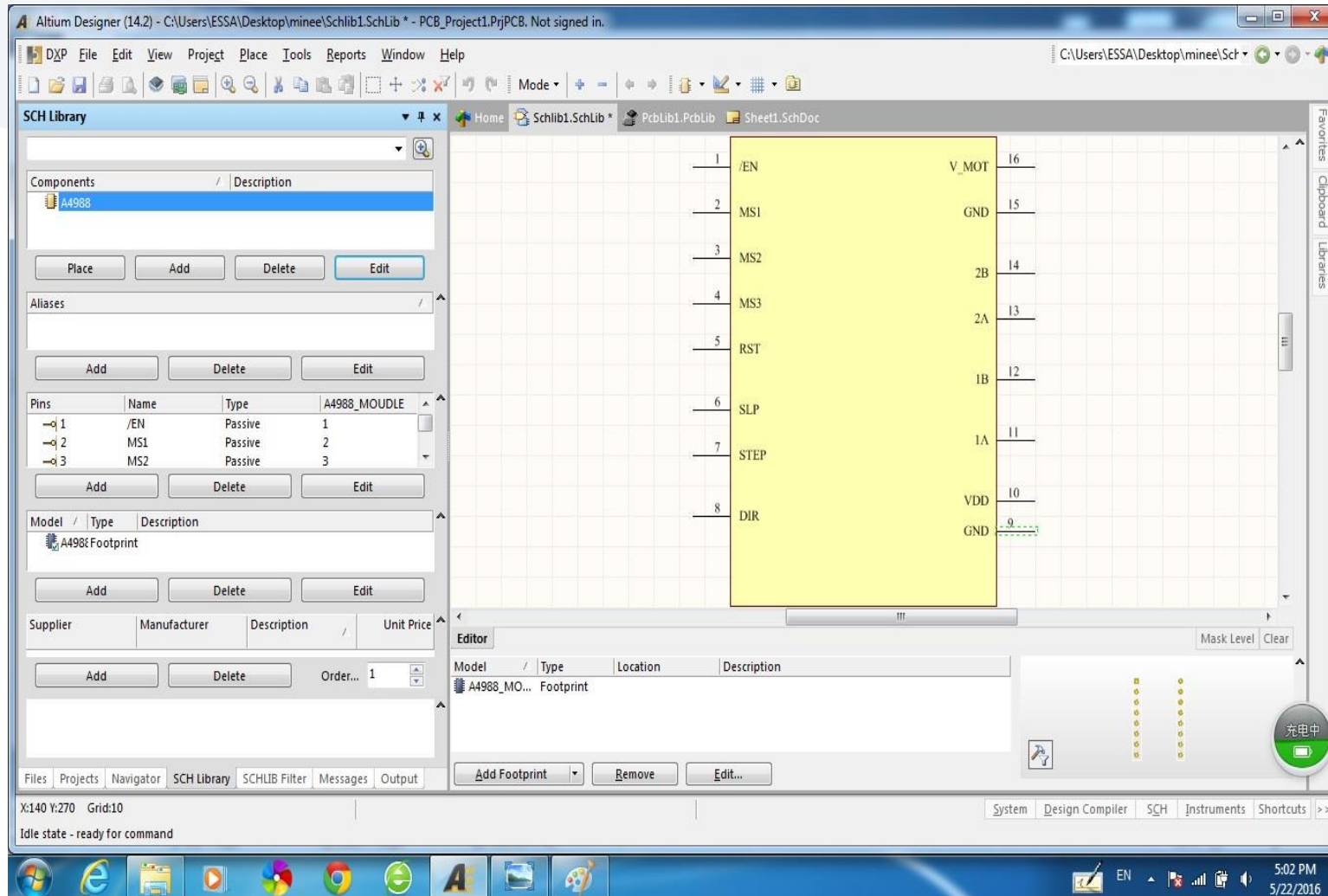


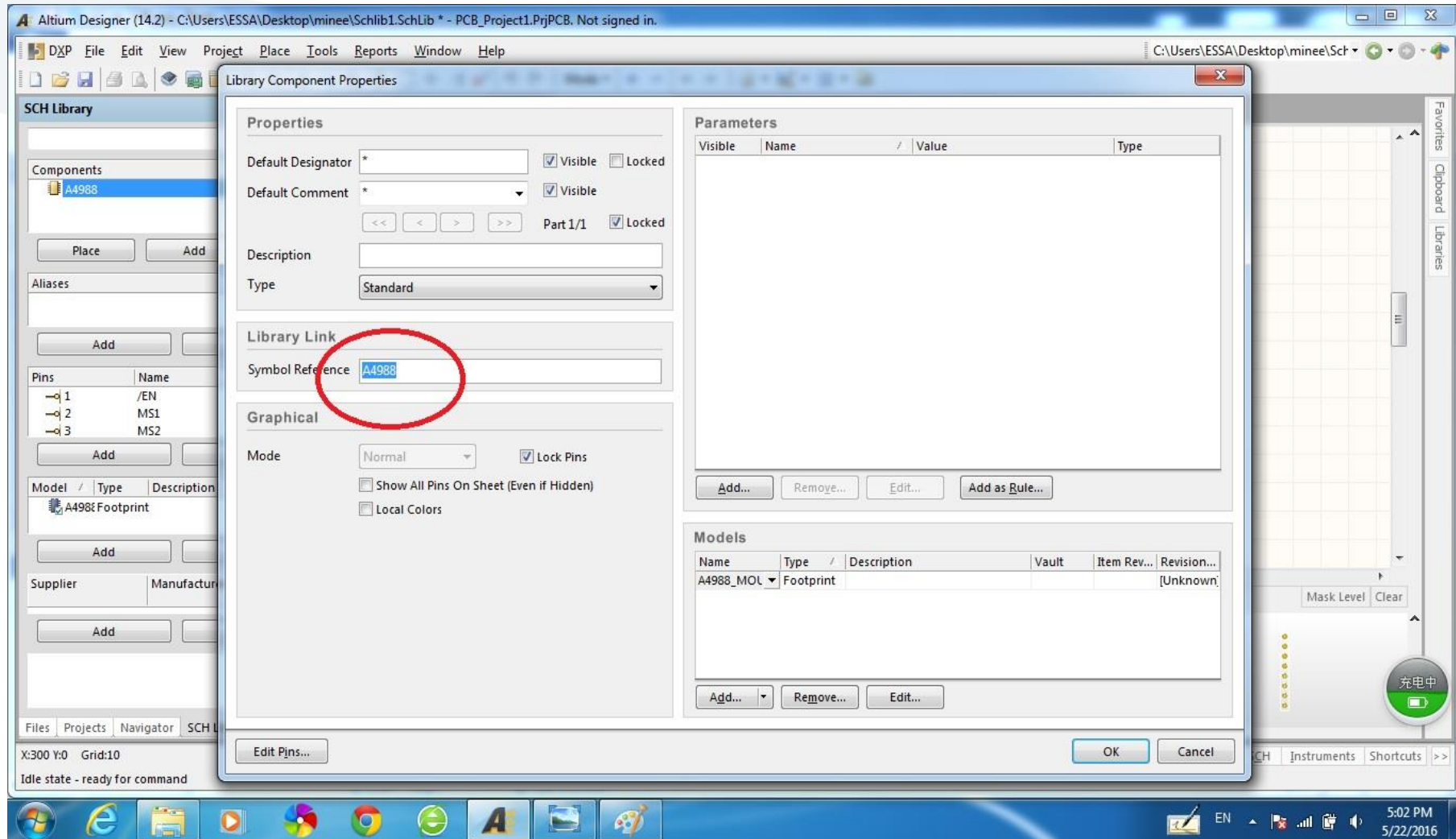


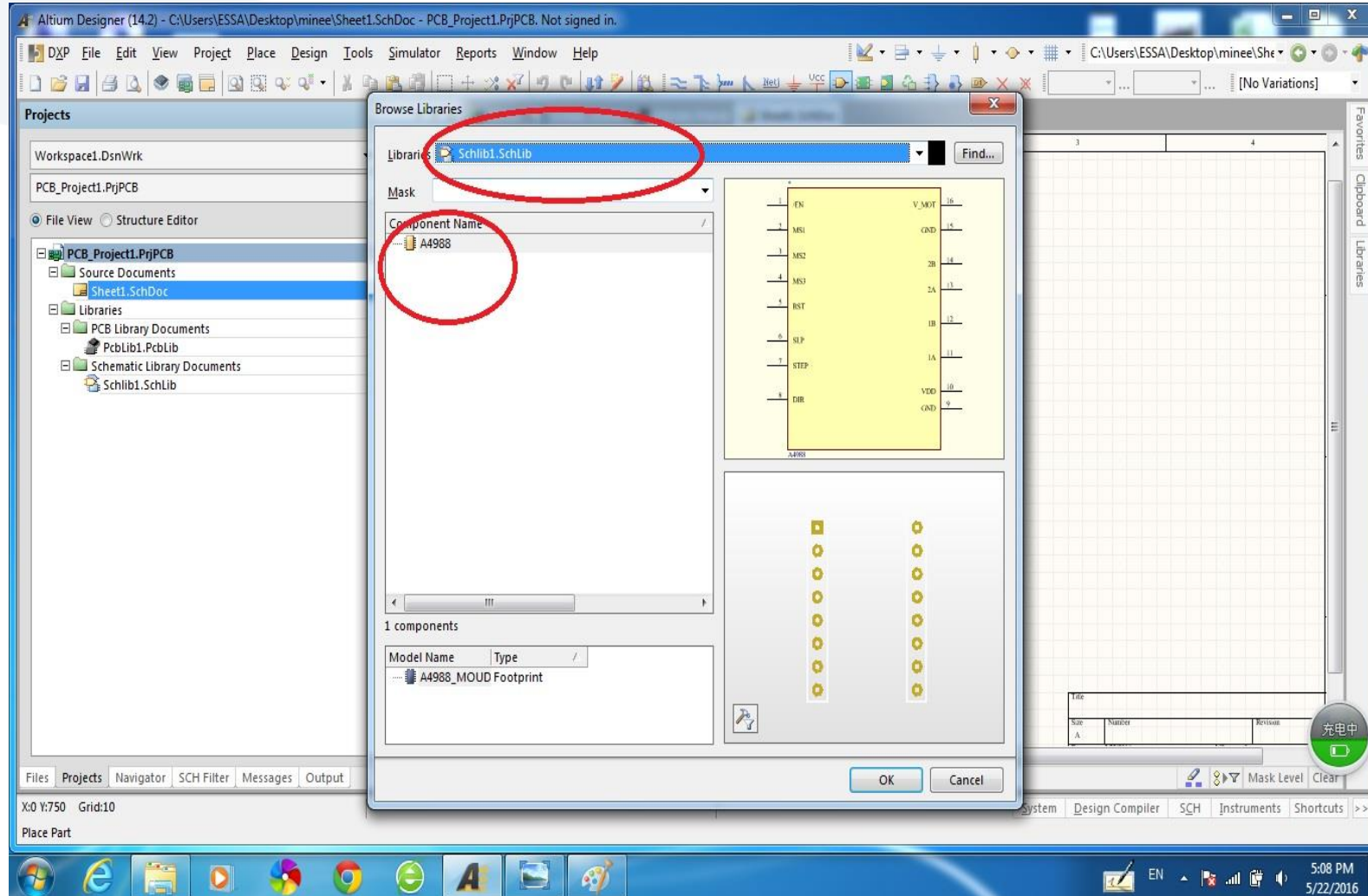


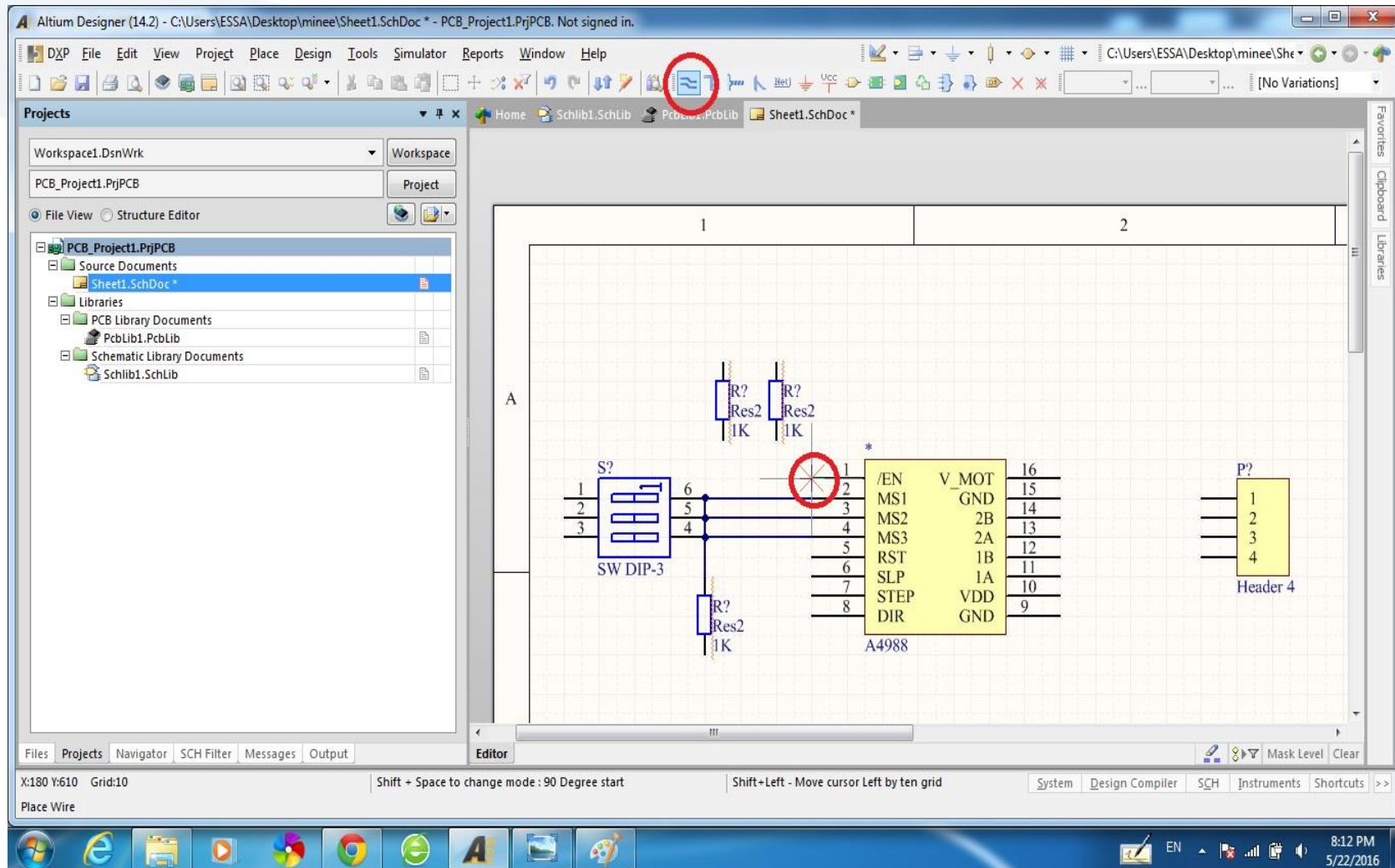












Altium Designer (14.2) - C:\Users\ESSA\Desktop\minee\Sheet1.SchDoc * - PCB_Project1.PrjPCB. Not signed in.

Properties for Schematic Component in Sheet [Sheet1.SchDoc]

Properties

