

ECE315 Introductory Microprocessor Laboratory

Lab 3

Parts Placement and Routing

1. Introduction

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. If you want some background information on what a PCB is, Altium provides some basic [information](#) about PCBs and how they are manufactured. Read through the start of the document through the section on Vias. **This lab will take more time than the previous labs, so get started sooner rather than later.**

After the components are transferred to the PCB doc, you will arrange the 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.

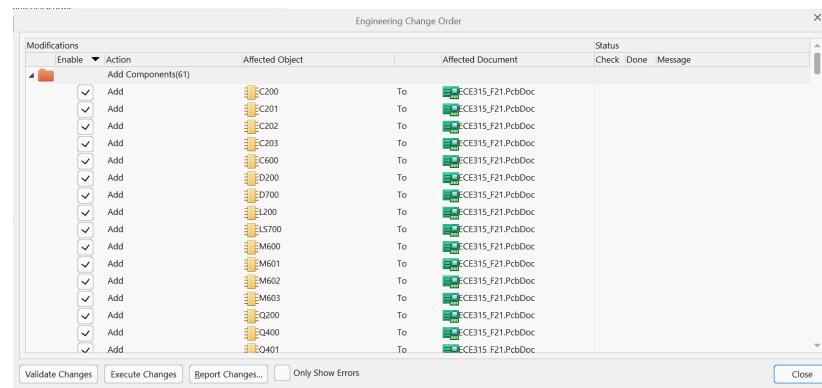
So what determines where components are placed? Components are sometimes placed in specific locations to meet the mechanical requirements of an enclosure. There may be a protrusion in the enclosure that prevents you from placing tall components in certain locations of the board. There may also be situations where the functionality of a sub circuit will dictate where components will be placed. For your design, the components placement is mostly driven by the “user experience”.

There is a link at the end of the Lab 3 manual that will walk you through examples of how to place parts and route the board. If you’re having trouble following the written instructions, be sure to watch the videos. **The videos show a slightly different circuit than what you are designing, but it will show you the basics of how to use Altium to place parts.**

2. Transfer Footprints to PcbDoc

In order to place the remaining parts, we need to transfer the PCB footprints from the schematic document to the PCB document.

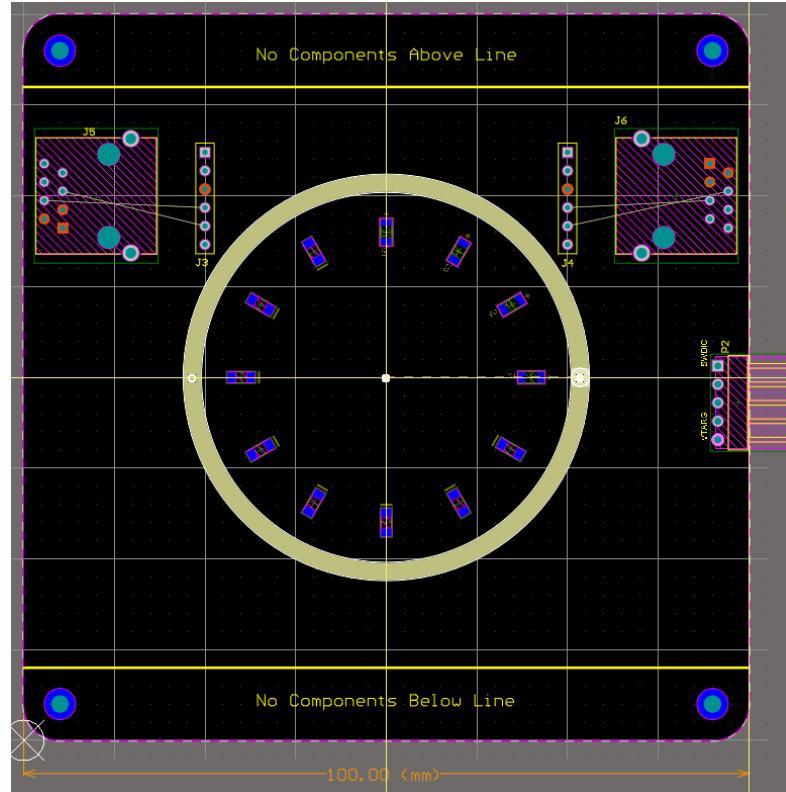
- a) Open 06 MCU.SchDoc
- b) Design □ Update PCB Document. The following dialog will appear.



- c) Click on Validate Changes.
- d) Click on Execute Changes.
- e) Select the checkbox "Only Show Errors". If any errors are reported, fix the errors reported and start at step b. again.
- f) Click on Close.

3. PCB Canvas Overview

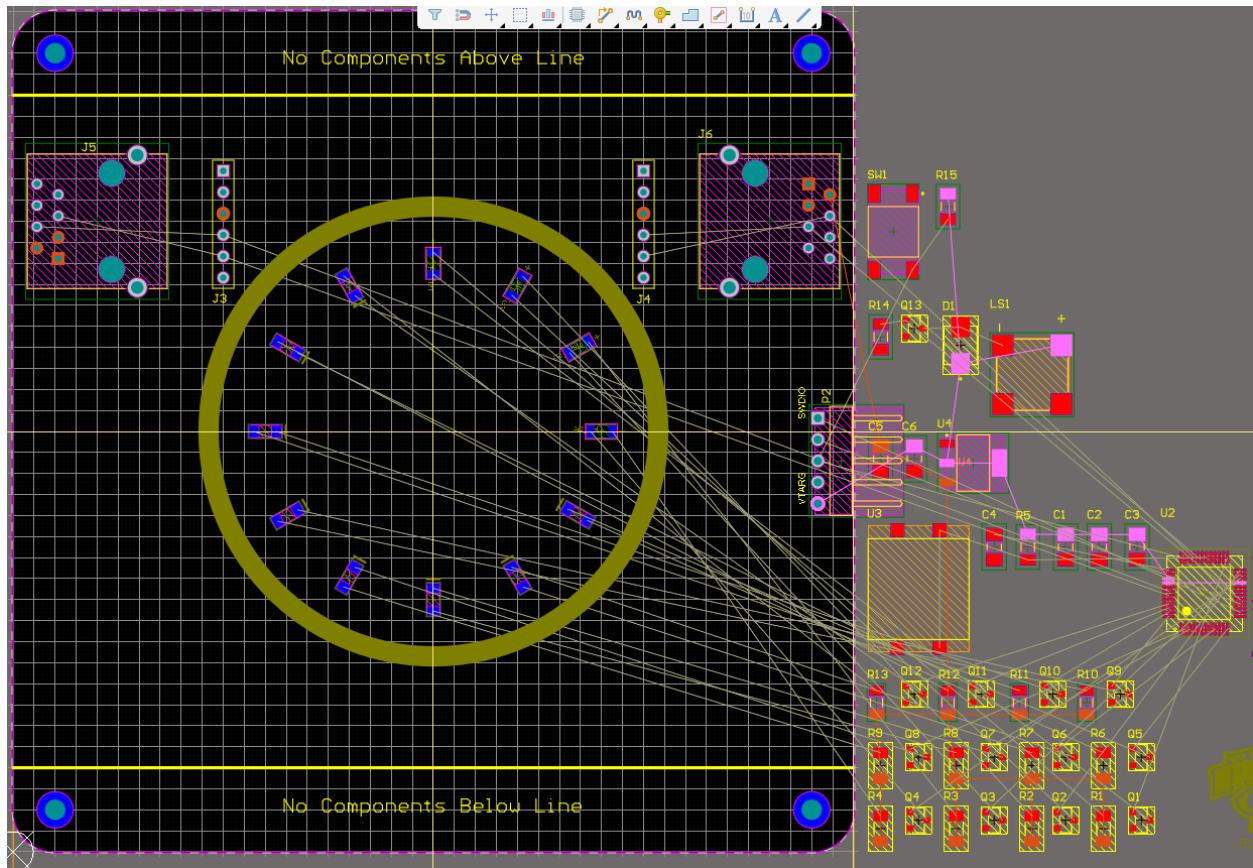
The shape and the size of your PCB has already been defined for you. This shape was chosen to minimize the cost of manufacturing the PCB.



