Introduction to Altium Designer 15.1

Prepared by:

Mahyar Bayran

Principles of Electronics

Sharif University of Tech.

Fall 2015

Content

- Goals
- Intro
- Creating PCB Project ,Schematic and PCB
- Adding Libraries
- Designing Schematics
- Designing PCB
- Common errors and tips
- Guide to online ordering a PCB
- References

Goals

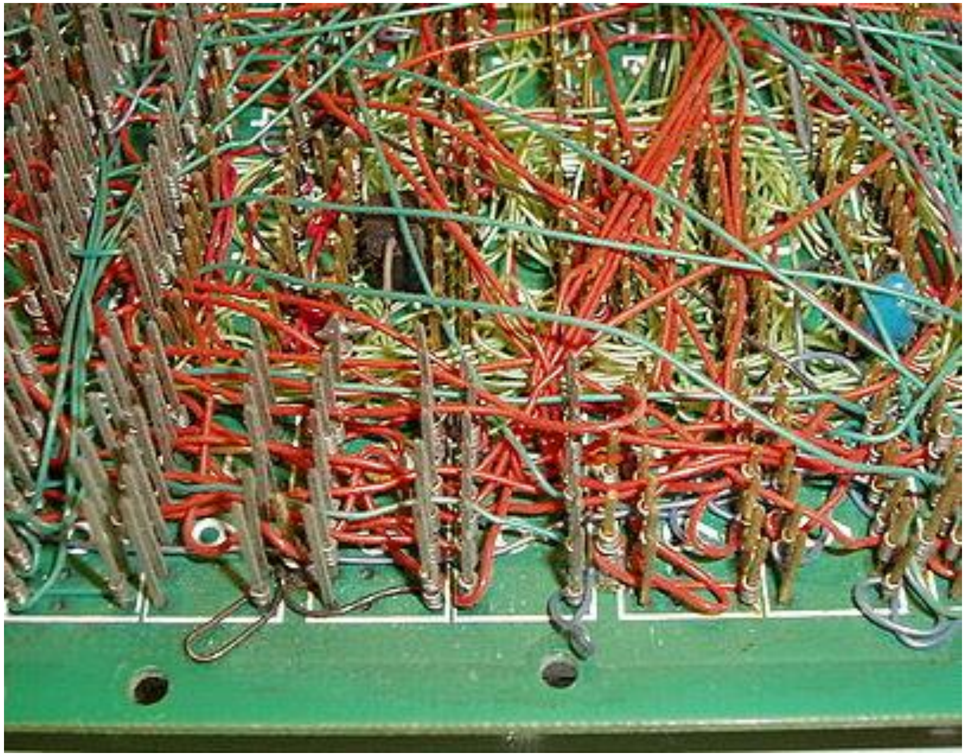
- Ability to design simple Schematic and PCB.
- Earn sufficient skills to do your course projects.
- Learn common errors.

Altium and PCB

- **Altium Designer** is an electronic design automation software package for printed circuit board, FPGA and embedded software design, and associated library and release management automation.
- A **Printed Circuit Board** (**PCB**) mechanically supports and electrically connects electric components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate.

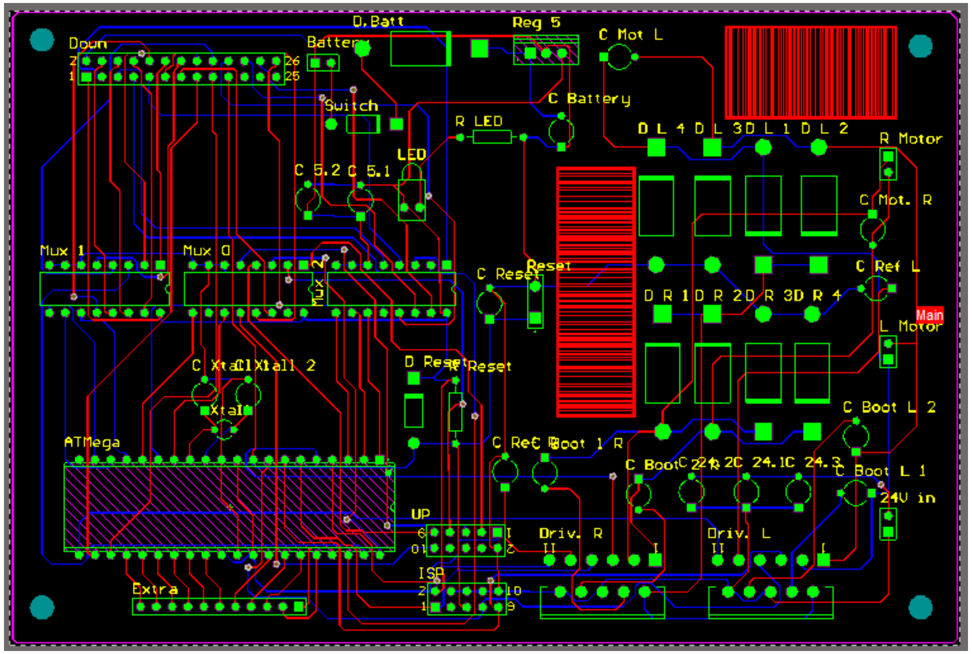
Early PCBs

- Before the advent of the PCB, circuits were constructed through a laborious process of point-to-point wiring.
- This led to frequent failures at wire junctions and short circuits when wire insulation began to age and crack.



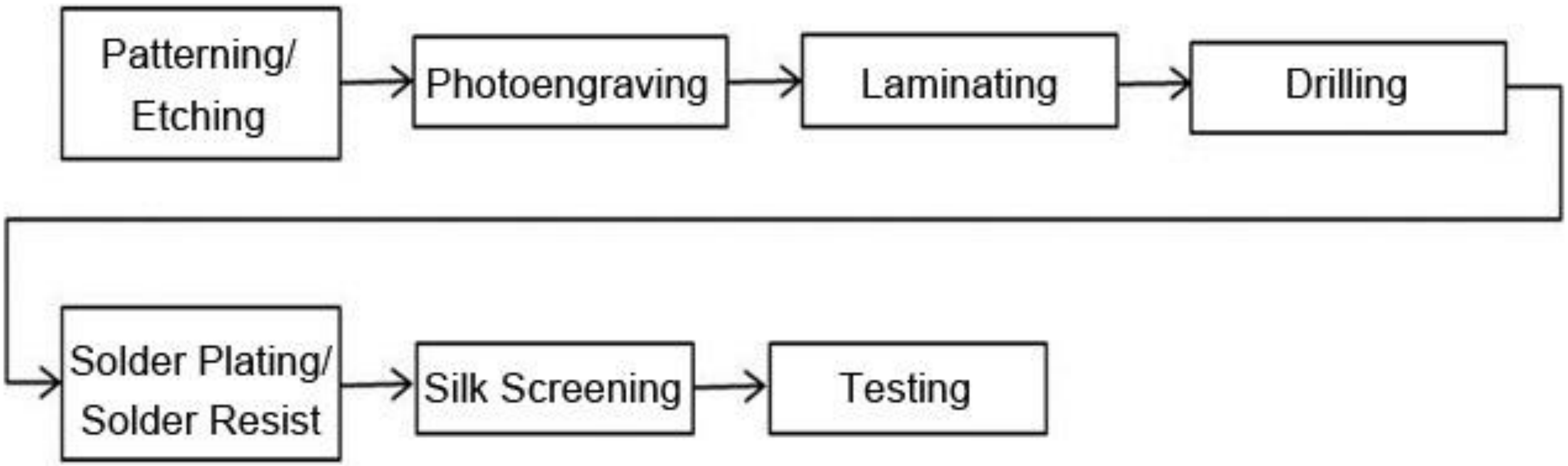
PCB samples

- You can see samples of Printed Circuit Board (PCB) at below: (holes are connected with conductive tracks)



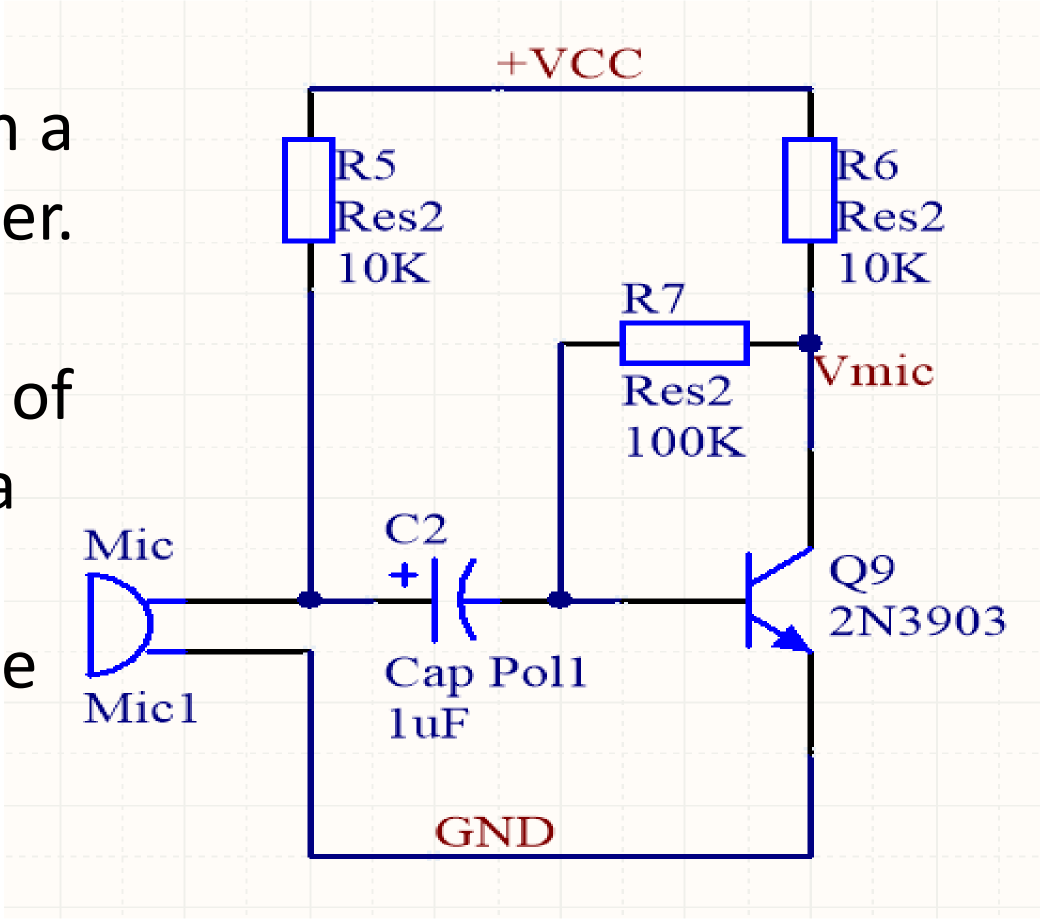
PCB fabrication process

- The complicated process of PCB fabrication can be summarized into this flow chart:



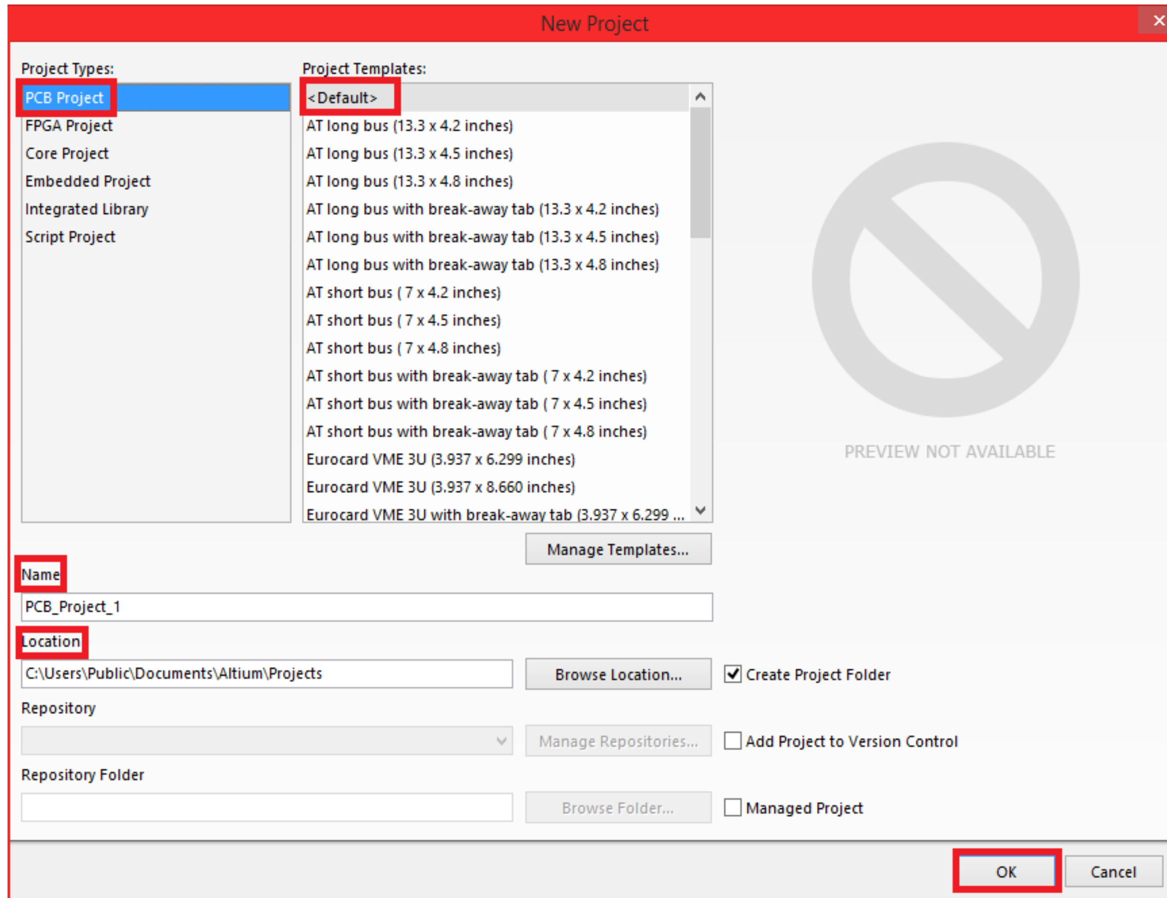
What's a Schematic?

- Schematic sheet is like what you draw to design a circuit on a piece of paper.
- You can see an example of a Schematic. (circuit of a CE amplifier with feedback for microphone input)



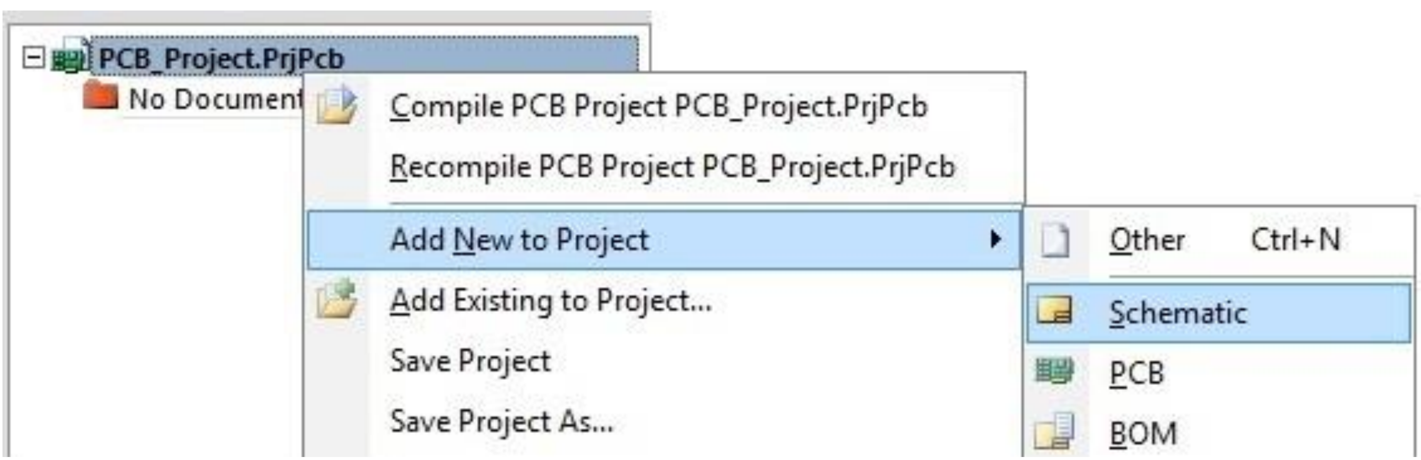
Creating PCB project

- Open the program. go to: **File\New\Project.**
- Select the options as you see, rename it and click **OK.**



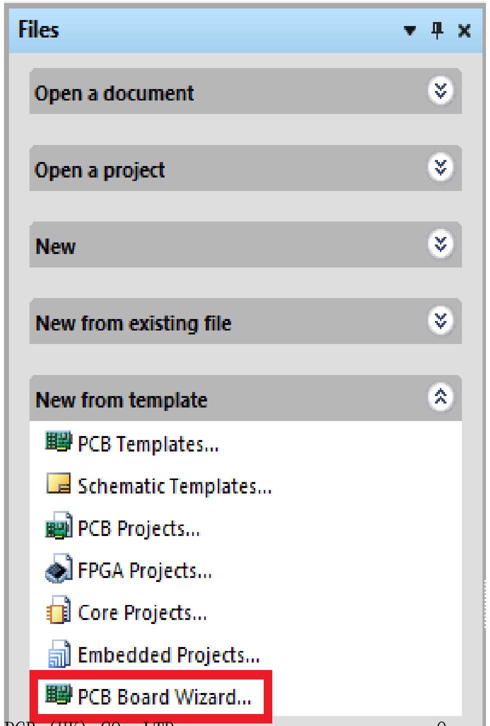
Creating Schematic

- Select **Projects** from the side toolbar.
- Right click on your PCB project.
- For creating a Schematic sheet, go to: **Add New to Project\Schematic**

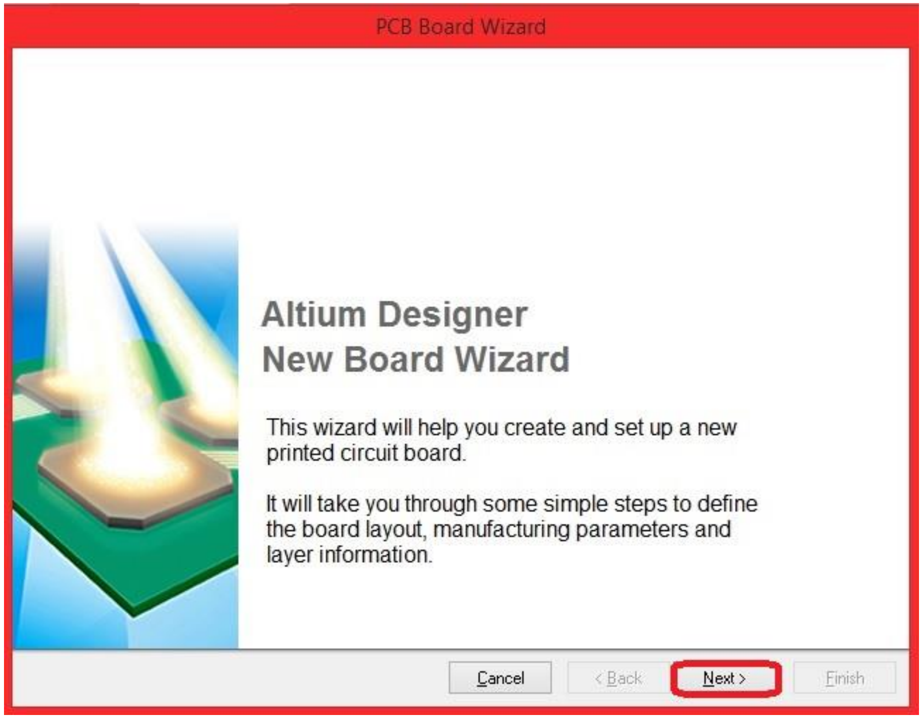


Creating PCB

- From the side toolbar (**Files**), select **PCB Board Wizard** from **New from template section.**

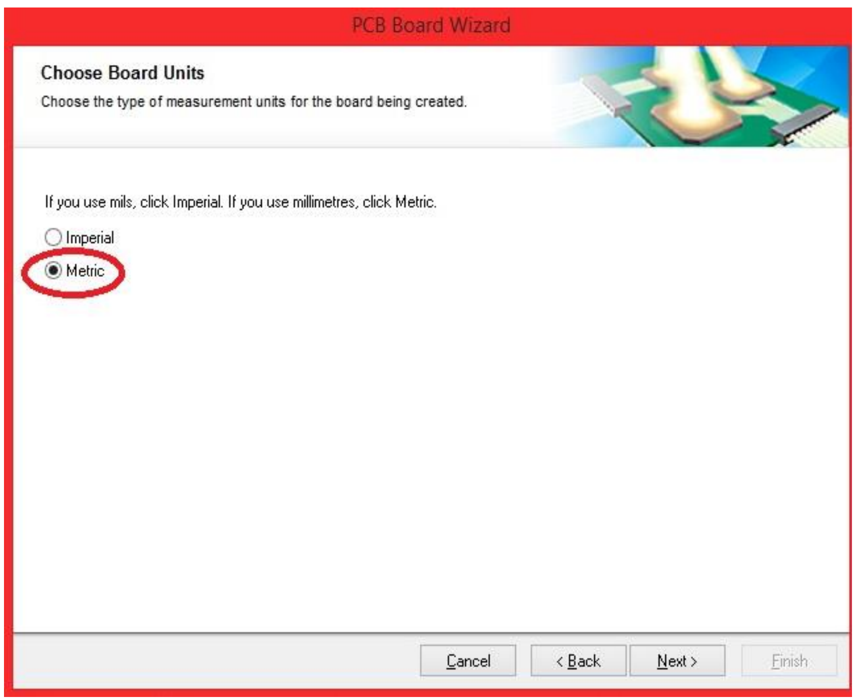


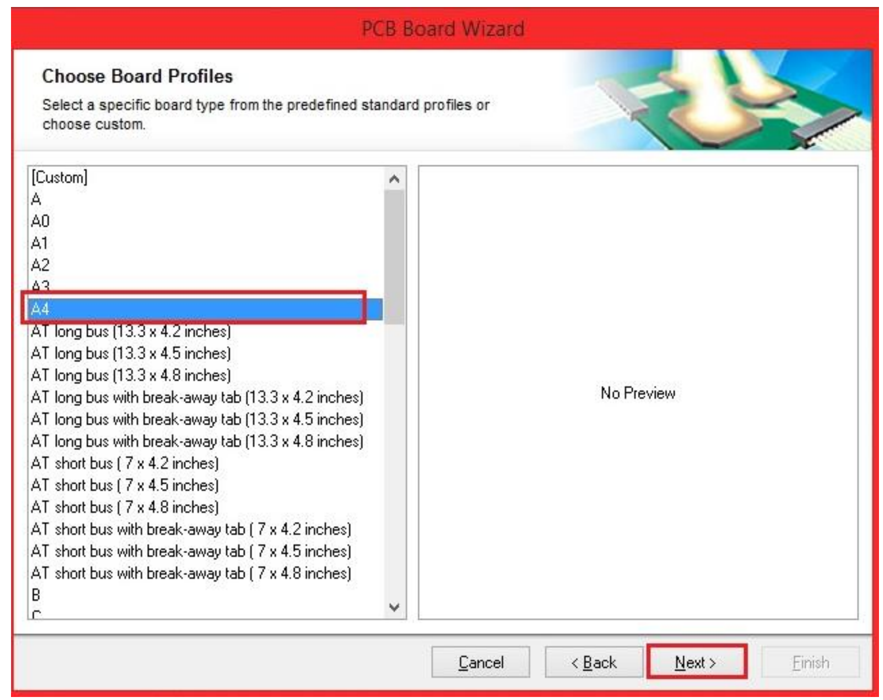
- Click **Next >**.