Designator: R? ☒ Visible ☐ Locked

Comment: Res2 ☒ Visible

Description: Resistor

Unique Id: OMMPNVRK

Type: Standard

Link to Library Component ☐ Use Vault Component

Design Item ID: Res2

☒ Library Name: Miscellaneous Devices.IntLib

☐ Table Name:

Graphical

Location X: 120 Y: 640

Orientation: 90 Degrees ☐ Locked

Mode: Normal ☒ Lock Pins ☐ Mirrored

☐ Show All Pins On Sheet (Even if Hidden)

☐ Local Colors

Parameters

Visible	Name	Value	Type
<input type="checkbox"/>	LatestRevisionDate	17-Jul-2002	STRING
<input type="checkbox"/>	LatestRevisionNote	Re-released for DXP Platform.	STRING
<input type="checkbox"/>	PackageDescription	Axial Device, Thru-Hole; 2 Leads; 0.4	STRING
<input type="checkbox"/>	PackageReference	AXIAL-0.4	STRING
<input type="checkbox"/>	Published	8-Jun-2000	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING
<input checked="" type="checkbox"/>	Value	10K	STRING

Models

Name	Type	Description	Vault	Item Rev...	Revision...
AXIAL-0.4	Footprint	Axial Device, Thru-Hole; 2 Leads; 0.4			[Unknown]
Res	Signal Inter.				[Unknown]
RESISTOR	Simulation	Resistor			[Unknown]

