



# Electrical and Computer Engineering

**ECE315**

**Lab 3:**

**Component Placement**

## 1. Lab 3 Overview

In Lab 2, you completed the schematics for your embedded system. In this lab, you will transfer the associated PCB footprints to a PcbDoc file and place components on the printed circuit board. Component placement is one of the most critical steps in designing a printed circuit board and can have a significant impact on the amount of work required to complete the routing of signals.

So what determines where components are placed? Components are sometimes placed in specific locations to meet the mechanical requirements of an enclosure. There may be a protrusion in the enclosure that prevents you from placing tall components in certain locations of the board. There may also be situations where the functionality of a sub circuit will dictate where components will be placed. Digital circuits are very good at producing noise, so we normally try to isolate analog components to avoid digital noise from bleeding into analog signals. For your design, the components placement is mostly driven by the “user experience”. All of the components that you will place will be placed on the bottom of the board.

## 2. Pre-Lab

- A. Open the Altium Designer Project that you completed in Lab 2 by clicking on the **ECE315.PrjPcb** PCB Project File.
- B. Open MotionW.PCBDoc. The shape and size of your printed circuit board has been defined for you. You **SHOULD NOT** modify the shape and size of your printed circuit board! You will also notice that the LEDs and associated bypass capacitors have been placed for you. These components have been locked in place, so **DO NOT MOVE** any of the LEDs/capacitors that have been placed for you.

You can watch the following videos that describe how to define and resize a printed circuit board for your reference.

[PCB Board Wizard](#)  
[Redefine Board Shape](#)

- C. Compile your design and transfer components to MotionW.PcbDoc. Be sure to fix any errors that you encounter after you have compiled the design. When all errors have been fixed, transfer the footprints to the PcbDoc.

You can watch the following video that describes the process of compiling your schematics and transferring the components.

#### [Compile Schematics and Transfer Components](#)

- D. Complete the placement of the components found on 03\_MCU.SchDoc and 08\_USB\_To\_UART.SchDoc. You DO NOT have to route any signals, but try to arrange components in a way that reduces the number of overlapping signals.

When placing parts, be sure to adhere to the following constraints:

- Try to keep all components from a schematic page close together.
- All components you are placing MUST be placed **on the bottom side** of the board.
- Each voltage pin (VCC/VDD) of the MCU should have a bypass capacitor within 200 mils of the pin.
- The microUSB connector should be placed as close to the edge of the board as possible. This will ensure that the microUSB cable used to connect to the board can connect correctly.



The following video demonstrates some basics of parts placement.

#### [PCB Parts Placement](#)

- E. Add text strings on the bottom overlay near the test points to indicate which net each test point is connected to. Labeling the test points is a convenience that will speed up the debugging process.

### 3. Pre Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab3_Altium.zip	Altium Project ZIP file. All schematic pages should be completed and parts for MCU and USB-to-UART pages should be placed. Text strings should be added near each test point.
LastName_FirstName_Lab3_Sch.pdf	Color PDF containing all the sheets in your design.  Use File→Smart PDF to generate the schematics. You can use the default settings, but only include the .SchDocs on the 3rd page of the dialog
LastName_FirstName_Lab3_Layers.pdf	Use Snipping Tool to capture the top and bottom layers of the board when you are in Single Layer mode (shift s). Use WORD to generate a single PDF with each layer on a separate page.

## 4. Lab Work

- A. Complete the placement of components from 02-Power.SchDoc and 07\_Buzzer.SchDoc.

You DO NOT have to route any signals, but try to arrange components in a way that reduces the number of overlapping signals.

- B. Modify all the reference designators so that they have a height of 35 mils and a width of 6 mils.

The following video shows you how to use a query to do this.

[PCB Query](#)

When you have finished, place the reference designators near their associated components. There should be no overlapping reference designators when you are finished.

## 5. Post Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab3_Altium.zip	<p>Altium Project ZIP file. All parts should be placed on the printed circuit board. There should be no parts flagged as causing a design error (no green error markers on the parts). All reference designators should be 35x6 mils. There should be no overlapping reference designators.</p> <p>The placement of parts should result in a near minimal number of overlapping net connections.</p>
LastName_FirstName_Lab3_Sch.pdf	<p>Color PDF containing all the sheets in your design.</p> <p>Use File→Smart PDF to generate the schematics. You can use the default settings, but only include the .SchDocs on the 3rd page of the dialog</p>
LastName_FirstName_Lab3_Layers.pdf	<p>Use Snipping Tool to capture the top layer and bottom layers of the board when you are in Single Layer mode (shift s). Use WORD to generate a single PDF with each layer on a separate page.</p>