Creating PCB

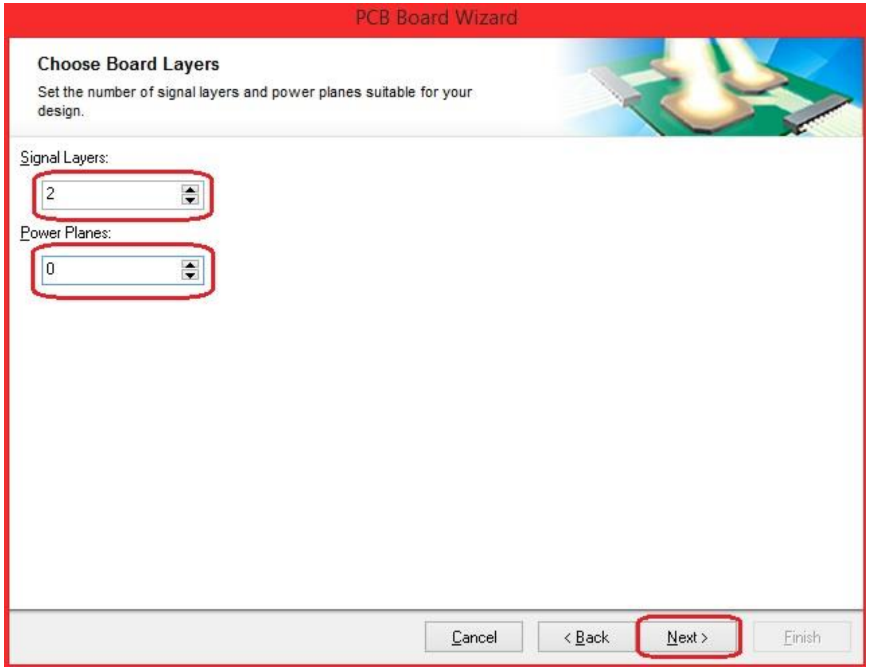
- Choose **Metric** from **Board Units**. Click **Next >**.
- Choose **A4** from **Board Profiles**. Click **Next >**.

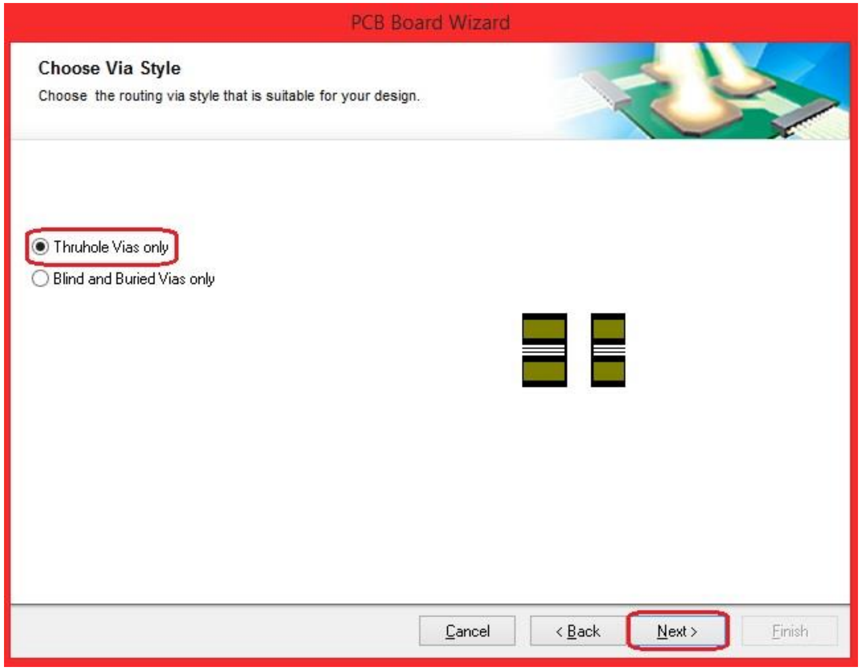




Creating PCB

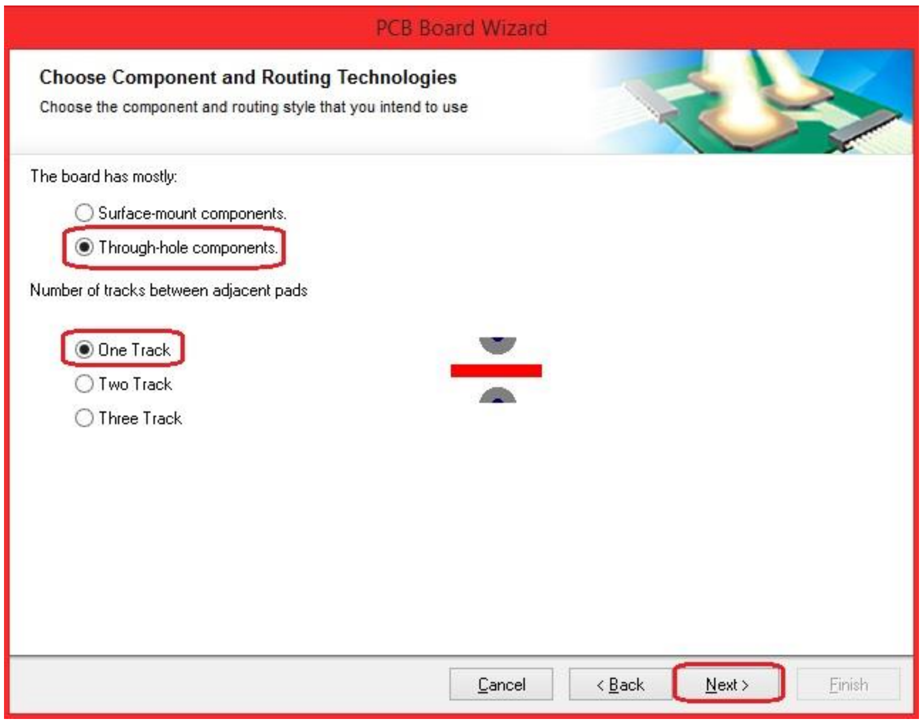
- Set **Signal Layers** to **2** and **Power Planes** to **0**. Click **Next >**.
- Choose **Thruhole Vias Only** from **Via Style**. Click **Next >**.

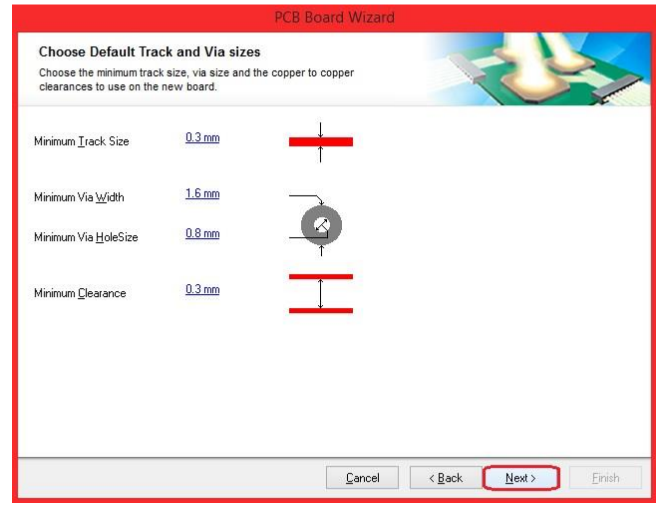


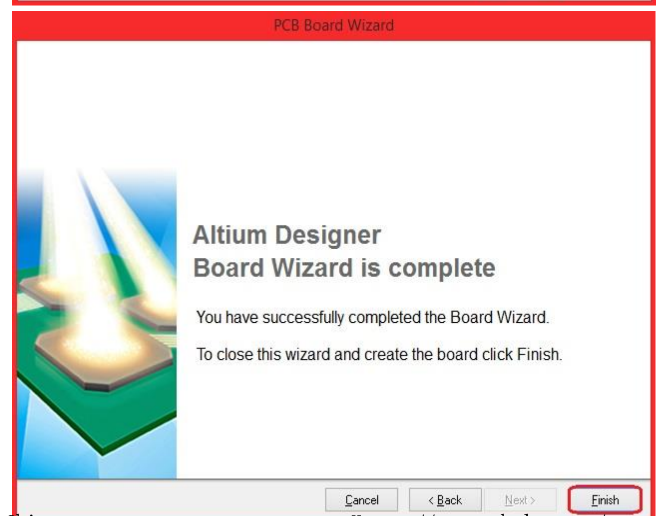


Creating PCB

- Choose **Through-hole components** and then **One Track**.







- Click **Next >** twice then **Finish**.

Adding Libraries

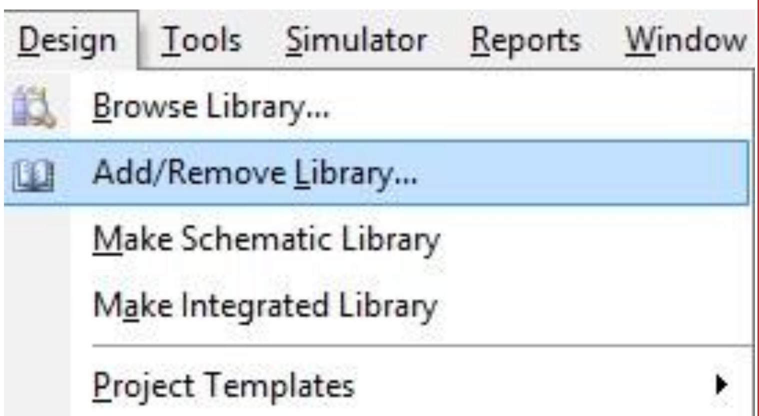
- After installing Altium Designer, you should install the libraries that include various devices needed in your Design (like resistors, capacitors, transistors, regulators, …).

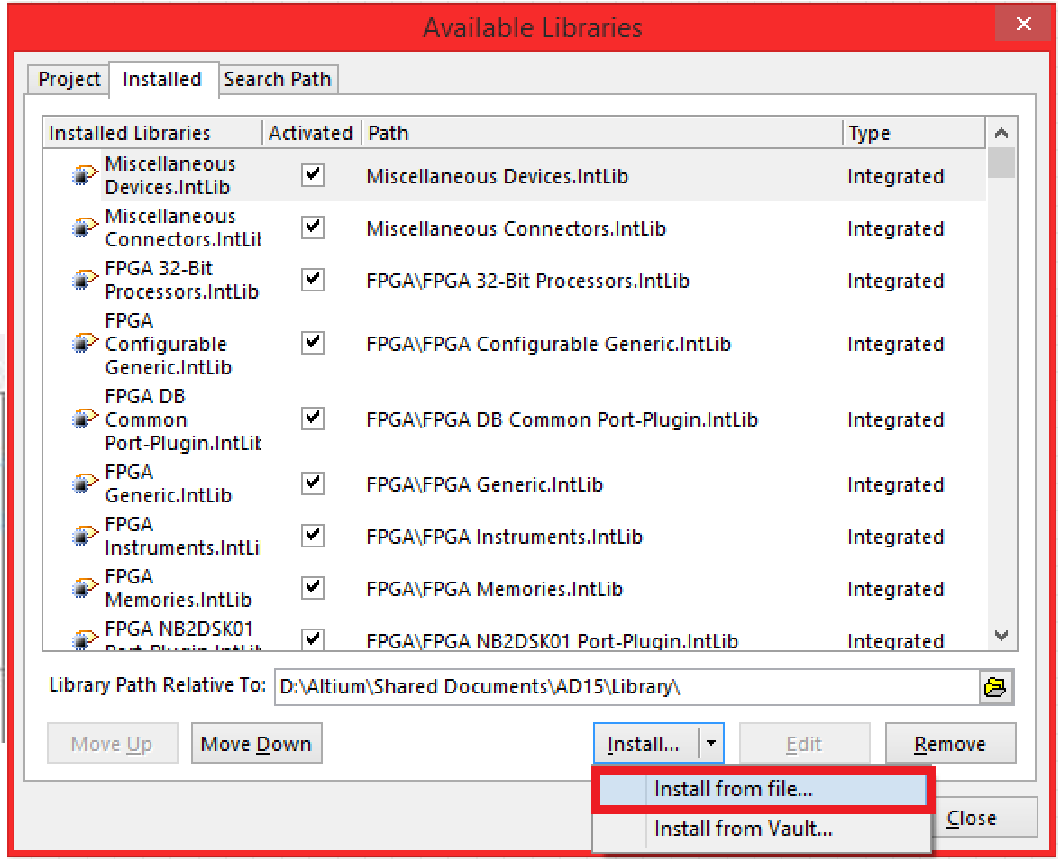
- Libraries can be found at:
  http://techdocs.altium.com/display/ADOH/Download+Libraries

- Usually libraries belong to different manufacturing companies and contain their production.