Workspace1.DsnWrk

PCB_Project1.PrjPCB

File View Structure Editor

PCB_Project1.PrjPCB

Source Documents

Sheet1.SchDoc *

Libraries

PCB Library Documents

PcbLib1.PcbLib

Schematic Library Documents

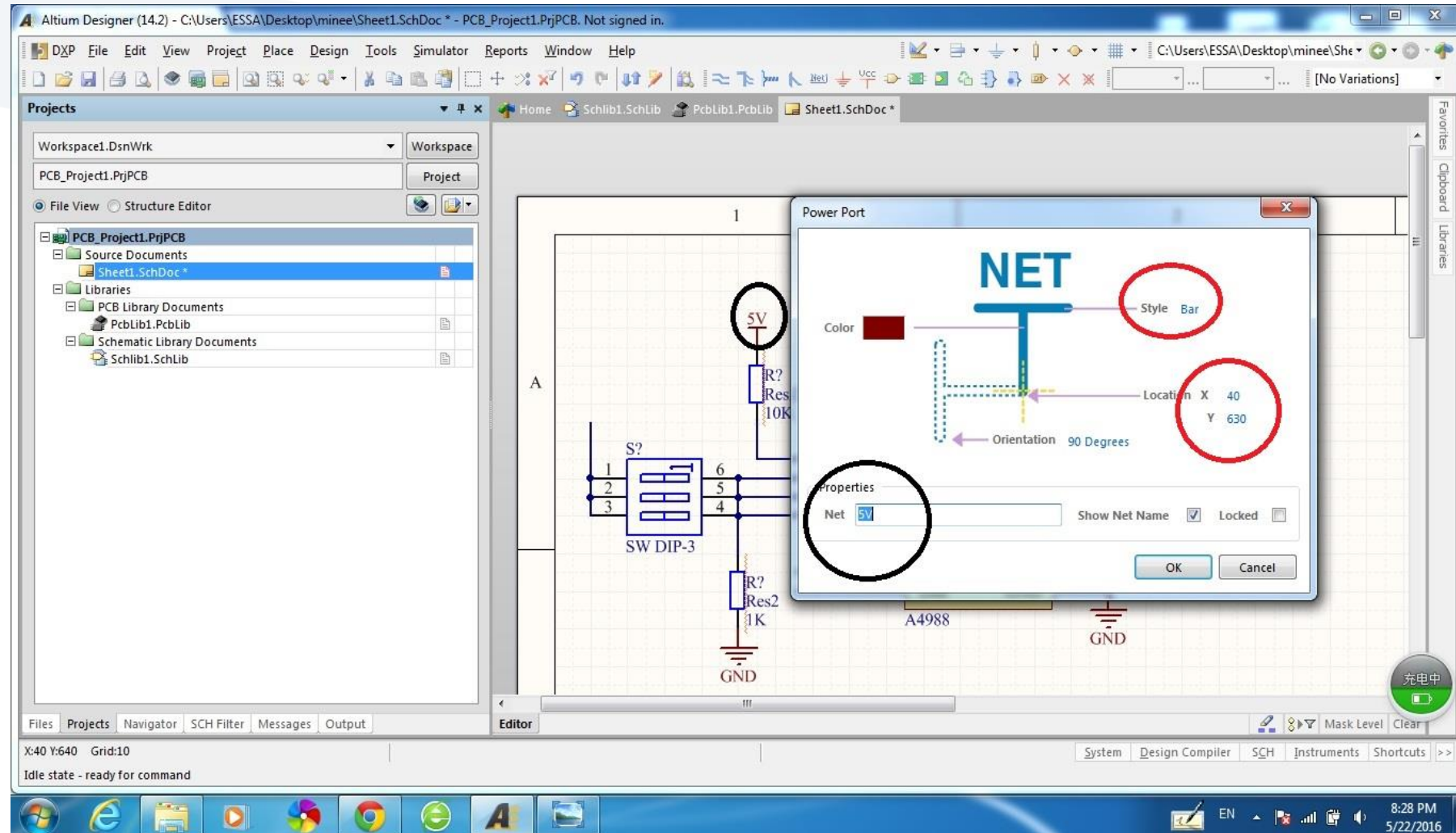
SchLib1.SchLib

Files Projects Navigator SCH

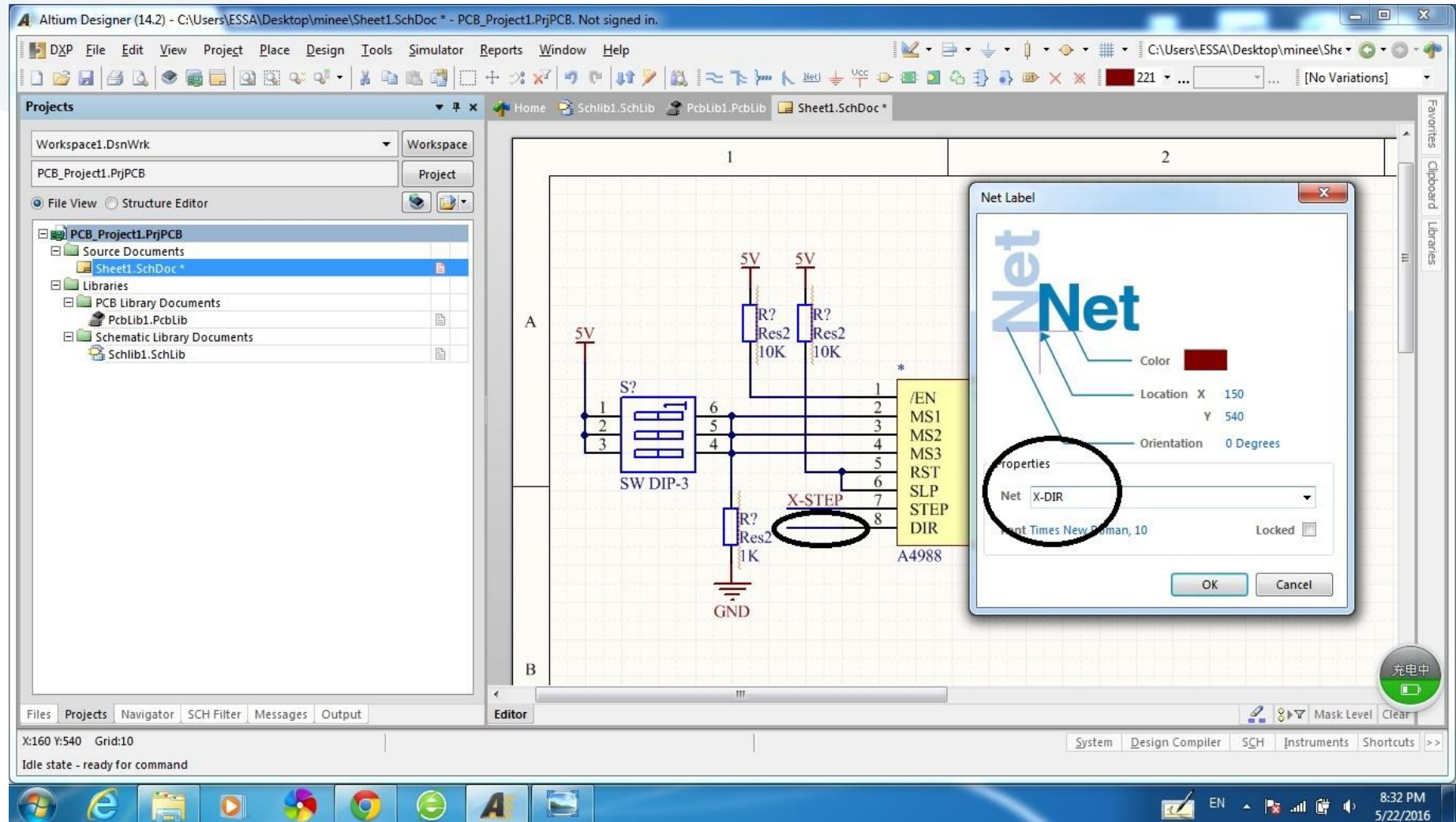
X:130 Y:650 Grid:10

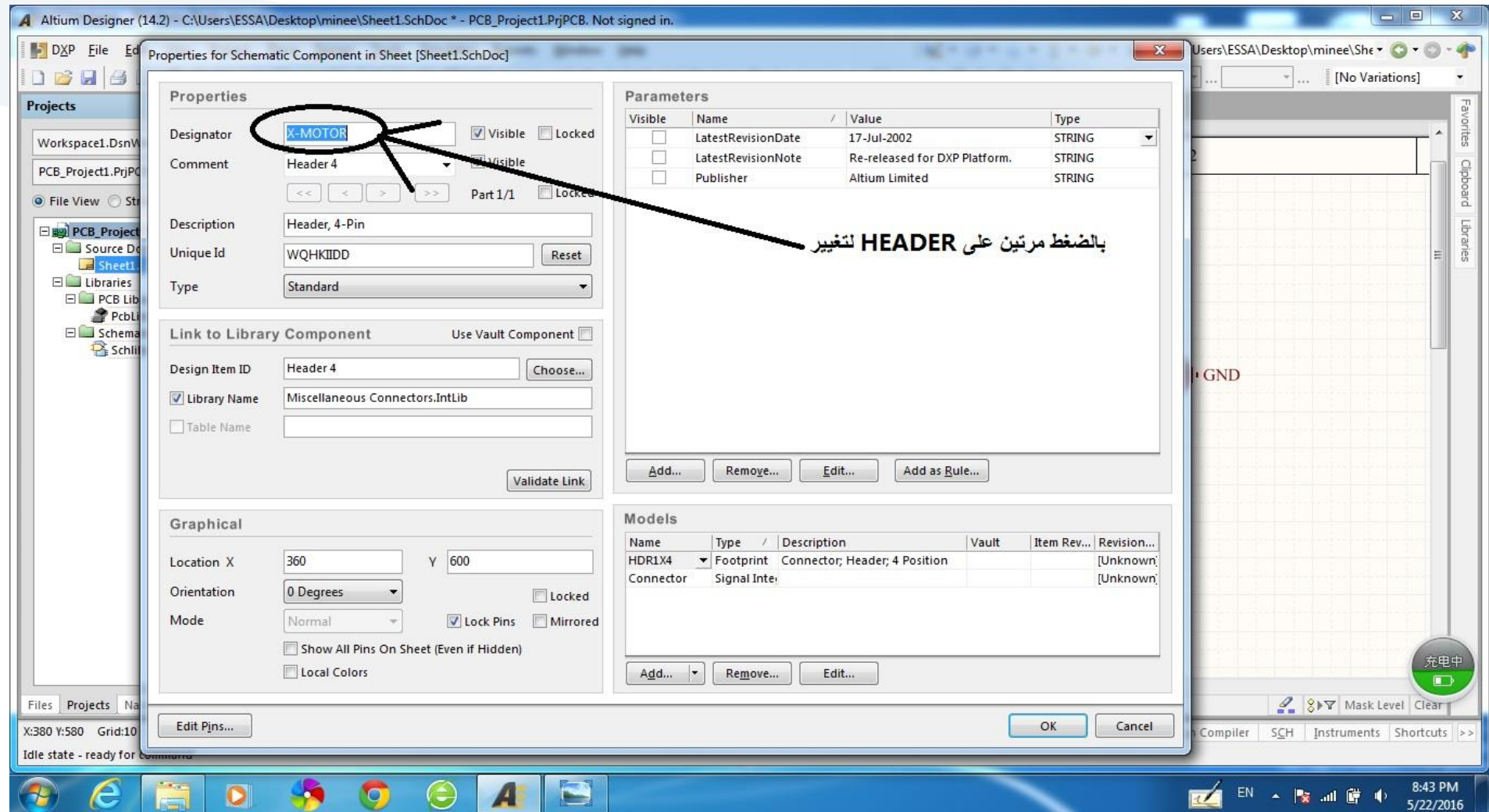