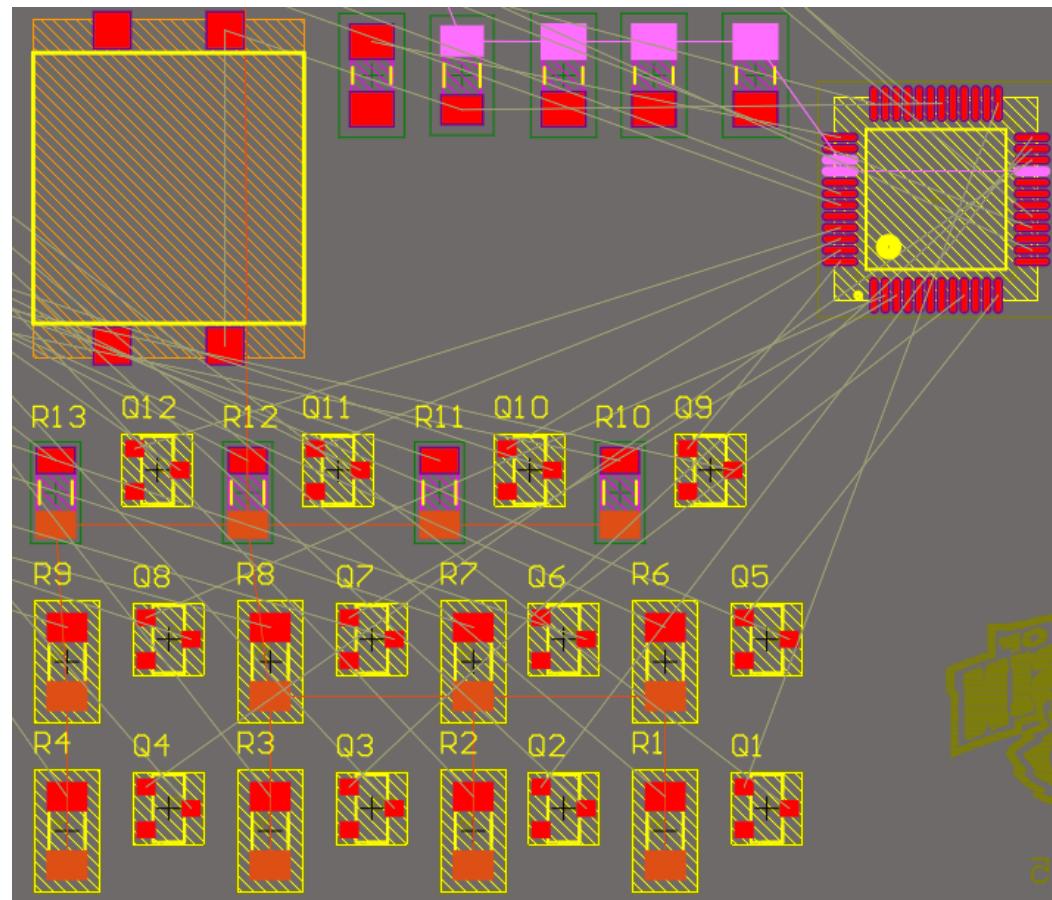
As you can see, the location of some of the components have already been determined. The RJ45 connectors (J5, J6), programming header (P2), and UART debugging headers (J3, J4) have been placed on the top side of the board.

The LEDs (blue pads) have been placed on the bottom side of the board. The locations of these LEDs have been locked so that they are placed on a polar grid that radiates from the center of the board.

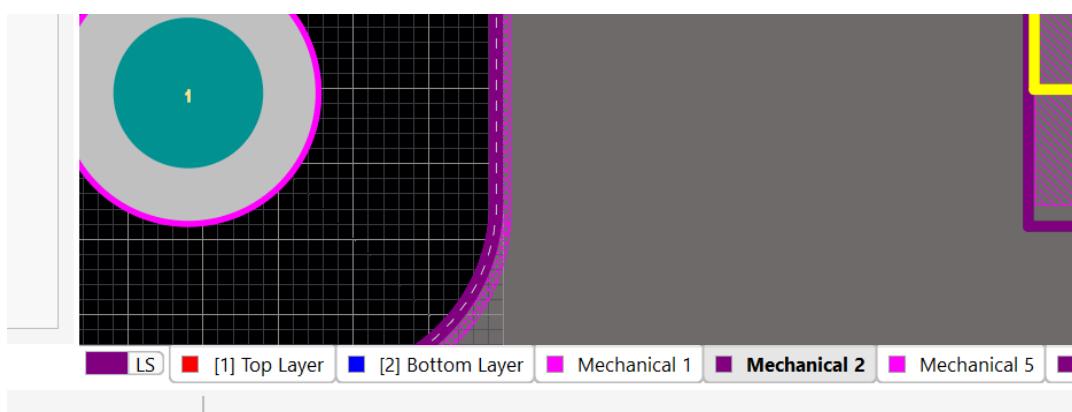
If you look to the right of the right of the PCB, you will see the components that were transferred to the PCB Doc. All of the components that will be placed on the top layer of the board have red SMD pads. The red pads are used as a visual indication that these parts are on the top layer of the board. Blue pads indicate that a part is on the bottom layer of the board.



There are also tan lines that run between some of the pads. These tan lines represent wires that need to be routed. The collection of unrouted wires is oftentimes referred to as the “rat’s nest”.



At the bottom of the screen, you will see several tabs that indicate the layers of the PCB. When routing wires (nets) on the top layer, you will want to make sure that the tab for the top layer has been selected.

