Electrical and Computer Engineering

ECE315

Lab 4:

PCB Routing

1. Lab 4 Overview

In Lab 3, you placed all the components for your embedded system onto your printed circuit board. In this lab, you route all of the signals to complete your printed circuit board. In your lab session, you will perform a design review of your printed circuit board to ensure that you are ready to place an order to have your PCB manufactured. **Routing of the printed circuit board will take a fair amount of time, so be sure to get started early!**

2. Pre-Lab

A. Open the Altium Designer Project that you completed in Lab 3 by clicking on the **ECE315.PrjPcb** PCB Project File.

B. Printed circuit board manufactures publish a set of manufacturing specifications that you must follow if you want your printed circuit board to be manufactured at a reasonable cost. Altium allows you configure a set of design rules to meet these specifications.

   The project you have been provided has been configured to meet the manufacturing requirements for lowest cost PCB manufactures. Watch the video on the course website related to PCB Design Rules.

C. You are now ready to route the LEDs that were placed for you. DO NOT route the 5.0V and GND signals connected to the LEDs. You will do that in a later step.

   You can watch the PCB Routing Video found on the course website to complete the routing of your LEDs.

   You are NOT allowed to use the auto router to complete the routing of the board. Using the auto router will result in no points being awarded for this lab.

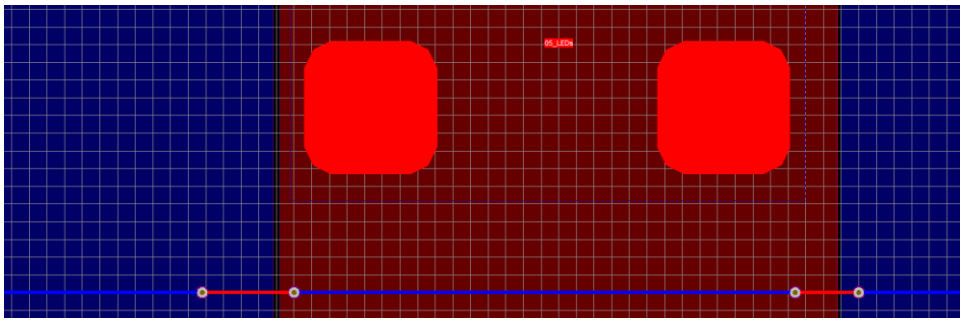
   You are NOT allowed to move any of the APA102C LEDs, mounting holes, or capacitors connected to the APA102C LEDs.

D. You will now define polygons for the signals related to 5.0V and GND. Starting at 11:22 of the routing video, follow along and create polygons for 5.0V on the top layer and GND on the bottom layer.

   Be sure to add vias to connect all of the LEDs' pad 3 to GND.

E. You are now ready to start routing the remaining connections on the PCB (minute 17 of the video). Here are a few items to consider as you start routing:

- Don't be afraid to rotate/move parts it's going to make routing the board easier. Rarely are all parts rotated in the most efficient manner after the first attempt at parts placement.
- Use an orthogonal approach to routing signals. When possible, vertical tracks should be on the top layer and horizontal tracks should be on the bottom layer. This is going to help avoid situations where a signal cannot be routed.
- If you have an exceptionally long track that create necks in your polygons, you can add a short length of track on the opposite layer. This is called stitching a signal. Stitching ensures that your polygons have good electrical connectivity and provide an adequate current return path.



F. Watch Adding Additional Text and Images video on the course website to see how to add your contact info to the PCB. I would suggest that the font size of your name is at least 120 mils. The rest of your contact information should be at least 100 mils.

G. Add ON/OFF text to the bottom silk screen of the board to indicate when power is being supplied by the battery.

H. Run a Design Rule check to verify that you have routed all of your nets and to find other issues that many lead to a non-functional PCB. You can run the design rule check from the PCB editor from Tools→Design Rule Check. You will want a report that lists 0 errors and 0 warnings.