Idle state - ready for command

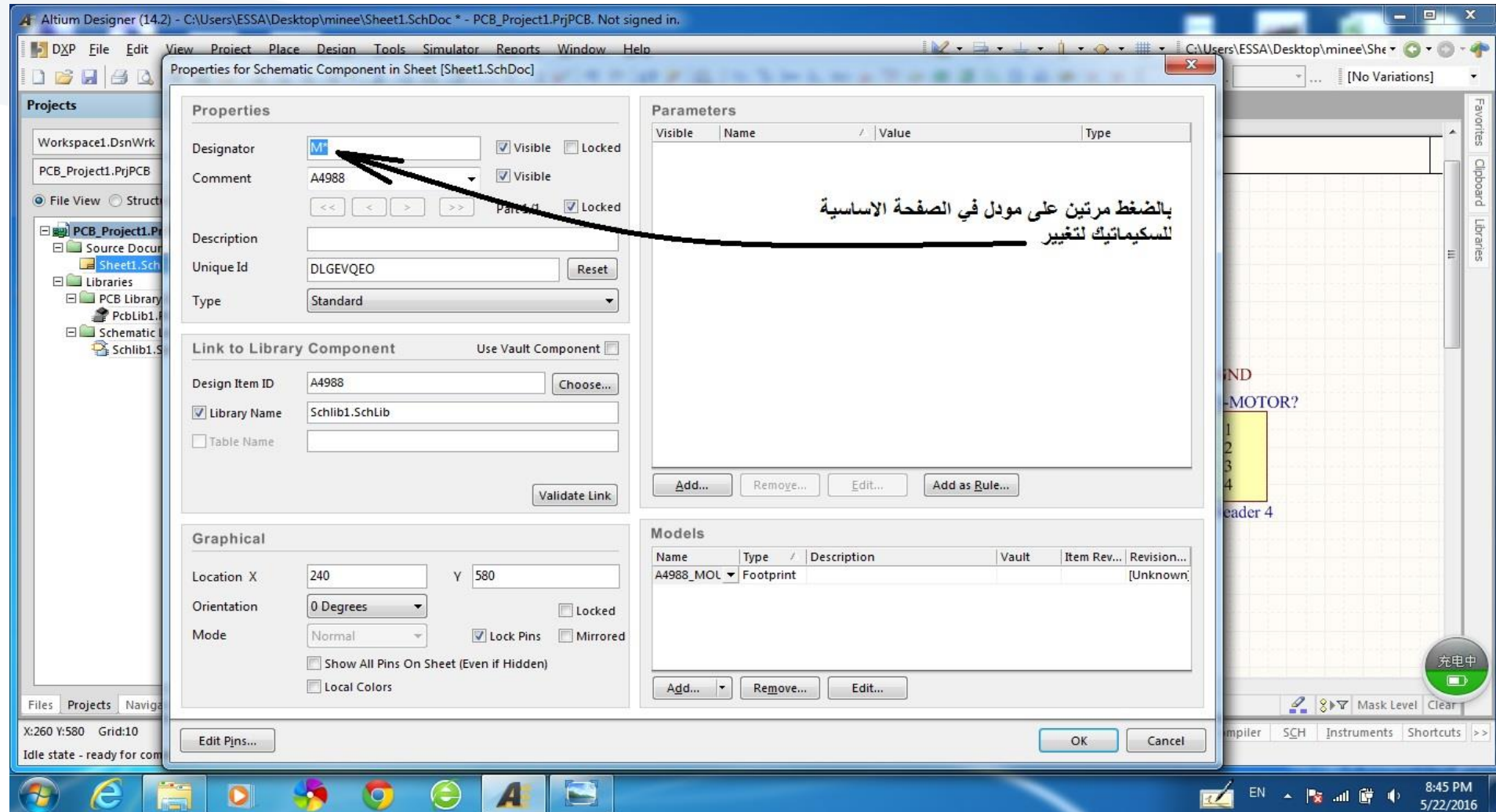
EN 8:27 PM 5/22/2016











Altium Designer (14.2) - C:\Users\ESSA\Desktop\minee\Sheet1.SchDoc * - PCB_Project1.PrjPCB. Not signed in.