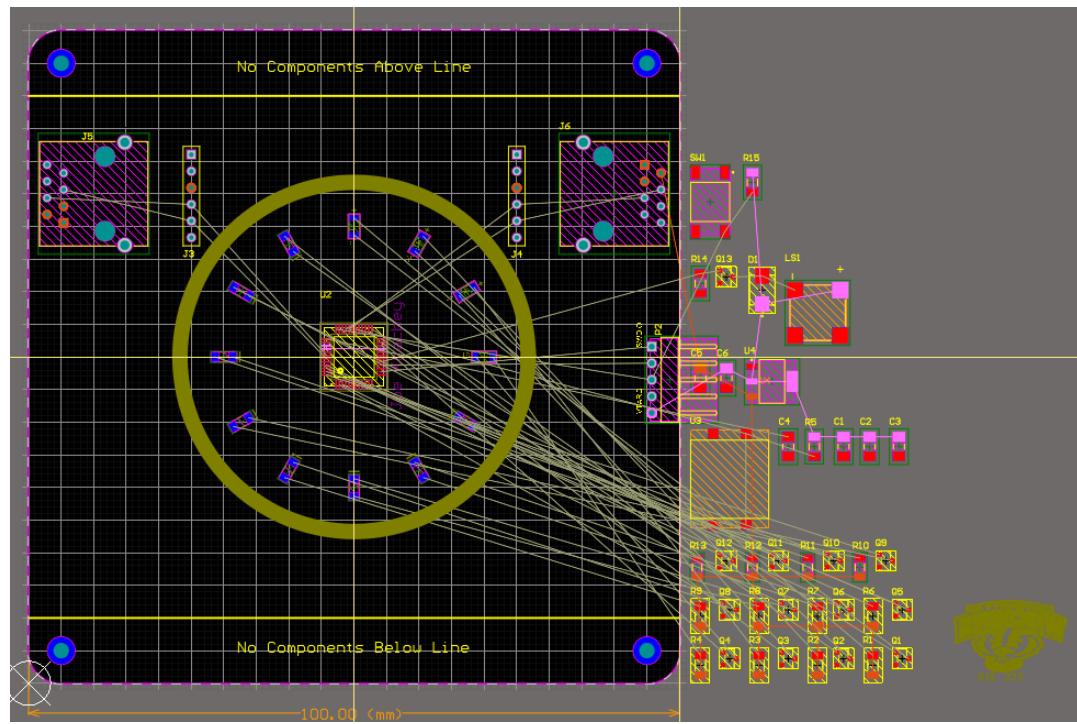


4. MCU Placement

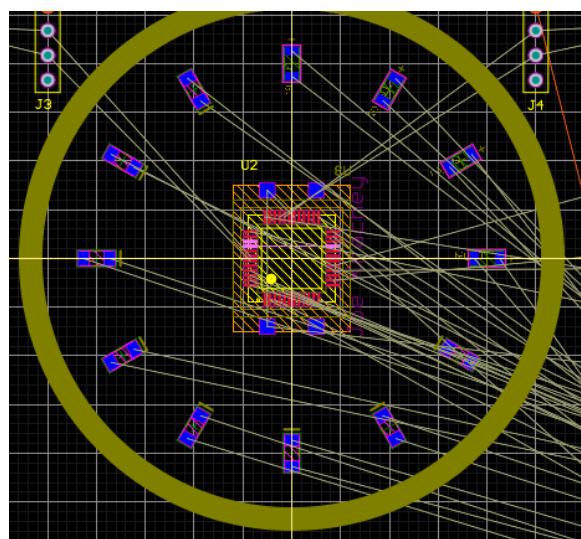
When deciding where to place parts on the PCB canvas, some parts need to be placed with care based on functional requirements of the board. The LEDs were placed on the bottom side of the board in a circular pattern so that the user can see the amount of time remaining tick down like a clock.

The MCU will be placed on the top side of the board so that the user does not see the microcontroller. There is no “functional” requirement on where to place the MCU. I would suggest placing the MCU in the center of the board (50mm,50mm). The parts placements shown in this document are a suggestion based on experience and trying several different attempts at “optimizing” the layout.

- a) Set the default unit of measurement to be mm. Typing **q** will toggle between mils and mm.
- b) Set the Grid to be 1mm. (**g**, then 1mm)
- c) Select MCU and drag it to the center of the canvas.
- d) Components can be rotated using the **Spacebar** as the part is being moved.

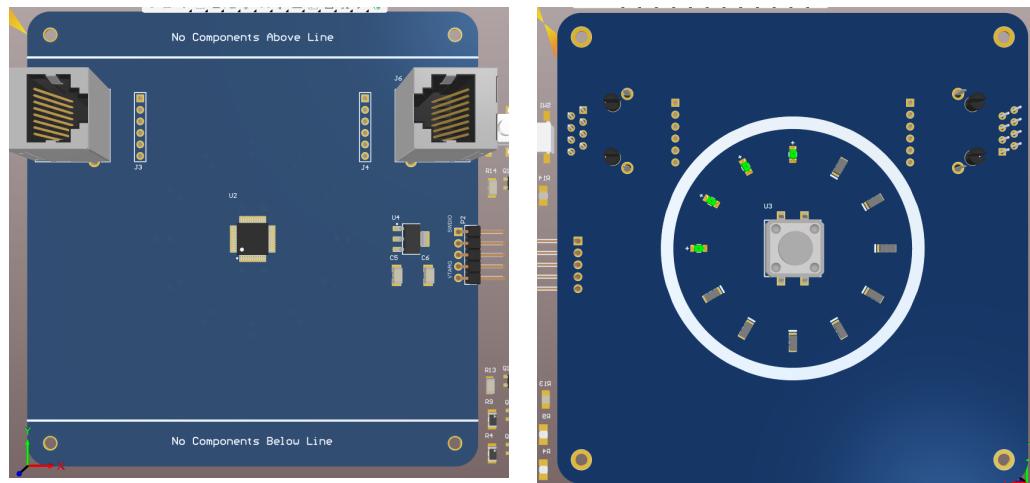


- e) Select the large tactile button. This part will need to be placed on the bottom side of the board. As you move the part, pressing the **L** key will toggle the part between the top and bottom layers. Move this part so that it is centered on the board (50mm,50mm).



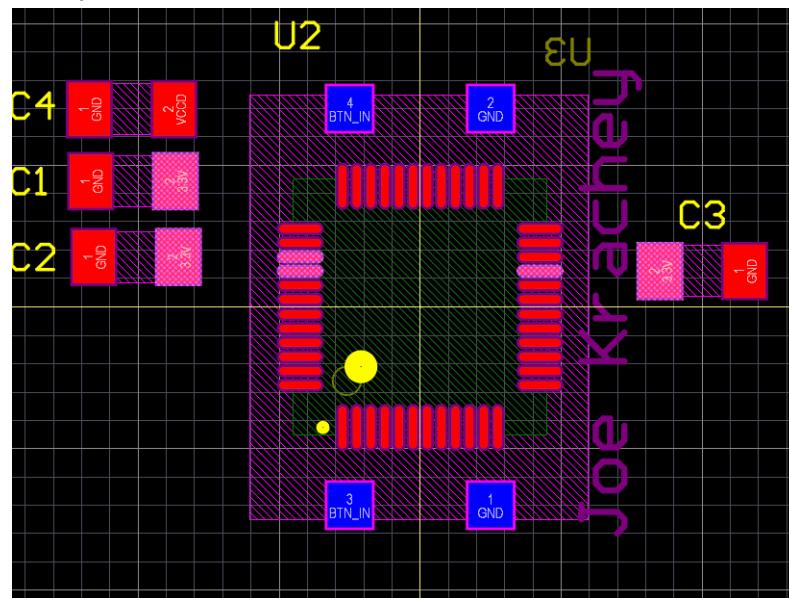
It may seem like the button and MCU are overlapping each other. Remember that one parts is on the top layer (MCU) and the other (button) is on the bottom. You can see a 3D rendering of the PCB by pressing the **3** key.

To flip the side of the board you are looking at, type **vb**.



