So, you took EE210 and learned all about how to capture schematics and perform simulations in Multisim, but now you want to make a PCB for your new circuit design. The 30-minute run through of Ultiboard left you scratching your head and you want to know how to make a PCB for REAL. This guide will show you the complete process for designing a printed circuit board, which includes schematic capture, simulation, and layout. Depending on the circuit you are making a PCB for, this process should take 3-5 hours to complete.

What is a PCB?
A printed circuit board, or PCB, is an assembly that mechanically supports electronic components while electrically connecting them through conductive traces. PCBs are typically constructed of a fiberglass substrate called FR4 that is covered in copper on both sides. The substrate acts as the mechanical core of the PCB that holds everything in place, while the copper layers are used to form traces, which are the copper connections that electrically connect each component.

![Cross Sectional View of PCB](image)

Why make a PCB?
You have constructed many circuits on a breadboard as part of your EE210 lab and you may wonder: Why go through all the trouble of making a PCB? There are several reasons to move a circuit from a breadboard to a PCB. PCBs are more mechanically stable, offer better performance if well designed, and can be mass produced. For these reasons, PCB design is often an essential part of designing electronics.

What you will need
- Computer with Multisim and Ultiboard
- Printer

Both of these materials can be found in Electrical Engineering Department computer labs.
Step One: Schematic Capture
In order to build a printed circuit board, the software first needs to understand what your circuit is. This is done through schematic capture, which you have been taught how to do as part of EE210.

*NOTE:* It is essential that your schematic is correct. Double and triple check your design to ensure accuracy. Being thorough in this step will save you time and money.

![Schematic Diagram](image)

Step Two: Simulation
To ensure your circuit works as intended, a circuit simulation should be completed whenever possible. Again, you will have learned how to do this as part of the EE210 curriculum. Test your circuit under as many conditions as you can to simulate real life. Not all circuits can be simulated in Multisim (microcontrollers, some ICs, etc), but often portions of your design can be tested individually.

![Simulation Image](image)

*Note:* It may be tempting to skip the first two steps of this guide and move straight to designing your printed circuit board, but these steps are crucial to ensuring that your PCB will be designed accurately and correctly.
Step 3: Layout

Once you have entered your design into Multisim and tested it, it is time to use Ultiboard to begin PCB layout. This document assumes you have learned the basics of Ultiboard in EE210; however, a basic diagram of Ultiboard is included below.

1. In Multisim select: Transfer -> Transfer to Ultiboard.
   a. This will transfer your design information to Ultiboard. You should see all of your components lined up next to each other next to a PCB outline.
2. Now in Ultiboard, use the design toolbox to make the board outline viewable and selectable.
3. Select the board outline and set it to your desired form factor.
   a. If you do not have a specific form factor target, make the board as small as possible.
4. Use the Selection toolbox to ensure components are selectable.
5. Double click the Copper Top layer in the Design Toolbox.
6. Make the ratsnest visible by checking the box next to ratsnest in the Design Toolbox

Note: Read the entire instruction set before beginning this step. Doing so will save you time and effort.

The ratsnest is a point-to-point view of the electrical connections in your circuit.
7. Arrange components within the board outline, using the ratsnest to guide placement of each component to minimize overlap.
   a. Allocate extra space for high power consumption components.
   b. Separate digital and analog components whenever possible.
   c. Place bypass capacitors as close to sources as possible.

Here is an example layout of the circuit from parts 1 and 2. The holes for each component are shown as dark blue circles. The light blue graphics show which components go where. Finally, the yellow lines are the ratsnest.

8. Use the “line” tool in the routing toolbox to electrically connect each component.
   a. Control which layer the trace is on using the design toolbox.
   b. Ensure that no traces on the same layer overlap.
   c. Never create 90 degree angles with traces. The “line” tool will automatically help you to make 45 degree angles wherever possible.
   d. Minimize the length of your traces whenever possible.

Here is the same layout with traces added. Green lines indicate top-layer connections while red lines are on the bottom layer. Can you spot any improvements that could be made to the board outline, component placement, or routing?

Note: Ultiboard is equipped with an autorouter function that can place traces for you; however, use of an autorouter is not recommended. The autorouter has no understanding and will create awful designs that will take more time to fix than making the traces yourself.

9. Run a DRC check by selecting Design -> DRC Check.
10. Run a netlist connectivity check by selecting Design -> Connectivity Check.
11. Once you have a design that passes DRC and connectivity checks, you have completed your PCB design!

DRC stands for Design Rules Check, which verifies your design can be physically realized.

A connectivity check ensures that your PCB electrically matches your original schematic.
Conclusion
Now that you have completed your schematic capture, simulation, and layout; you are ready to send your design to a PCB manufacturer. Most manufacturers take about a week to make your PCB and send it to you. Once you have your PCB in hand, you can solder your components on and your circuit will be complete!

Appendix
Picture on first page taken from:


Answer to question in step 3.8:

- Board outline: Remove extra space on left side
- Component placement: Components can be placed closer together.
- Routing: Eliminate right angles and unnecessary turns in the traces.