DXP File Edit View Project Place Design Tools Simulator Reports Window Help

Projects: Workspace1.DsnWrk, PCB_Project1.PrjPCB, File View, Structure Editor

PCB_Project1.PrjPCB
 Source Documents: Sheet1.SchDoc *
 Libraries: PCB Library Documents, PcbLib1.PcbLib, Schematic Library Documents, SchLib1.SchLib

Schematic Diagram: A circuit diagram showing a microcontroller (A4988) connected to a stepper motor (X-MOTOR). The microcontroller is connected to a switch (SW DIP-3) and a motor driver (A4988). The motor driver is connected to a motor (X-MOTOR). The diagram includes various components like resistors (Res2, 10K), capacitors (C1, 100K), and a header (Header 4).

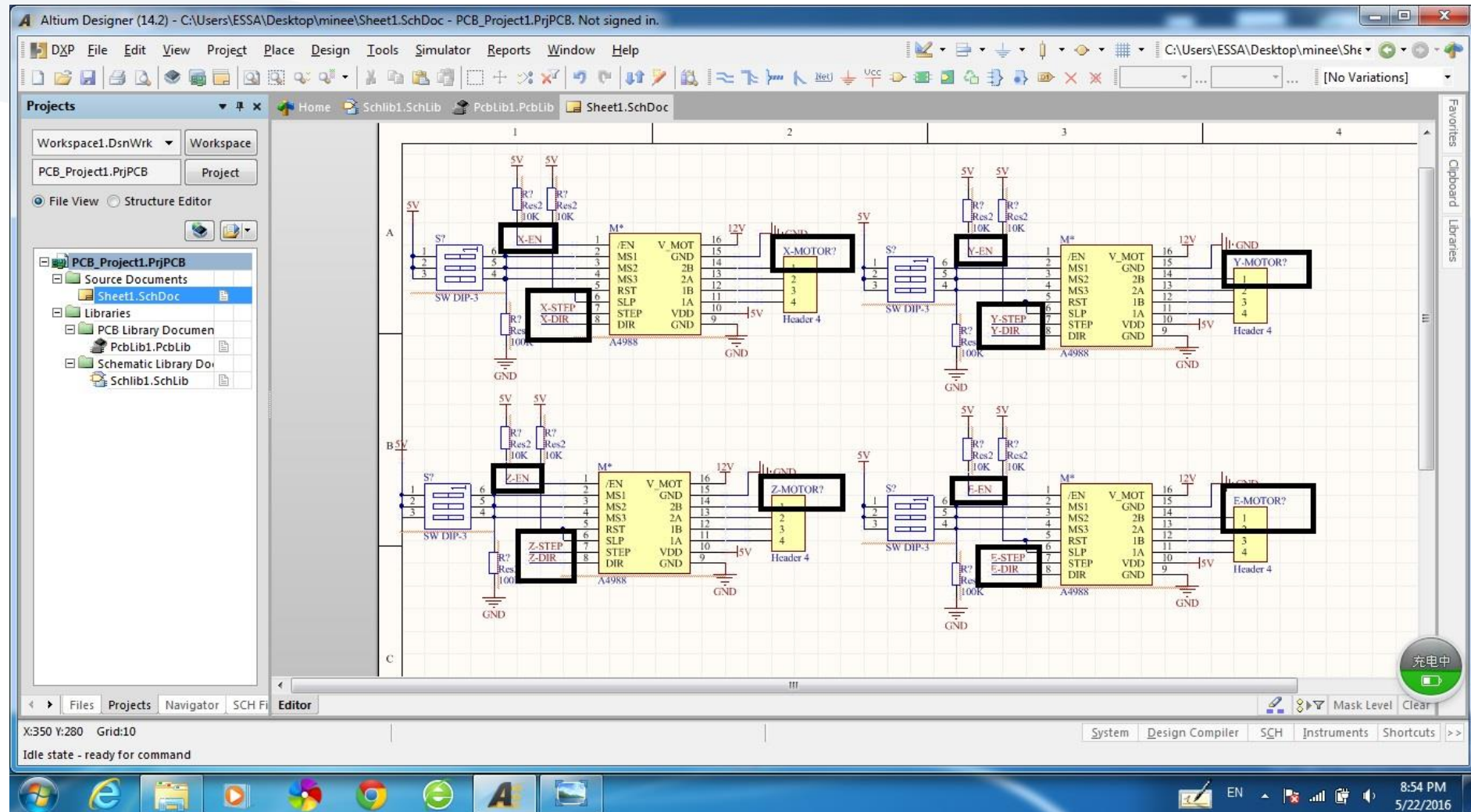
لتحرك العناصر كقالب واحد او نسخها و لصقها باستخدام
 كونترول سي او كونترول في

Files Projects Navigator SCH Filter Messages Output Editor

X:410 Y:510 Grid:10
 Idle state - ready for command

System Design Compiler SCH Instruments Shortcuts >>

8:48 PM 5/22/2016



نهاية المحاضرة
Thanks