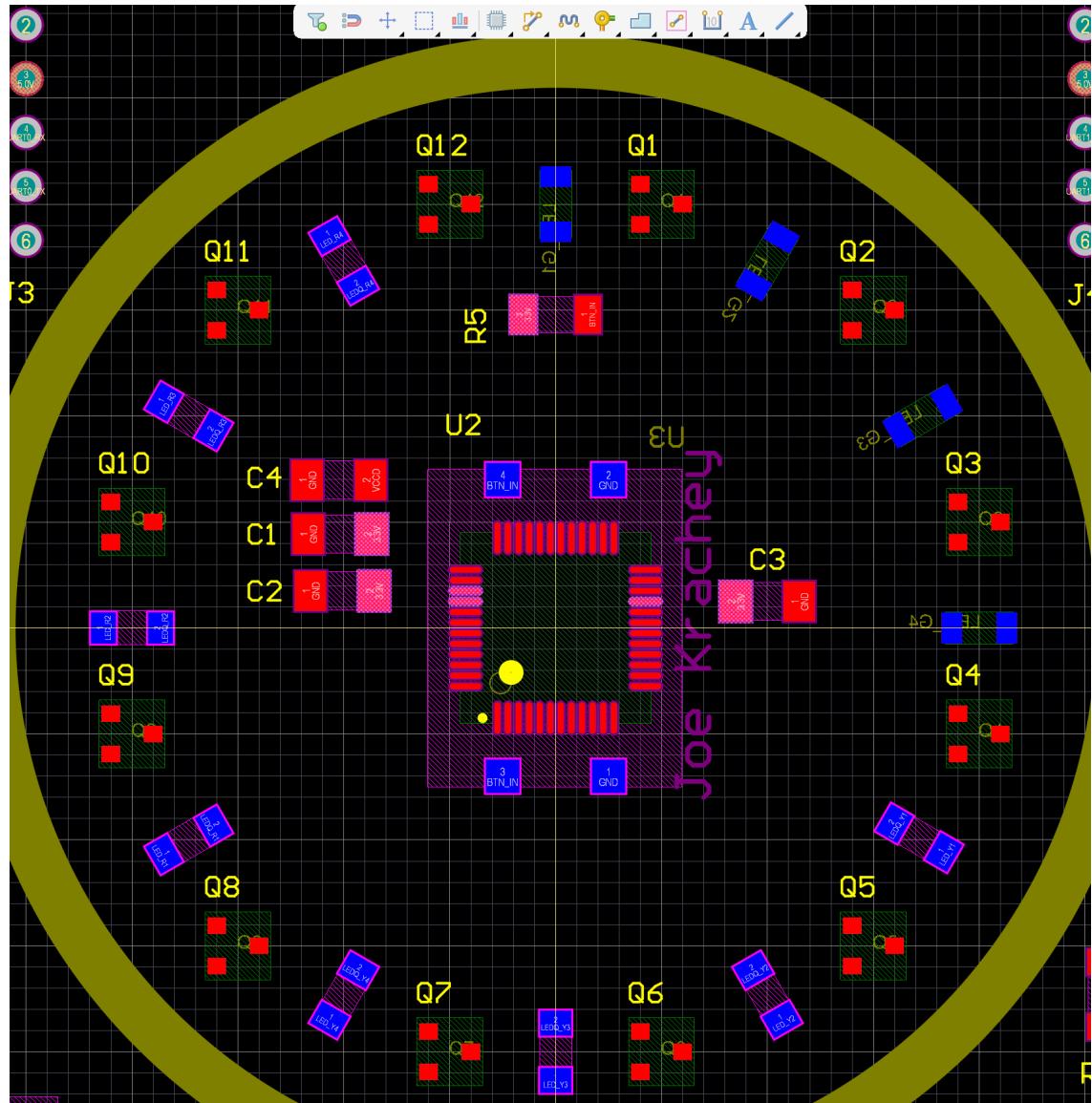
When you are finished looking at the PCB in 3D mode, press 2 to return 2D mode.

- f) Move the by-pass capacitors so that there is one capacitor within 5-6mm of the supply pins.

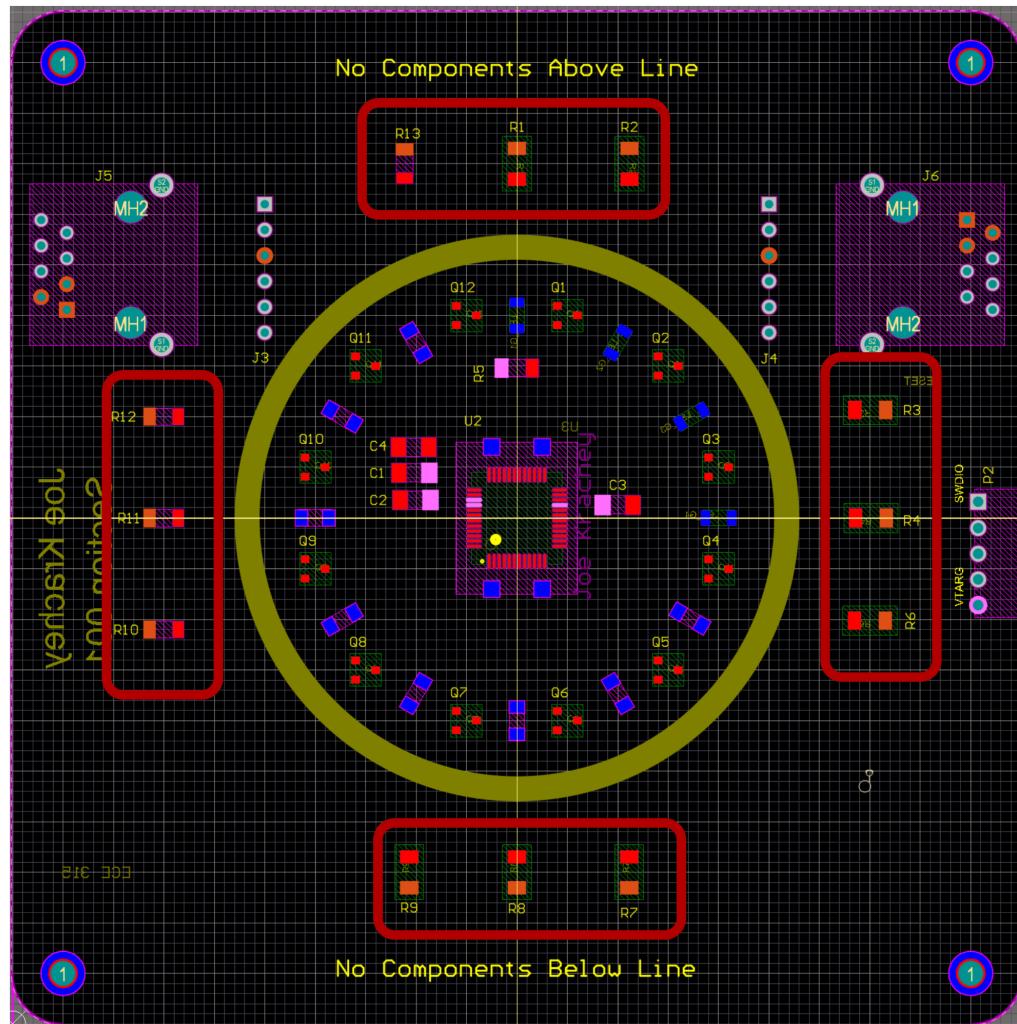


You can rotate parts and designators by pressing the **SPACE** bar.

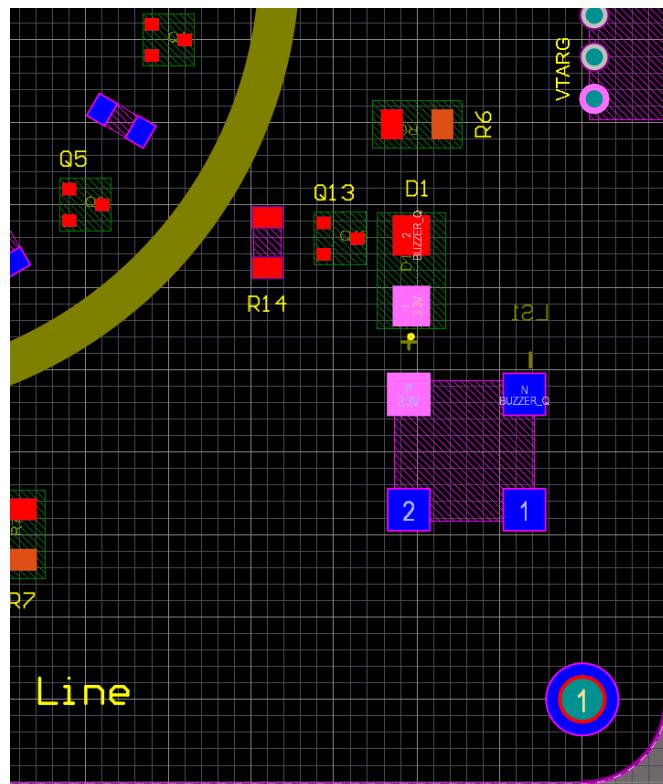
- g) Move the N-Channel FETs so that they are centered on the 1mm grid. You can arrange them as shown below.