Adding Libraries

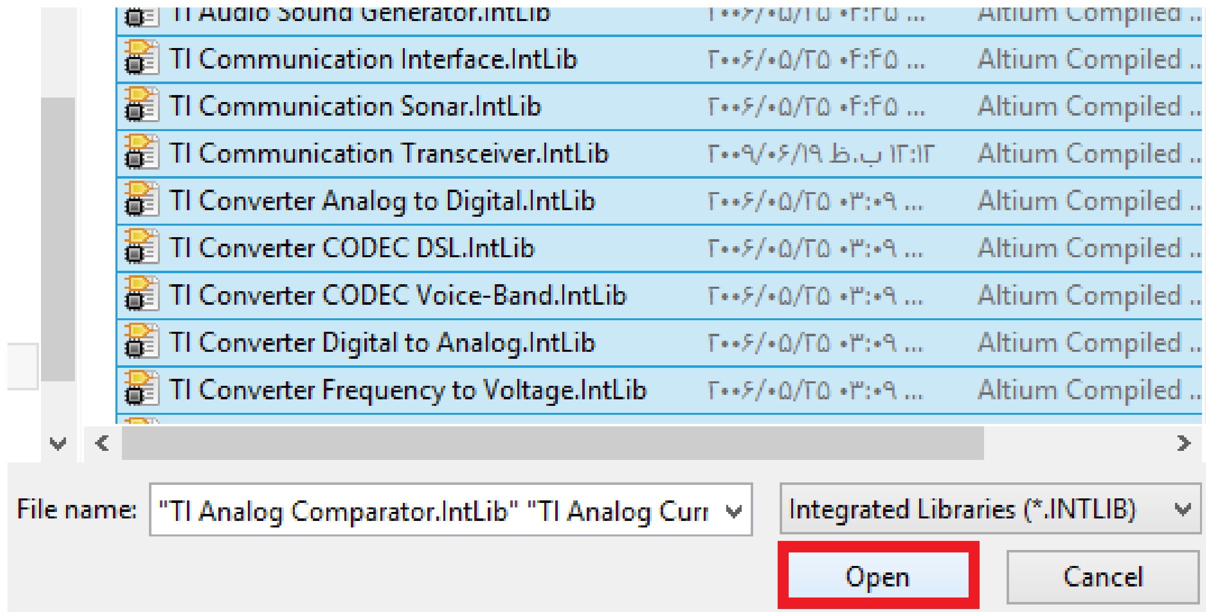
- Go to: **Design\ Add/Remove Library\ install\ install from file.**





Adding Libraries

- Select the downloaded libraries and click **Open.**



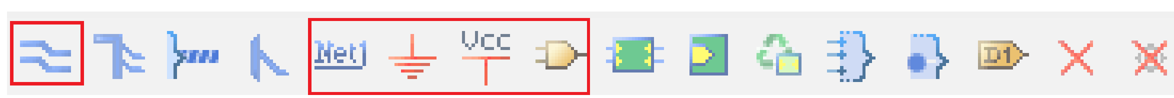
- Libraries should be installed by now.

Designing Schematics

- Before starting to work on Schematics, you should know which devices you are going to use and how they are connected.

- Choose the Schematic sheet which you created before.

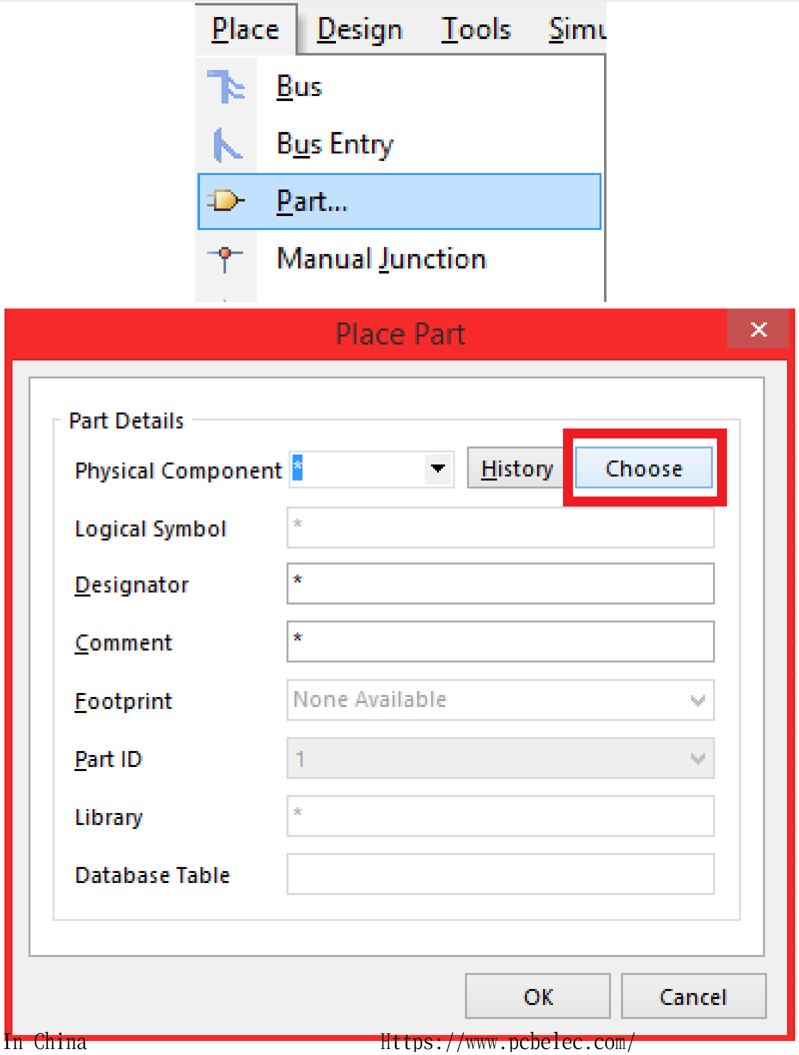
- At the top, there is a designing toolbar.



- You probably won't need most of them. In your project, you'll mostly need **Place Wire**, **Place Net Label**, **GND** and **VCC** symbol and **Place Part** (you'll also find these tools in **Design** at top).

Designing Schematics

- Let’s start by placing parts.
- For placing a part, you must have installed the libraries and also you must know which company makes that part.
- Go to: **Place\Part\Choose**.

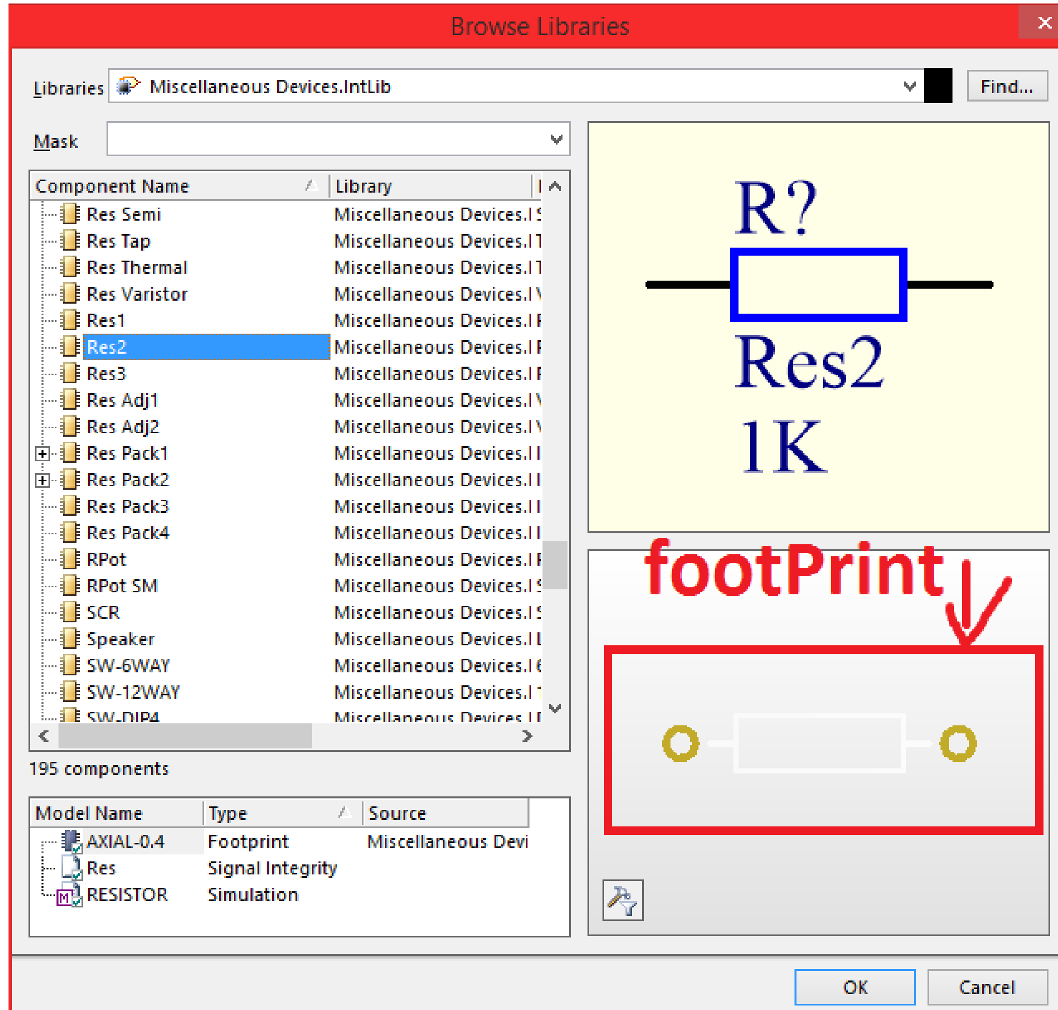


Designing Schematics

- Select the library you want to place part from and select the part.

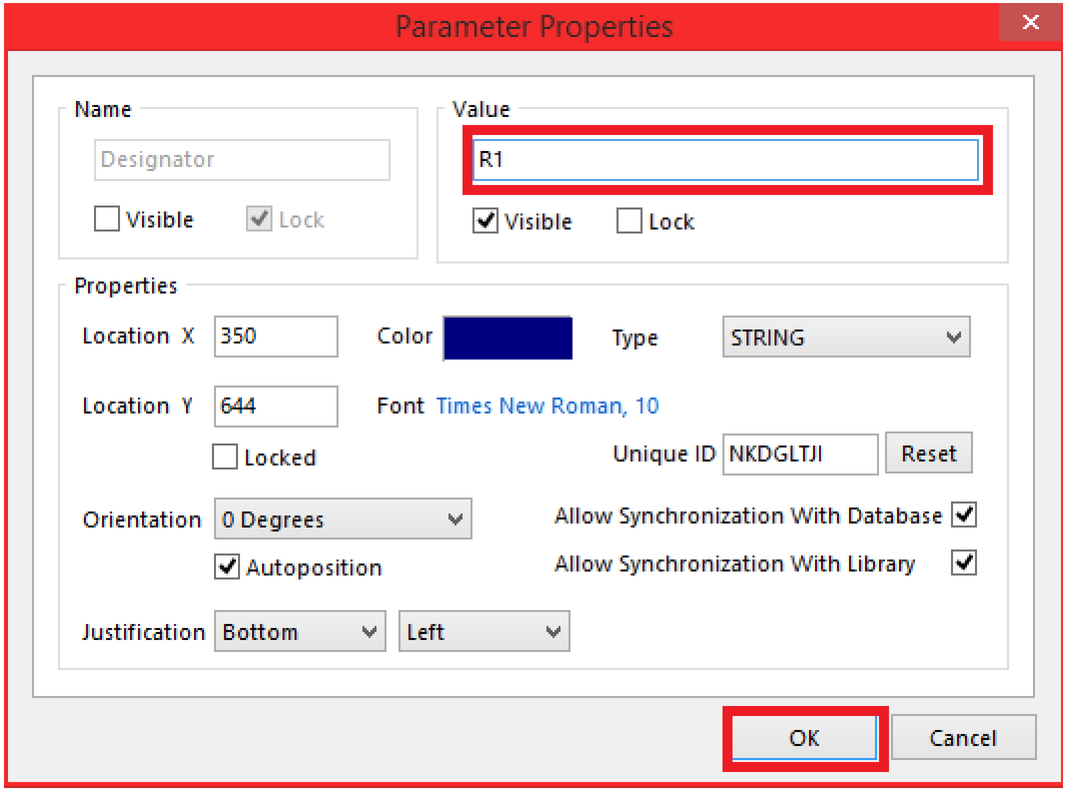
Important Note: be sure that selected parts, have valid **footprints**!