3. Pre Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab4_Altium.zip	Altium Project ZIP file. All schematic pages should be completed, all parts are placed, routing is completed, contact information has been added, and the Design Rule Check results in 0 errors.
LastName_FirstName_Lab4_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_Lab4_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 Snipping Tool to capture the top and bottom layers of the board when you are in 3D mode (3). Use WORD to generate a single PDF with each layer on a separate page.

4. Lab Work

A. Your TA will assign groups of three people to perform a design review of your final .PcbDoc. The purpose of the design review is to help ensure that your design is ready to be fabricated. During the design review process, each of the designs will be examined using the check list found at the end of this document.

B. Once you have fixed any issues found in the design review, use Generating BOM and Build Files video to generate gerber files, NC Drill files, final schematics, and a BOM for your board.

C. If you do not have an account with Digikey, you will need to register an account with Digikey and order all the parts in your BOM. See the video related to this on the class website.

   I would highly suggest ordering some extra parts. It is easy to lose the capacitors and resistors that you are placing, **so order 10 extra capacitors**.

   If you are ordering as a group of 3+ students, I would also suggest that the group orders at least 1 extra of all your parts. If your board does not function correctly, you can use the extra parts to build a functional board using a known good PCB provided to you.

D. Order 10 copies of your PCB from PCBWay. The Ordering Parts and PCB video will demonstrate how to order the parts and printed circuit board.

   NOTE: You can save on shipping if you order in groups. One student will need to have the ZIP files with the gerbers and NC drill files for each student sharing the shipping cost. That student will upload and quote each card separately, but when you pay for the boards, you can choose to ship them together.

5. Post Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab4_Altium.zip	Altium Project ZIP file. All schematic pages should be completed, all parts are placed, routing is completed, contact information has been added, and the Design Rule Check results in 0 errors.
LastName_FirstName_Lab4_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_Lab4_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 Snipping Tool to capture the top and bottom layers of the board when you are in 3D mode (3). Use WORD to generate a single PDF with each layer on a separate page.

LastName_FirstName_Lab4_BOM.xlsx	The BOM generated by Altium. Each part should contain the quantity and manufacturer part number.
LastName_FirstName_Lab4_Orders.pdf	Use WORD to generate a single PDF with the following items: Use Snipping Tool to capture a picture of your Digikey Invoice after you have purchased your parts. Use Snipping Tool to capture the production status from PCB Way after you have paid for your printed circuit board. A PDF (or .jpg/png) of the completed design review. The image **must** have signatures Use Snipping Tool to capture an image of the resign rule check showing no errors or warnings.

6. Design Review Check List

	Schematic Requirements	Completed
1	All Nets have net names	
2	All parts have a unique reference designators	
3	All parts have supplier links that include the manufacturer and part num.	
4	Reference Designators are clearly visible and do not overlap.	
5	All integrated circuits have a 1uF bypass capacitor on EACH supply pin.	
6	Programming header has the correct nets on each pin	
7	MCU has the programming interface connected correctly.	
8	MCU has power and ground applied to the correct pins.	
9	MCU has the SPI interface correctly connected to the APA102C LEDs	
10	MCU has the UART interface correctly connected to the MPC2221A	
11	The piezo buzzer is connected to a GPIO pin on the MCU	
12	All unused pins should be marked with No ERC Connectors	

	PCB Requirements	Completed
1	Bypass Capacitors within 200 mils to the pin they service.	
2	Power signals like 3v3 should be carried by very thick traces (25-200mils) or preferably by dedicated polygon pours/planes.	
3	No conductive material within 50 mils of the board edge.	
4	There should be a 10 mil trace on mechanical layer 2 that defines the board outline	
5	Minimum hole size set to 15 mils	
6	Minimum annular ring of 6 mils on all vias and through hole parts.	
7	All nets are routed	
8	Polygon pours used to route 5.0V and GND.	
9	Contact information has been placed on the PCB	
10	Reference designators for each part clearly indicate which part they are associated with	
11	All test points have a text string in a silk screen layer indicating what net they are connected to.	
12	Design passes Design Rule Check with 0 errors and 0 warnings	

Originating Engineer Signature : _____

Reviewer #1 Signature :_____

Reviewer #2 Signature :_____