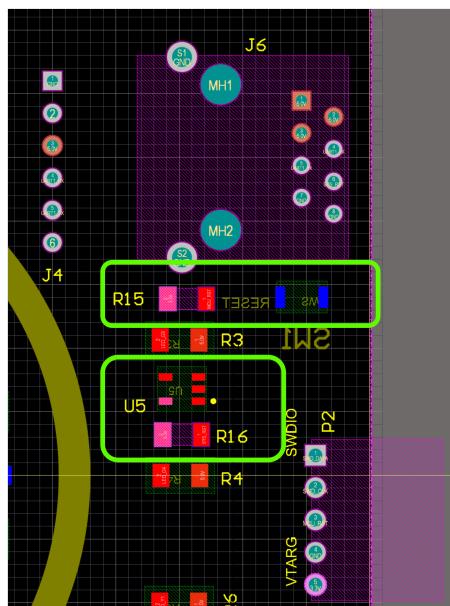
- h) Arrange the current limiting resistors so they are near the LEDs they are connected to.



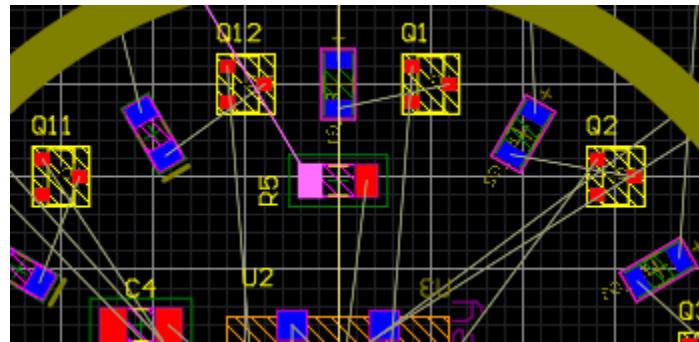
- i) Move the components for the buzzer into the lower right hand corner. Make sure the buzzer is placed on the bottom side of the board.



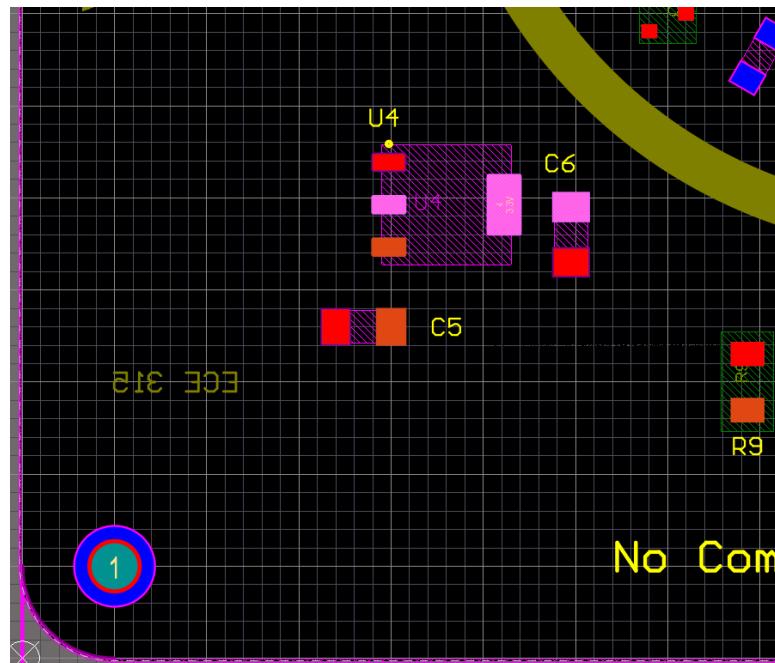
- j) Move the XRES push button, pull-up resistor, and AND gate so that it is above the programming interface. The button itself should be moved to the bottom layer.



- k) Move the last pull-up resistor so that it is above the user push button.



- l) The components related to the voltage regulator should be arranged to be similar to the image below.



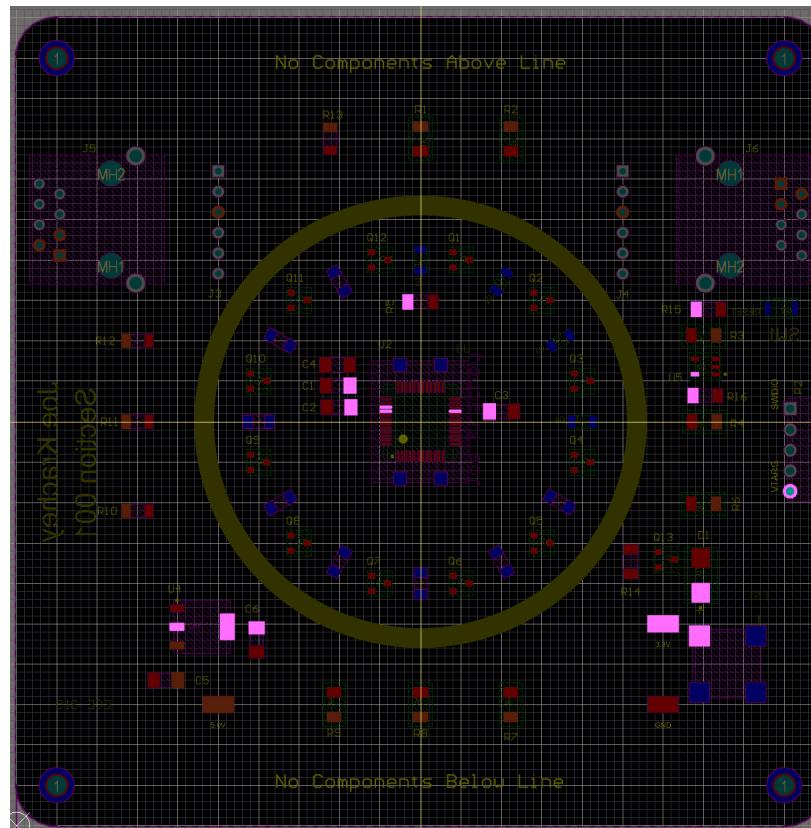
5. Polygons

[Polygon pours](#) are used in Altium designer to create solid areas on the PCB.

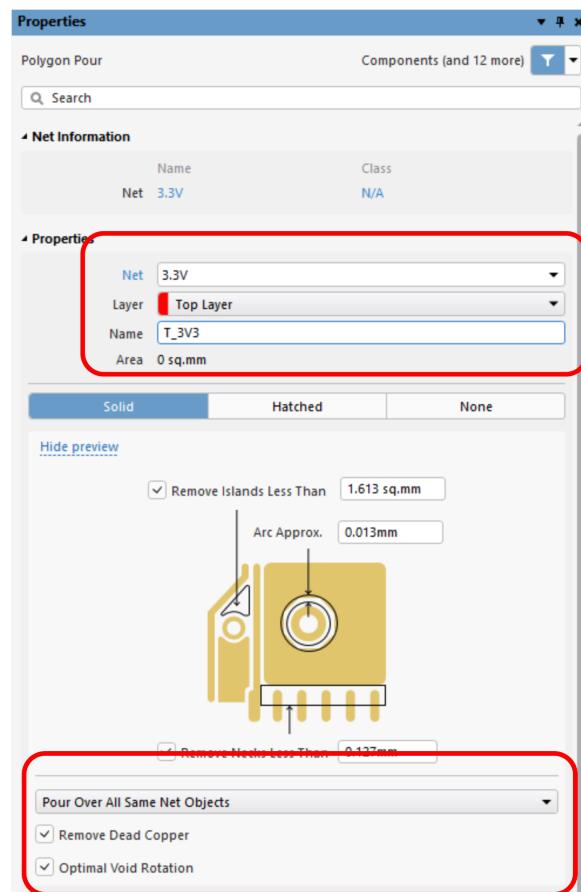
Polygons are commonly used to create power planes for a printed circuit board. Connecting signals with a polygon is beneficial for a few different reasons:

- Planes of conductive material have more ability to attenuate noise
- Planes of conductive material have less impedance, allowing the system to deliver larger amounts of current.
- When a signal is connected to a plane, we can quickly interconnect component pads that are connected to the signal.