- Footprint defines the location of a device and therefore the location of pins and their holes' sizes on PCB.



Designing Schematics

- Put the device in Schematic.
- Double-click on the name.
- Rename the device in **Parameter properties**.
- **Tip**: Always rename the devices.
- **Note**: For rotating the device, press **space** key while dragging the device.



Designing Schematics

- You can change the value of a device by double-clicking on the value.

Tip: Check the datasheet of the part first!

- You can find basic electronic devices like Resistor, Capacitor and… in **Miscellaneous +Devices.IntLib** Library.

Designing Schematics

- Now that parts are placed, they need to be connected. For that you can use Wires or Net Labels.

- **Tip**: divide your circuit into different areas and use wire for connections inside areas and use Net Label for connections between areas.

- **Note**: Nets with same label are connected to each other.

- **Tip**: Try to use Net Label more often (but no so much) in your designs, because it increases your circuit's readability.

Designing Schematics

- After putting a **Net Label** on the port, you can edit the **Net Label**’s string by double clicking on it.

- **Note**: You can use 2-Pin Header for Power ports.

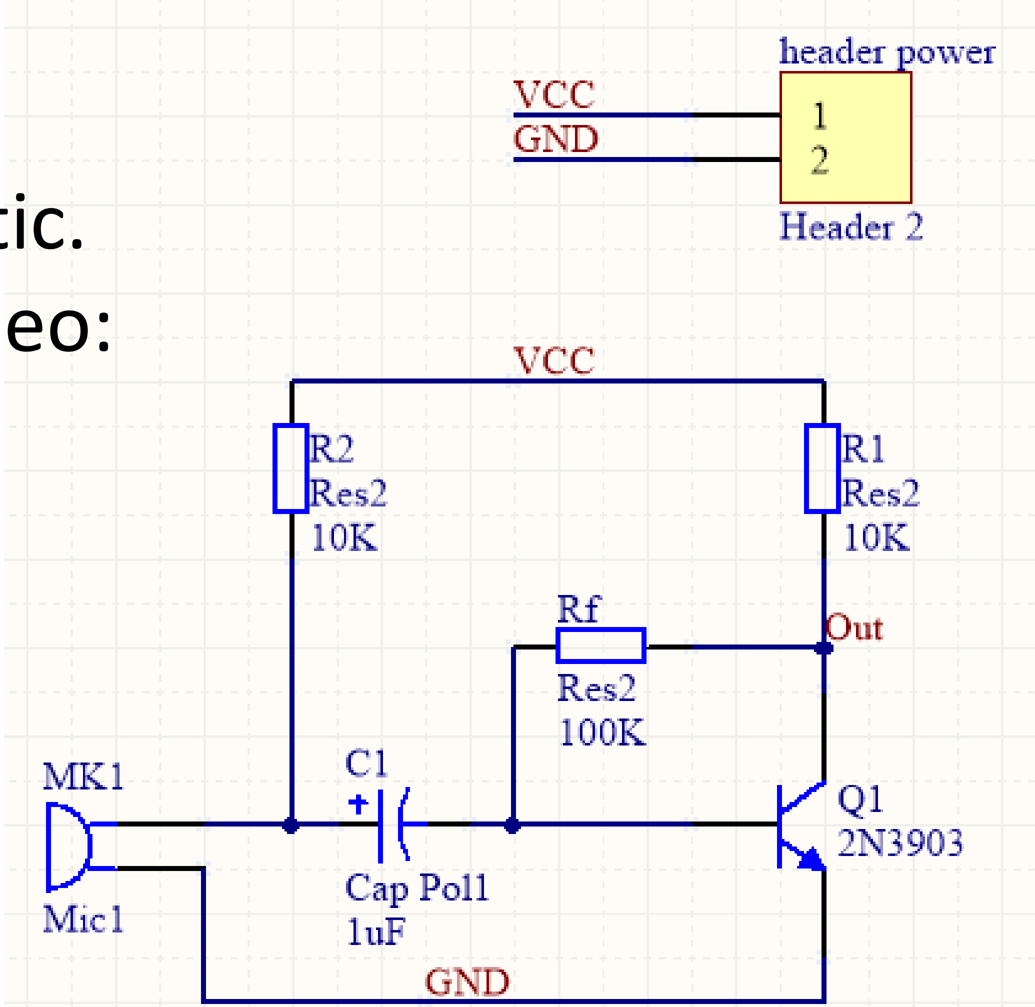
- **Tip**: Always use Net Label for GND and VCC nets. It will take much less space.

- **Tip**: Always make sure that wires and net labels are not in the air!

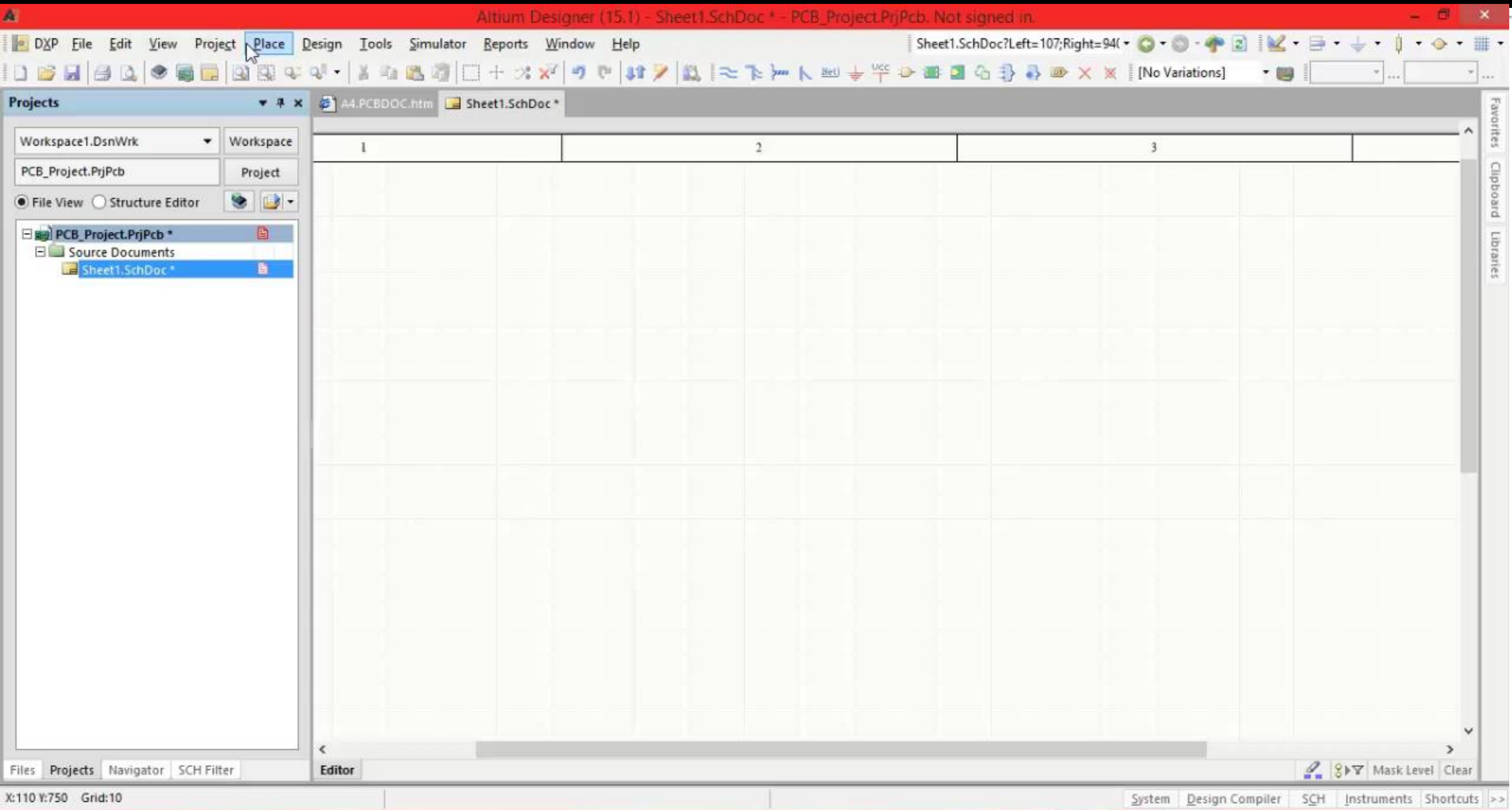
- **Note**: Make sure the devices don’t have same names.

Designing Schematics

- Now we're going to design this Schematic. Pay attention to Video:



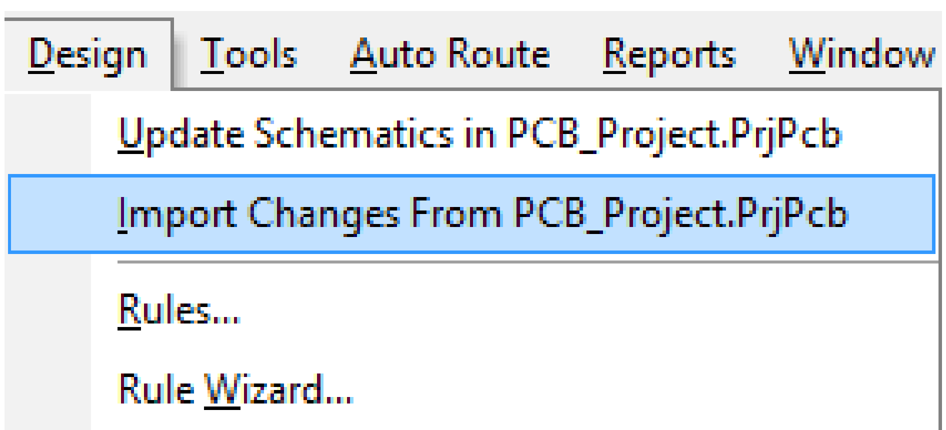
Designing Schematics



Designing PCB

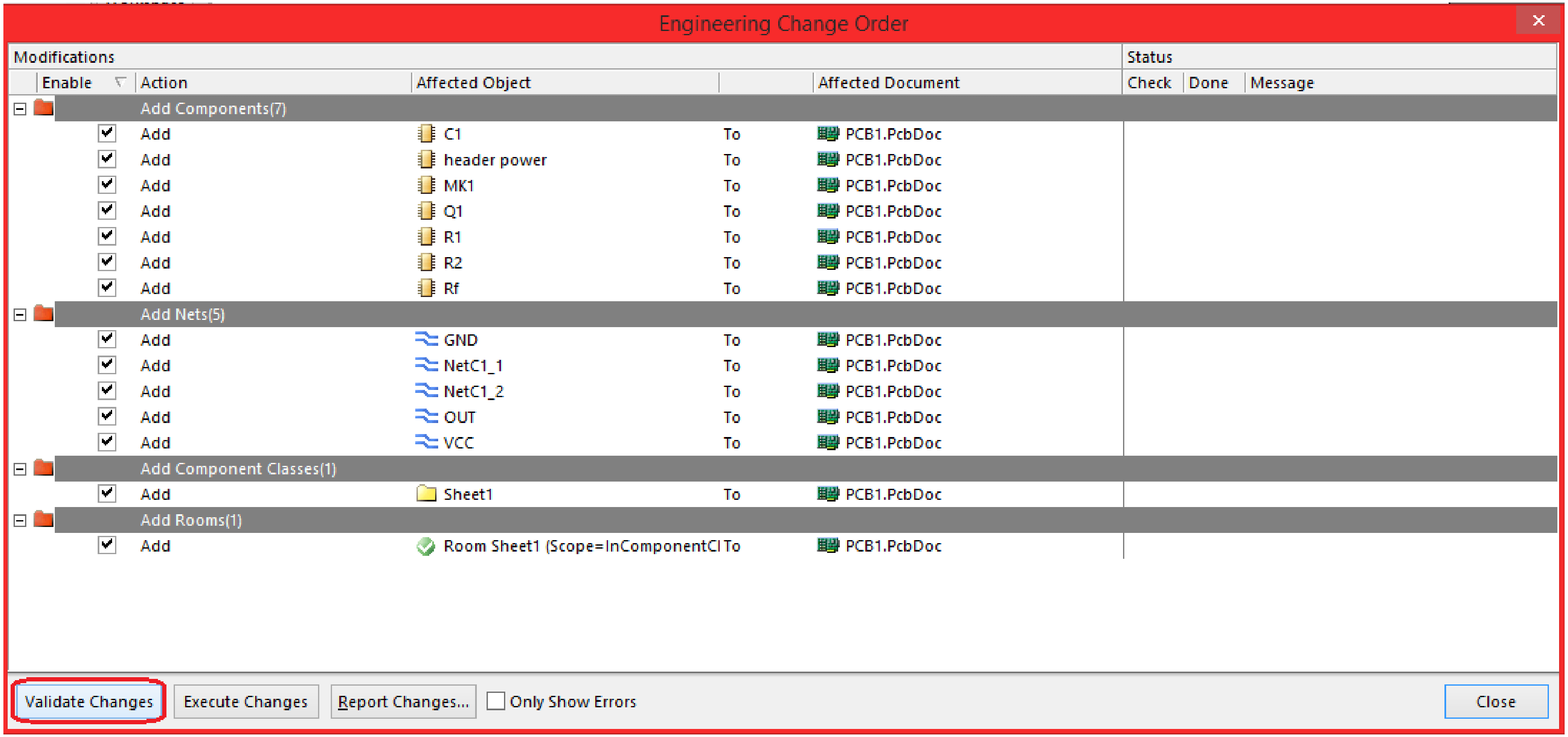
- After creating PCB file, both PCB and Schematic files must be added to the same project.

- Go to: **Design\ Import Changes from PCB_Project.PrjPcb**



Designing PCB

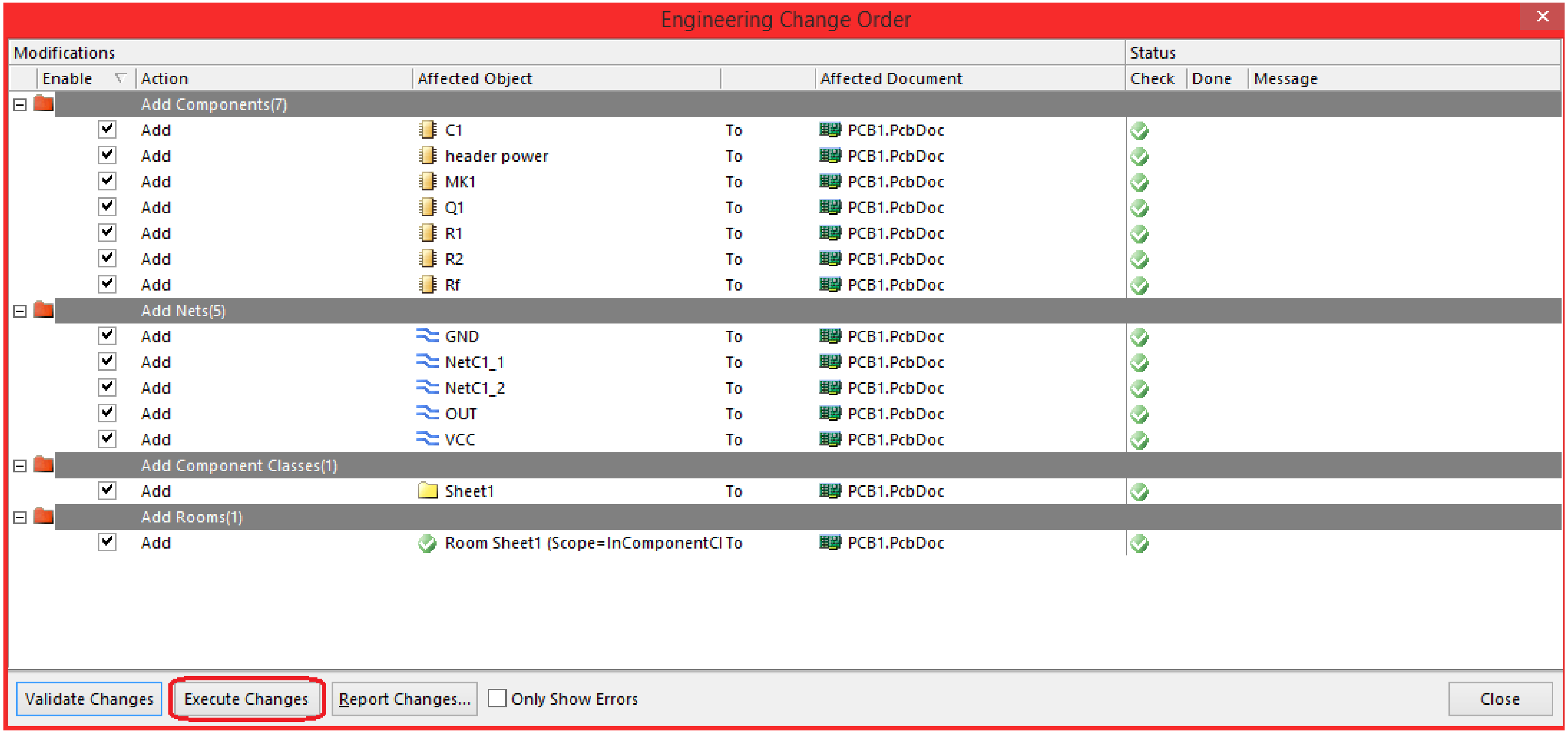
- Now click **Validate Changes:**



Enable	Action	Affected Object		Affected Document	Check	Done	Message
	Add Components(7)						
✔	Add	C1	To	PCB1.PcbDoc			
✔	Add	header power	To	PCB1.PcbDoc			
✔	Add	MK1	To	PCB1.PcbDoc			
✔	Add	Q1	To	PCB1.PcbDoc			
✔	Add	R1	To	PCB1.PcbDoc			
✔	Add	R2	To	PCB1.PcbDoc			
✔	Add	Rf	To	PCB1.PcbDoc			
	Add Nets(5)						
✔	Add	GND	To	PCB1.PcbDoc			
✔	Add	NetC1_1	To	PCB1.PcbDoc			
✔	Add	NetC1_2	To	PCB1.PcbDoc			
✔	Add	OUT	To	PCB1.PcbDoc			
✔	Add	VCC	To	PCB1.PcbDoc			
	Add Component Classes(1)						
✔	Add	Sheet1	To	PCB1.PcbDoc			
	Add Rooms(1)						
✔	Add	Room Sheet1 (Scope=InComponentCl	To	PCB1.PcbDoc			

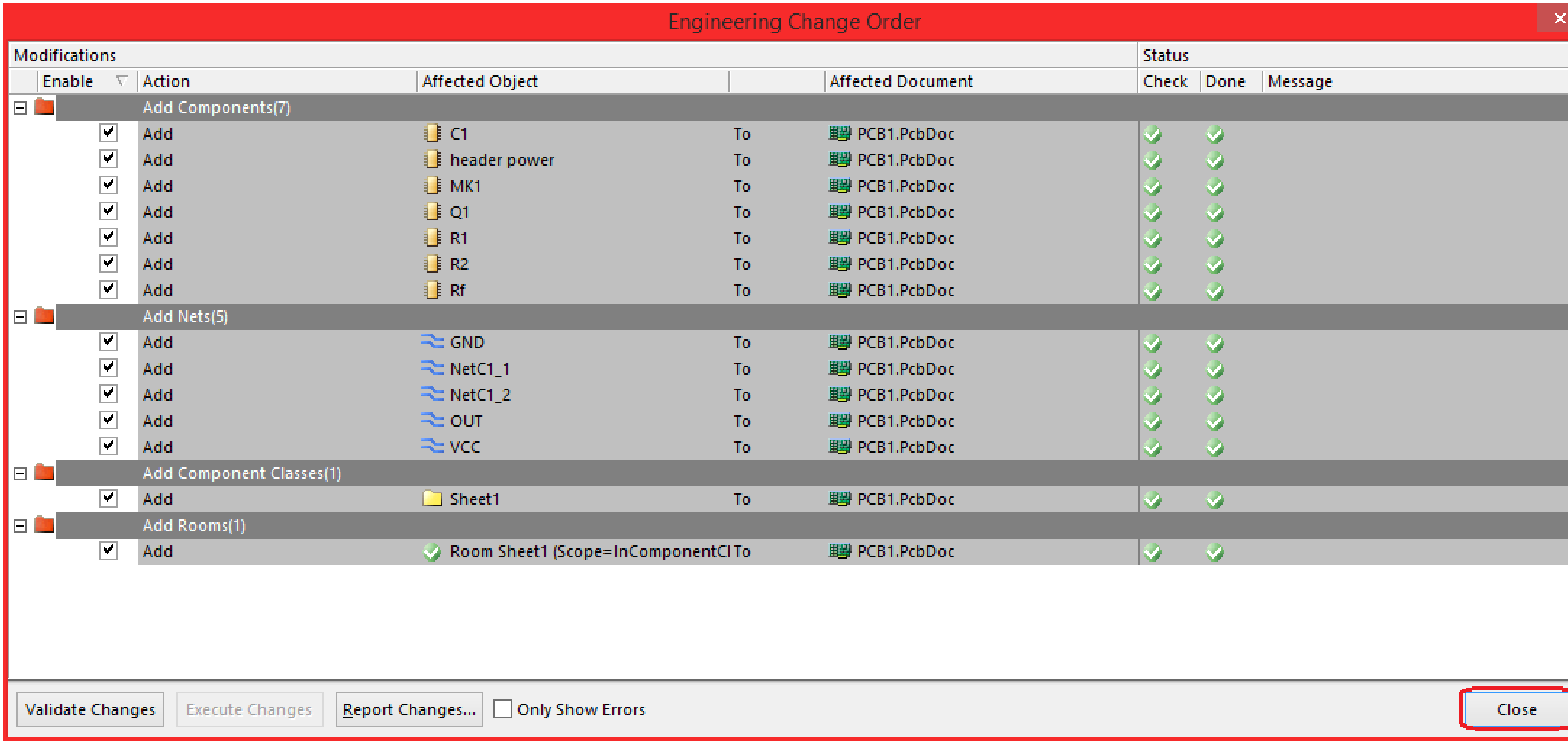
Designing PCB

- Now click **Execute Changes:**



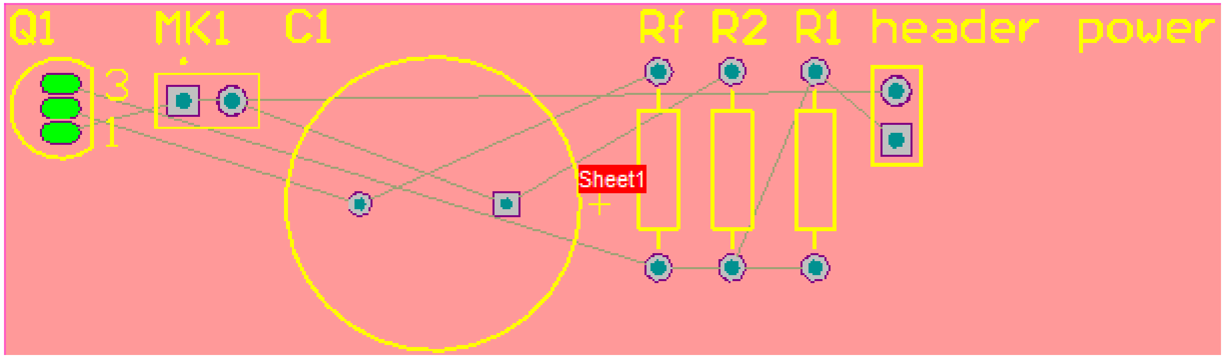
Designing PCB

- Now click **Close**:



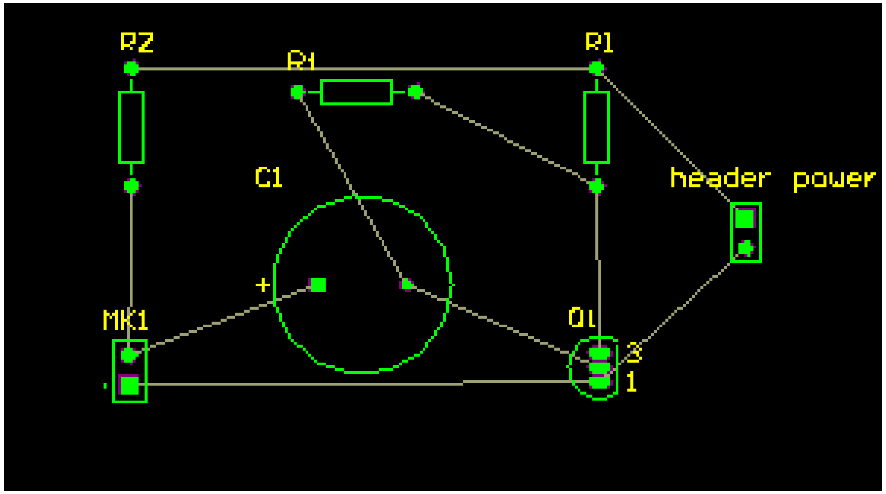
Designing PCB

- **Note**: Make sure that there is no error after the update.
- After importing, you can find the added parts inside the pink area, outside of PCB area (black area).



Designing PCB

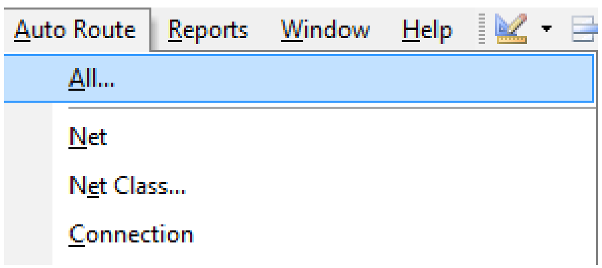
- Put the devices in appropriate locations inside the black area.



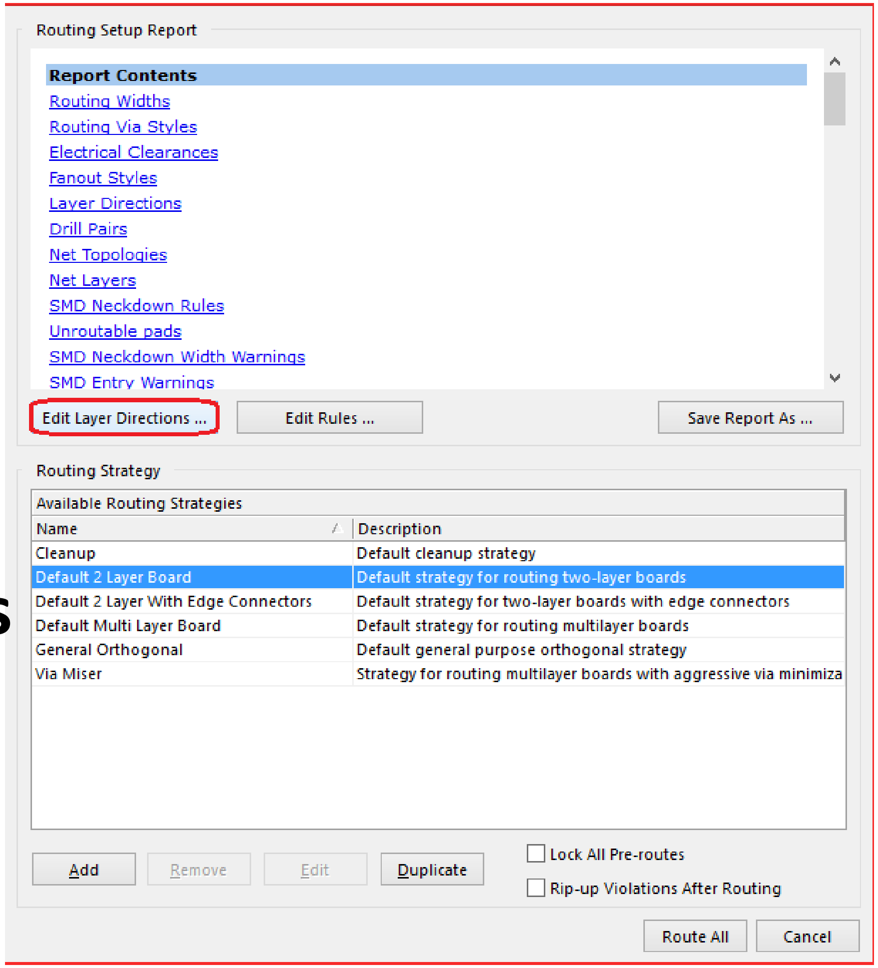
- Now you should connect devices with **Routes**.
- **Routes** are conductive tracks connecting the holes.
- The best way is using **Auto-Route**.

Designing PCB

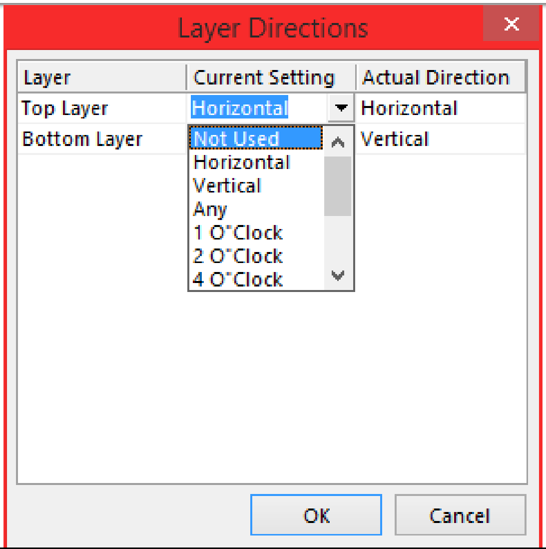
- Go to: **Auto-Route\ All…**



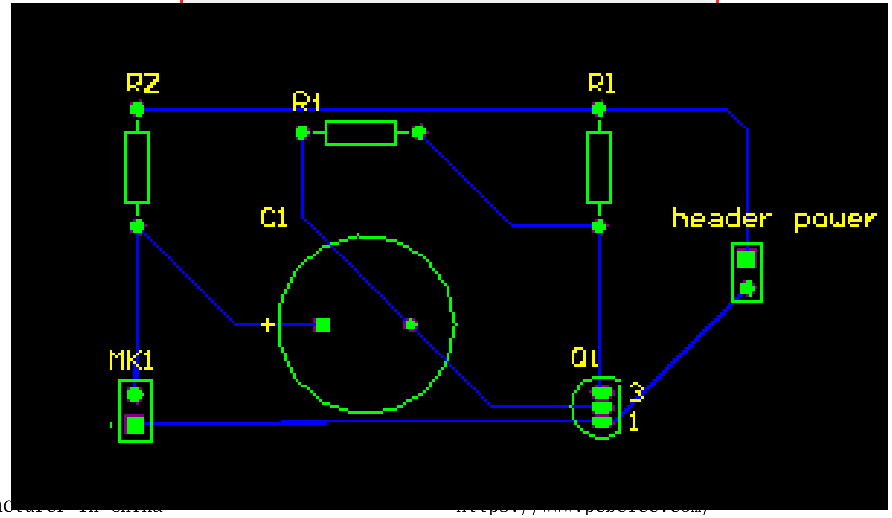
- Click **Edit Layer Directories**



Designing PCB



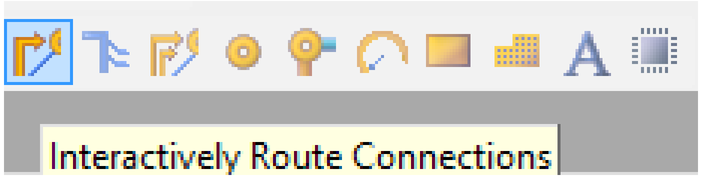
- Click on **Current Setting** of **Top layer** and set its value to **Not Used**. Click **OK**.
- Click **Route All**.
- Your PCB should look like this:



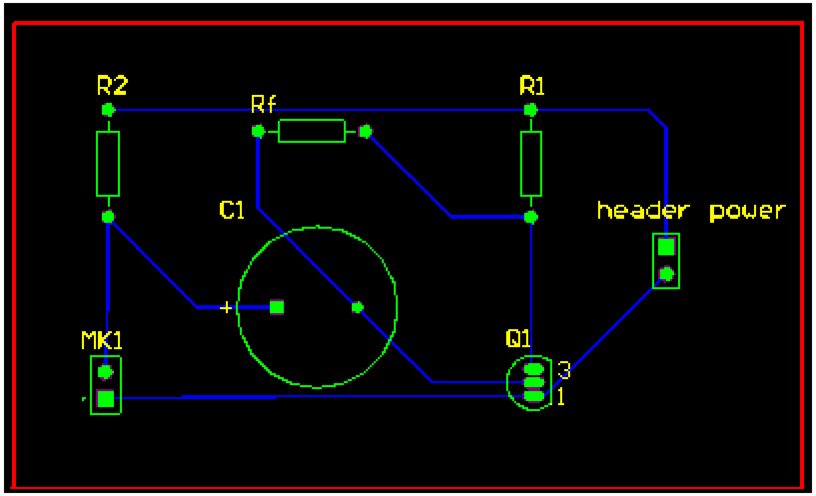
Note: you can also manually modify routes by dragging them and using **Interactively Route Connections** .

Designing PCB

- Now we're going to set the size of PCB.
- Select **Interactively Route Connections** from top toolbar.



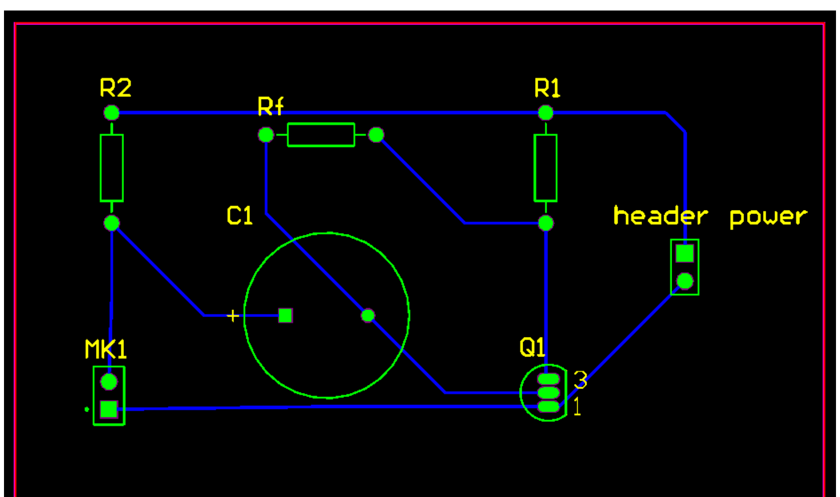
- Surround the devices by routes and make a rectangle like this: (red rectangle)

