• 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.
• 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 the 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**.
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.
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).
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.
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.
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.
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
Now click **Validate Changes**: 

![Engineering Change Order](image)

<table>
<thead>
<tr>
<th>Modifications</th>
<th>Action</th>
<th>Affected Object</th>
<th>Affected Document</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Components(7)</td>
<td>Add</td>
<td>C1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>header power</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>MK1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>Q1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>R1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>R2</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>Rf</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td>Add Nets(5)</td>
<td>Add</td>
<td>~GND</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>~NetC_1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>~NetC_2</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>~OUT</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td></td>
<td>Add</td>
<td>~VCC</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td>Add Component Classes(1)</td>
<td>Add</td>
<td>Sheet1</td>
<td>PCB1.PcbDoc</td>
</tr>
<tr>
<td>Add Rooms(1)</td>
<td>Add</td>
<td>Room Sheet1</td>
<td>PCB1.PcbDoc</td>
</tr>
</tbody>
</table>
Designing PCB

- Now click **Execute Changes**: 

![Engineering Change Order](image)

<table>
<thead>
<tr>
<th>Modifications</th>
<th>Action</th>
<th>Affected Object</th>
<th>Affected Document</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Components(7)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>C1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>header power</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>MK1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Q1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>R1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>R2</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Rf</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add Nets(5)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>GND</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Net1_1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Net1_2</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>OUT</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>VCC</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add Component Classes(1)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Sheet1</td>
<td></td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
<tr>
<td>Add Rooms(1)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add</td>
<td>Room Sheet1</td>
<td>(Scope=InComponentCITo)</td>
<td>PCB1.PcbDoc</td>
<td></td>
</tr>
</tbody>
</table>
Now click **Close**:
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:

 lesbienne: you can also manually modify routes by dragging them and using **Interactively Route Connections**.
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)
Double-click on routes and check **Locked** and **Keepout**.

Your PCB should look like this: (surrounded by pink routes)
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.
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.
**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!
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:
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:

  *(Persian text)*

  *(Arabic text)*

  *(Chinese text)*

  *(Different for PCBs)*
References

- https://www.wikipedia.org/
- https://learn.sparkfun.com/tutorials/pcb-basics
- http://www.altium.com/
If you have any questions, you can contact me by email with title of “Altium”.

sharif.pelec@gmail.com
Thank you!