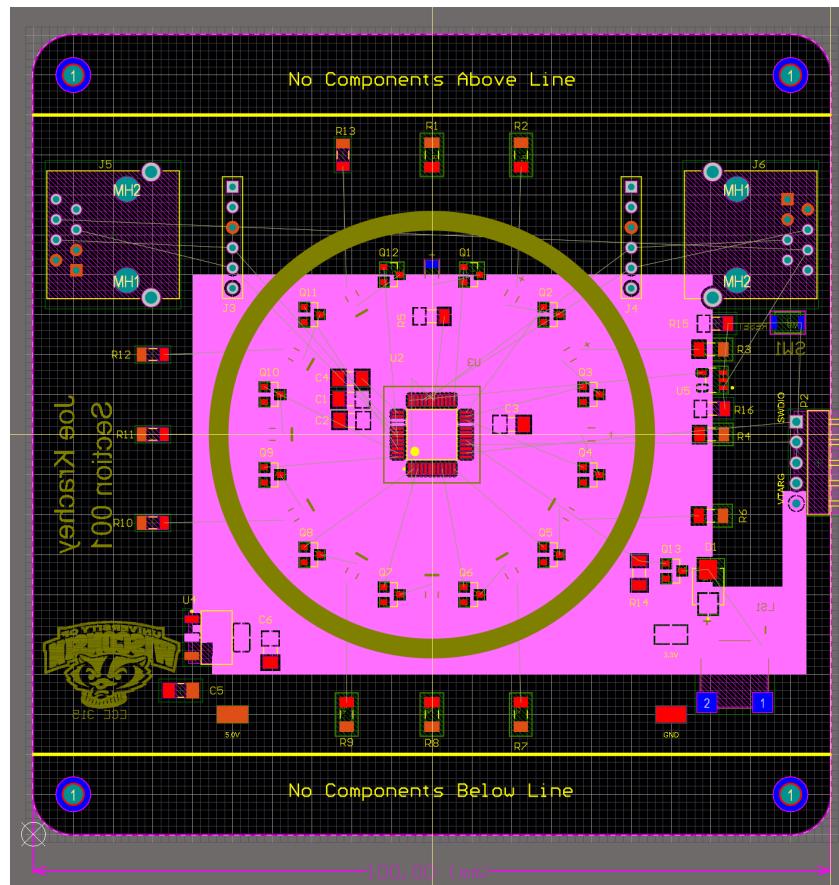
We are going to use polygon pours to interconnect the GND, 3.3V, and 5.0V rails. We will start with the 3.3V polygon. We will place this polygon on the top layer of the board. In order to determine where to draw the polygon, we can see where all of the 3.3V pins are located using **CTRL + LEFT MOUSE CLICK**. You can press the 'l' key to make the selected pins more pronounced.



- a) A polygon is placed by typing **pg**. Before placing the first vertex of a polygon
- b) Press **TAB**
- c) Select 3.3V as the signal name.
- d) Make sure the layer is listed as Top Layer
- e) Give a polygon name as T_3V3. This is an arbitrary name, but it tells the user it is on the Top Layer and is the 3.3V net.
- f) Select “Pour over all Same Net Object”
- g) Click on Remove dead copper
- h) Press Enter



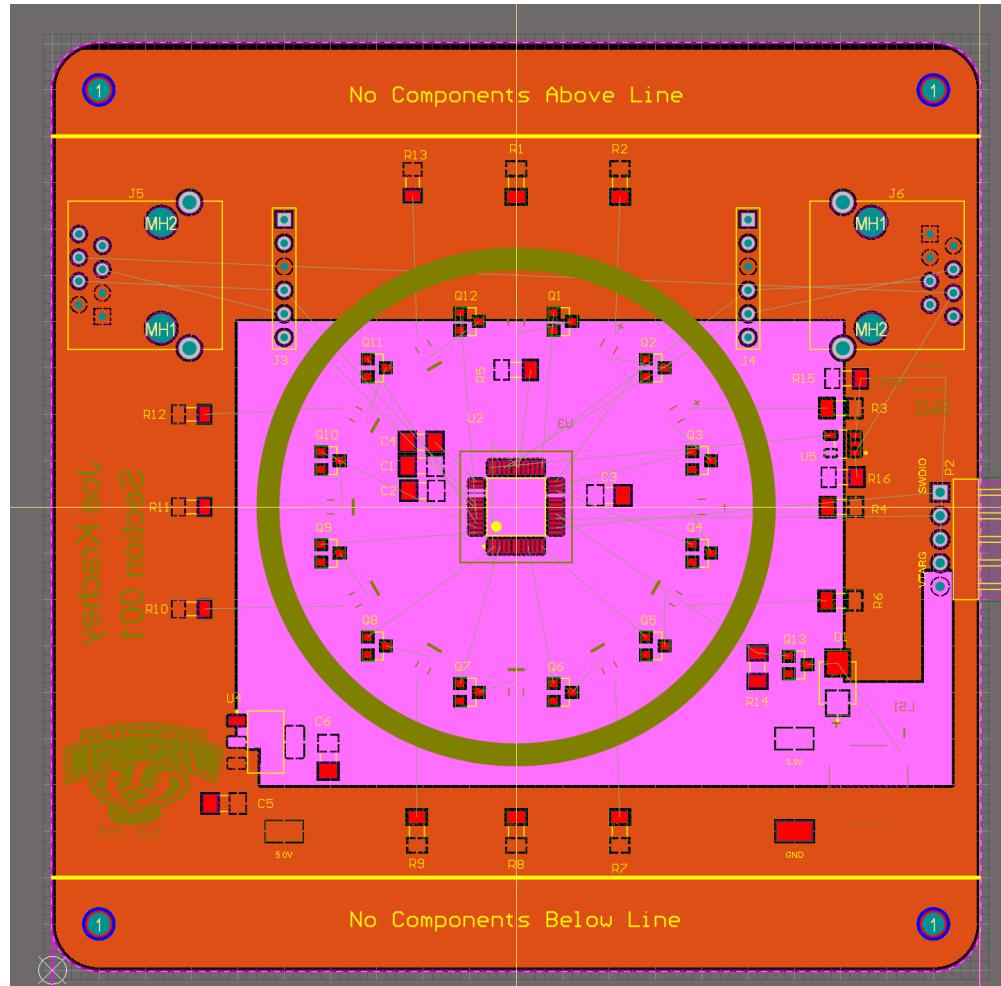
- i) Left clicking on a polygon will set the location of the vertices in the polygon. Draw a shape that covers the 3.3V pins. You can either use the arrow keys or the mouse to create the shape of the polygon (You are not limited to rectangles). Each time you click the left mouse button, you set a vertex of the polygon.