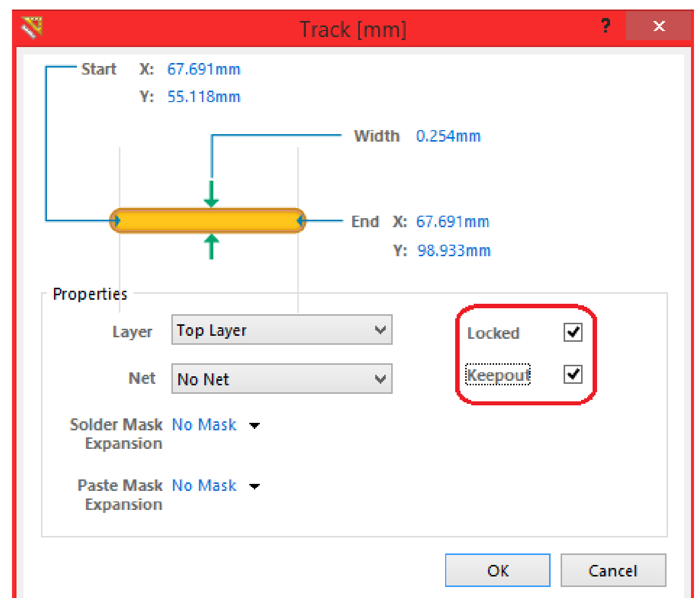


Designing PCB

- Double-click on routes and check **Locked** and **Keepout.**

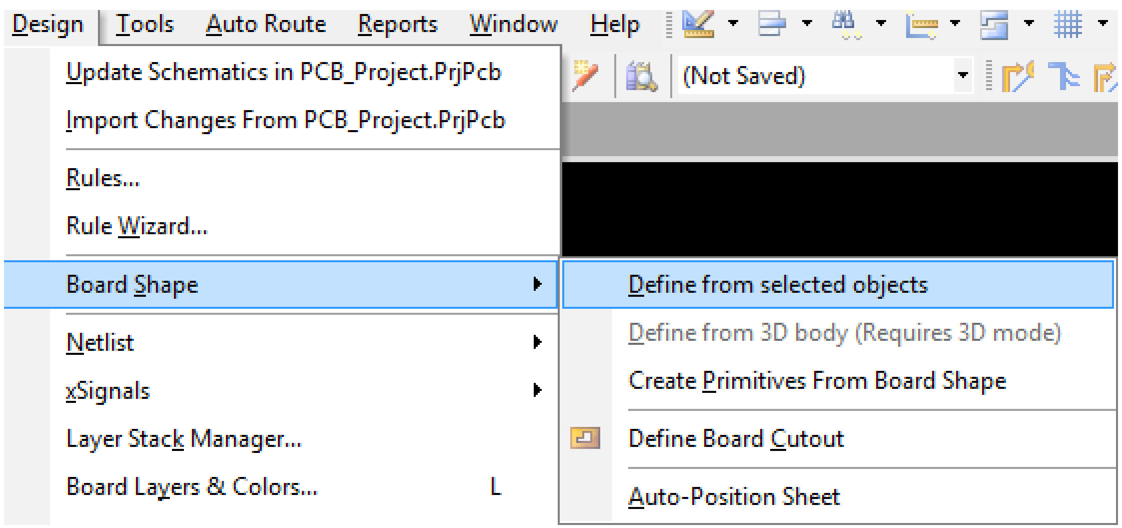
- Your PCB should look like this: (surrounded by pink routes)

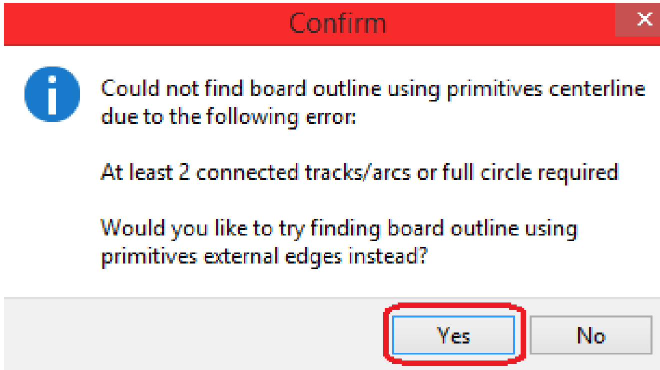




Designing PCB

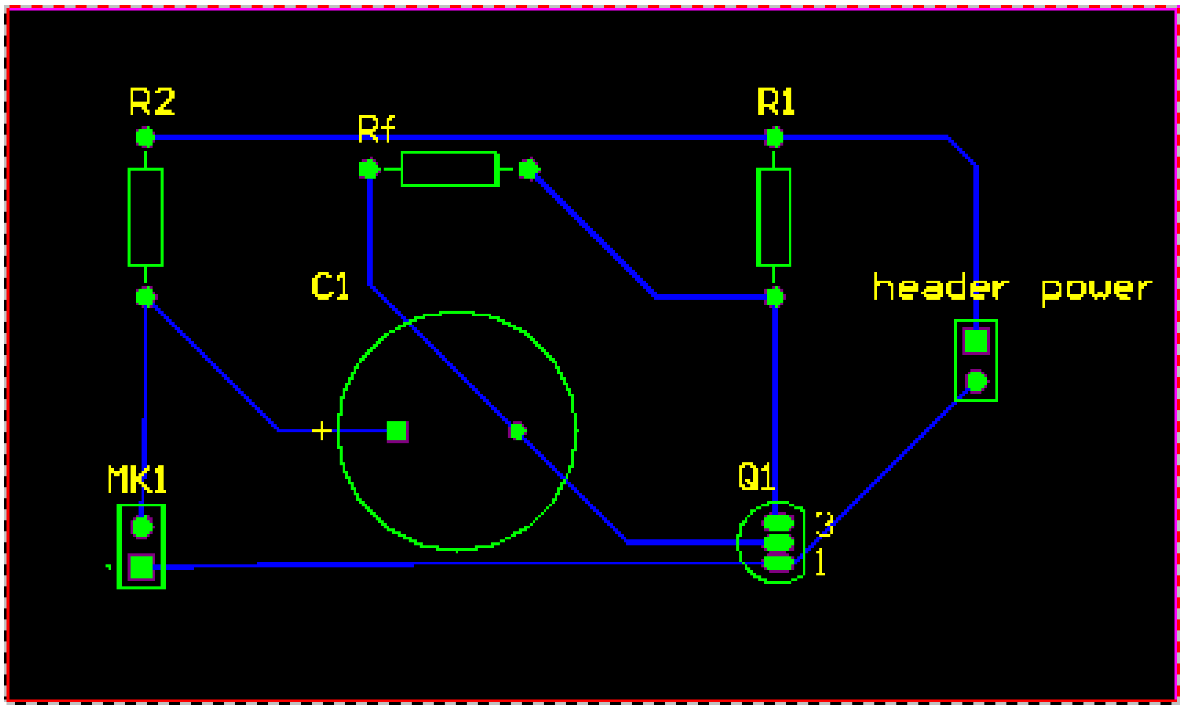
- **Note**: Pink routes, are called Keep-out routes.
- Select all devices and routes (including Keepouts).
- Go to: **Design\ Board Shape\ Define from selected objects**.
- Click **Yes**.





Designing PCB

- Now Your PCB should only have black areas only inside Keepouts, like this:



Common errors and tips

- Remember! any problem you run into in Altium can be found in internet. Look for answers in different forums.
- While opening a PCB file, you may get "**Please wait a moment**" crash. I solved this problem by running the program with **Integrated Graphics** of PC.
- Don't make your board too small. you'll have problem soldering the elements. Also don't put devices too close to each other.

Common errors and tips

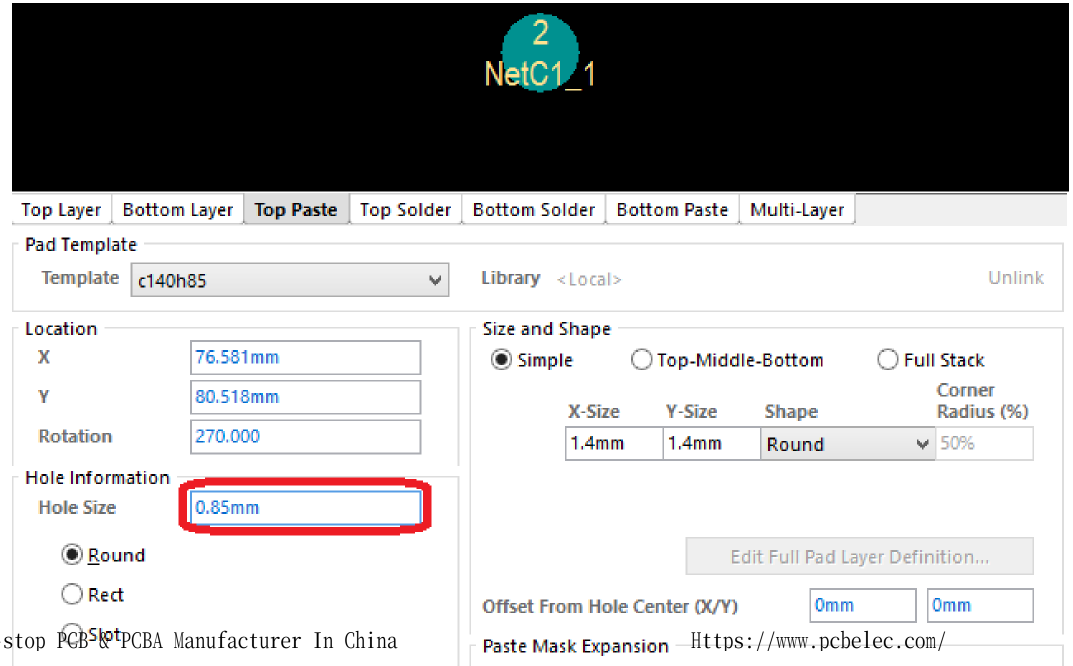
- Errors while importing from Schematic to PCB are mostly caused by these:

1. Some of your devices don’t have valid footprints.
2. You have some nets on air!

- You may not find your desired device in the libraries. In this case you should replace by a device which has the same footprint, so make sure size of the Pin-holes are the same.

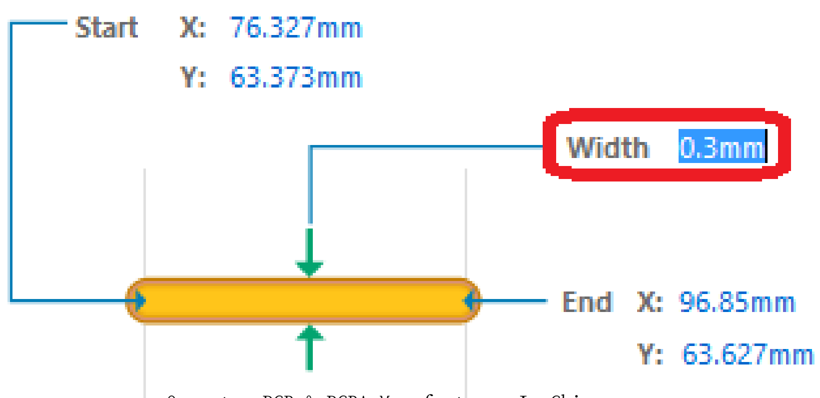
Common errors and tips

- **Important Tip:** Make sure that routes on PCB, are not too close to holes.
- If necessary, you can change the size of the holes, by double clicking on the hole and editing the **Hole Size.**
- For new size holes, add 0.1 mm tolerance to diameter of the device's pin.
- Check web for answers!



Common errors and tips

- You can change the width of a route too.
- This comes handy for routes that pass huge current. If you don’t increase its width, then that route will just melt!
- To do so, just double-click on the route and change the width:



Guide to online ordering a PCB

- Try your best to make your PCB, a One layer PCB, so it will cost less. use only bottom routing (the blue one).

- It’s not necessary to spend a lot of money on PCB, so try to choose cheaper options while ordering a PCB.

- Calculate width and height of the board by checking the coordinates (located at left-top of workspace) of board corners.

Guide to online ordering a PCB

- Recommended options for simple PCB:

نوع مدار چاپی:	یک لایه
نوع فیبر:	فایبر گلاس
ضخامت مس:	35میکرون
پوشش نهایی:	آبکاری قلع و سرب(فقط برای بردهای بدون چاپ محافظ)
چاپ محافظ:	ندارد
رنگ چاپ محافظ:	ندارد
چاپ قطعات:	ندارد
رنگ چاپ قطعات:	ندارد
ضخامت فیبرمدار چاپی(میلیمتر):	1.6
طول برد (میلیمتر):	80
عرض برد (میلیمتر):	100
تعداد:	1
برش نهایی:	دستی

different for PCBs

References

- https://www.wikipedia.org/
- https://learn.sparkfun.com/tutorials/pcb-basics
- http://www.altium.com/