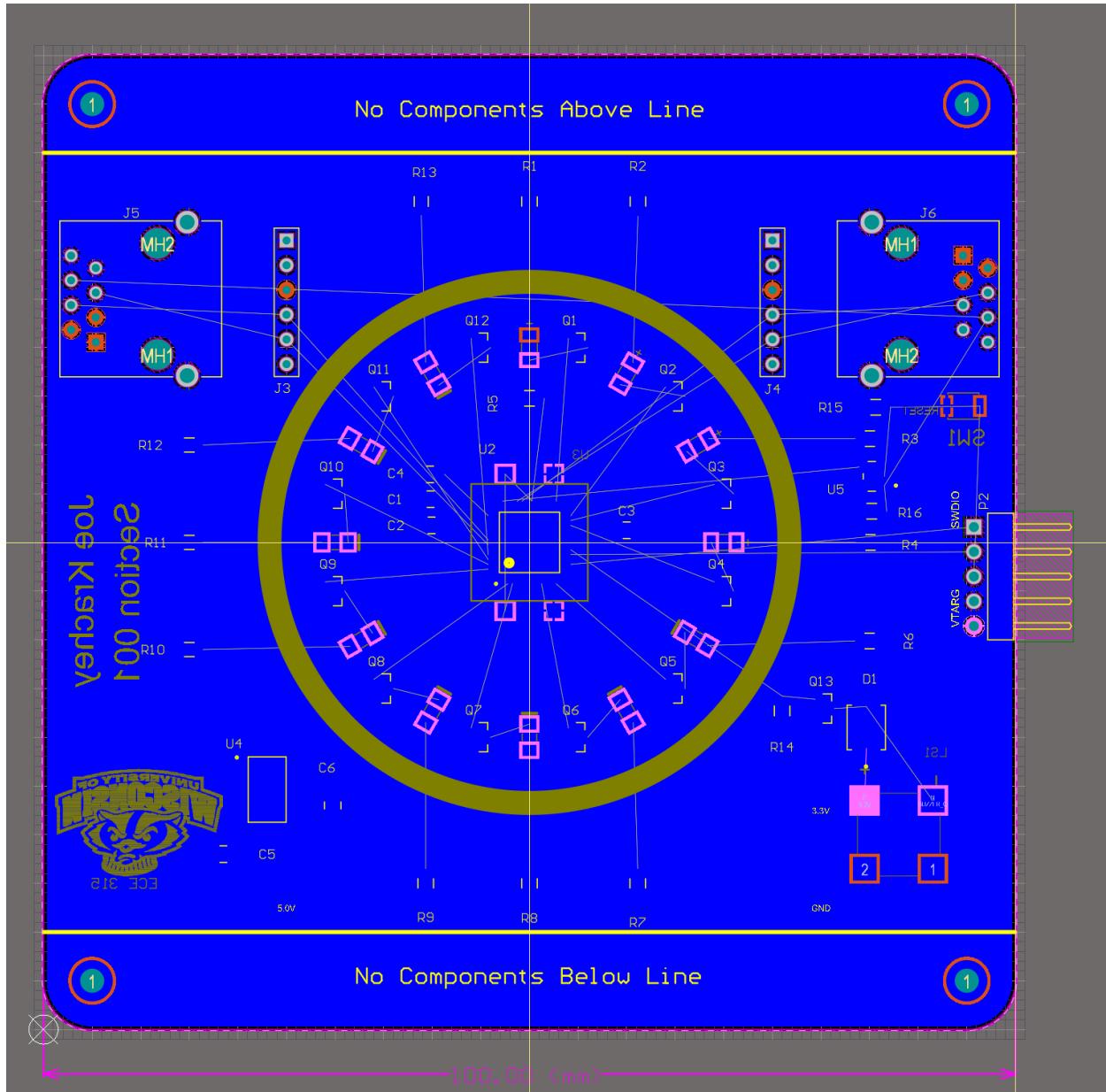
NOTE: Your polygon placement and shape DO NOT need to be an exact match of what is shown. The images shown are for demonstration purposes only.

- j) Press CTRL-C to unselect the 3.3V Net.

- k) Repeat the steps above for placing a 5.0V polygon on the **top layer**. This polygon should cover the entire board. When you are finished, your top layer should look like the image below.



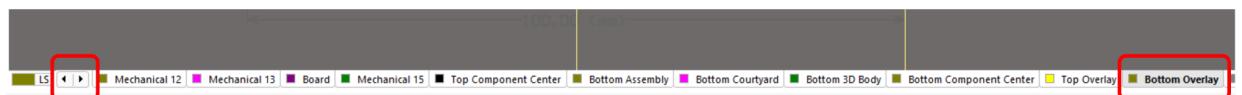
- I) Repeat the steps above for placing a GND polygon on the **bottom layer**. This polygon should cover the entire board. When you are finished, your top layer should look like the image below. Make sure to click on the bottom layer so you can view the bottom layer polygons.



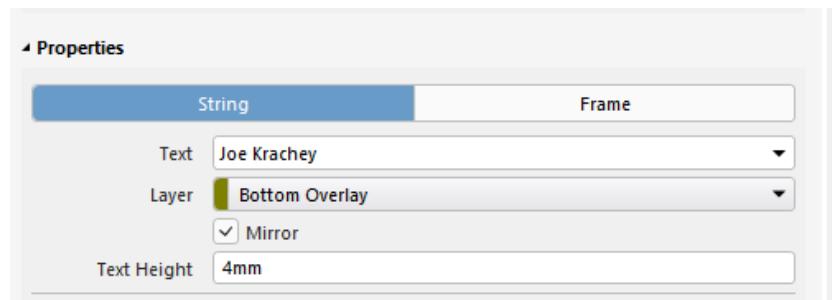
- m) Add your test points to the design. Place the test points on the top layer. Select a location where the test point is contained within the perimeter of the board.

6. Adding Informational Text

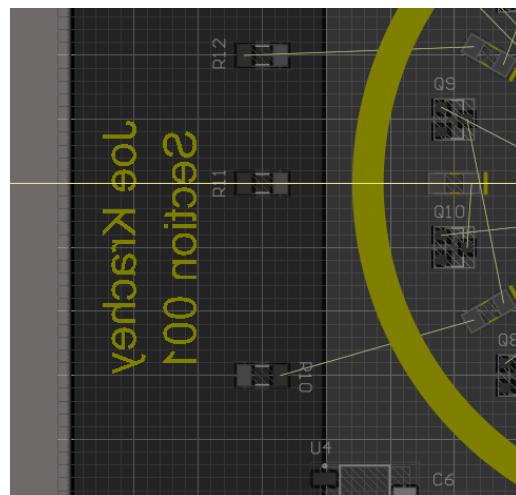
- Select the Bottom Overlay as the current layer. This can be done by using the arrows in the lower left corner.



- Place a string (**ps**).
- Press TAB and change the string to be your name.
- Make sure the Mirror check box is chosen.
- Change the Text Height to 4mm.



- Place the string near the left hand side of the board.
- Repeat this process and add your Section Number to the board.
- When you are done, it should look like the image below.



- i. Add strings to the top layer of the board for each test point.
 - i. The text should be 5V, 3V3, and GND
 - ii. The strings you place should be 1mm Text Height
 - iii. Because this is the top layer, do NOT select the Mirror check box.

7. Routing Signals

- a) You can place a track in Altium using the **pt** (place track) keys.
- b) Click in the center of the pad that will start the route. As you move the mouse, a track will be placed along the direction you move the mouse.
- c) You can place inflection points by left clicking on the PCB canvas.
- d) An orthogonal routing approach will help save time and effort in routing. All tracks on the top layer should go in the same direction (X direction). All tracks on the bottom layer should go in the opposite direction (Y direction).
- e) When trying to switch which layer a track is on, you can type the **+** key. This will place a via on the board and swap the currently active layer from top to bottom or from bottom to top.

The following [video](#) will show you some of the basics of routing signals.

After watching the tutorial video, you should now begin the process of routing your PCB.

- 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.
- Don't be afraid to rotate/move parts if it makes 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. This is going to help avoid situations where a signal cannot be routed.
- Be sure that you connect all of the GND pads to the GND polygon on the bottom layer of the board using vias.
- Verify that all power pins are connected to their respective polygon pours.
- All nets should be routed.

8. Running a Design Rule Check

When you have finished routing your board, run a design rule check (Tools→Design Rule Check). Examine the results of the design rule check and fix the errors that are reported. You can ignore any errors related to silk screen or board clearance.

When you have removed any errors that are reported save a screen capture Design Rule Verification Report Summary. Name the screen capture DesignRuleCheck.png.

Example

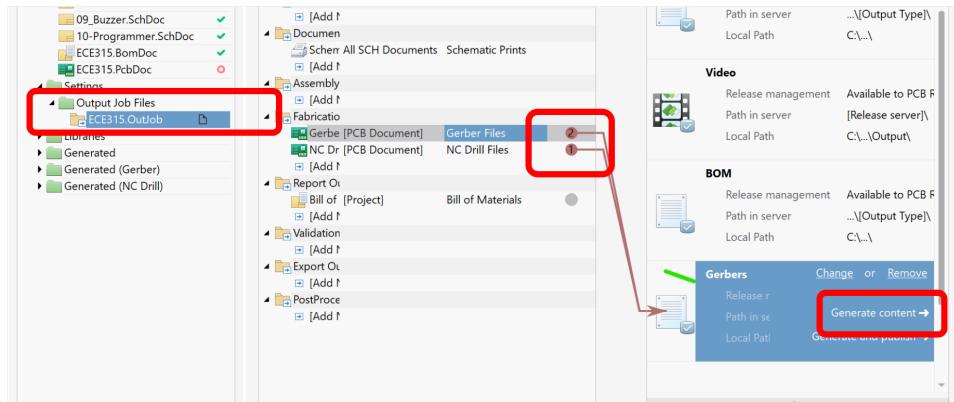
Summary	
Warnings	Count
Total	0
Rule Violations	
Clearance Constraint (Gap=10mil) (All).(All)	0
Short-Circuit Constraint (Allowed=No) (All).(All)	0
Un-Routed Net Constraint (All)	117
Modified Polygon (Allow modified: No), (Allow shelved: No)	0
Width Constraint (Min=99999mil),(Max=100mil),(Preferred=10mil),(HasFootprint("BUCKYBADGER") or HasFootprint("Motion_W"))	0
Width Constraint (Min=10mil),(Max=100mil),(Preferred=10mil),(All)	0
Power Plane Connect Rule(Relief Connect)(Expansion=20mil),(Conductor Width=10mil),(Air Gap=10mil),(Entries=4),(All)	0
Minimum Annular Ring (Minimum=6mil),(All)	0
Hole Size Constraint (Min=15mil),(Max=200mil),(All)	0
Hole To Hole Clearance (Gap=10mil) (All).(All)	0
Minimum Solder Mask Sliver (Gap=0mil) (All).(All)	0
Silk To Solder Mask (Clearance=0mil),(sPad),(All)	0
Silk to Silk (Clearance=1mil) (All).(All)	0
Net Antennae (Tolerance=0mil) (All)	0
Board Clearance Constraint (Gap=0mil) (All)	0
Height Constraint (Min=0mil),(Max=1000mil),(Preferred=500mil),(All)	0
Total	117

When you have completed your layout of the PCB, save a screen capture of your PcbDoc. Be sure to have the Top Layer selected and the entire board is visible. Name the screen capture Routing.png.

9. Generating Build Files

You will now need to generate a set of build files, or gerber files, to send to a PCB manufacturer. You can use the following instructions to generate these files. Make sure that you have finished your design and that your design has passed the design rule check before generating your build files.

- a. Delete the ECE453.OutJob file from your project
- b. Download the ECE315.OutJob file from the Lab 3 Assignment. Save it into your project folder.
- c. Right click on the ECE315.PcbPrj and select “Add Existing to Project”. Select the ECE315.OutJob
- d. Click on Output Job Files and then Click on ‘Generate Content’ to generate the gerber files.



- e. From your project directory, enter the Output subdirectory.
- f. Select both the Gerber and NC Drill directories in Windows Explorer, right click, and add them to a Zip file. Name the Zip file ECE315_LastName_Firstname.zip. Replace LastName and FirstName with your first and last name.

10. Forming a Group

You will need to form a group of 4 students for the remaining labs. You can use this [form](#) to register for a group. If you do not have a group, please fill out the form and indicate that you need to be assigned to a group.

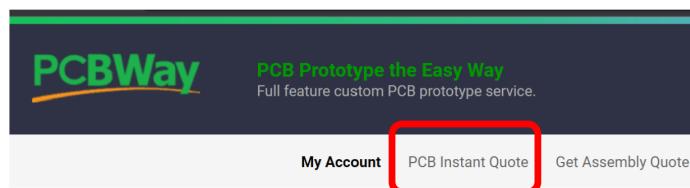
Groups will need to be submitted no later than Friday Oct 13th.

11. Ordering PCBs

You will need to order your printed circuit board from a company called [PCBWay](#). The cost for producing and shipping your PCB is roughly \$30. The PCB is only \$5 of that cost, the rest is shipping. I would recommend that a single person from your group orders all of the PCBs at once. You will have to pay much less in shipping per student if you do. As an example, if you order all 6 of your boards at once, you will pay \$30 for the PCBs and a total of \$30-\$40 in shipping.

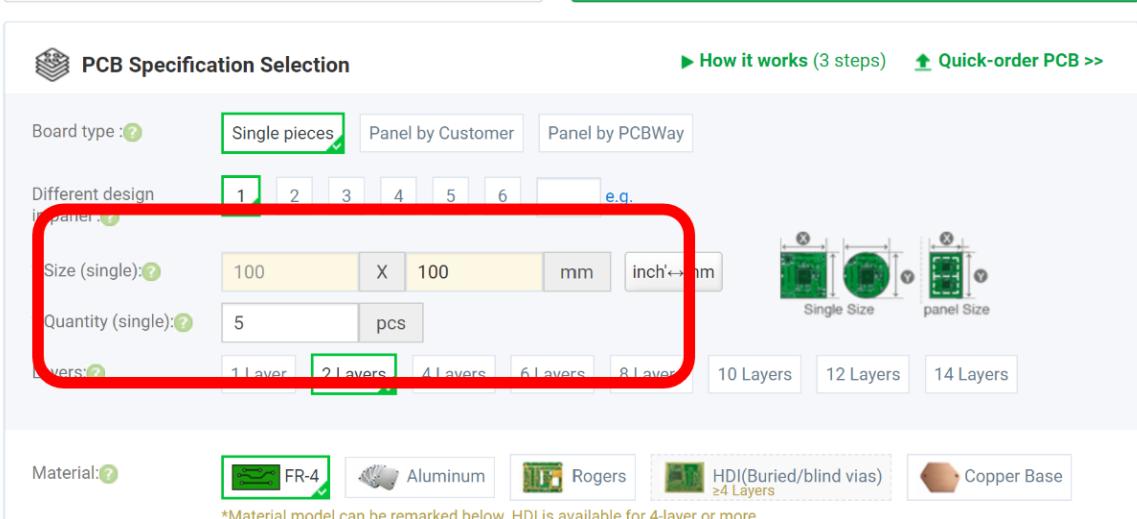
Your PCBs need to be ordered by the end of the day on Wednesday Oct 18th. You are required to turn in a PDF that shows all of your Teams boards have been ordered.

- a. Sign up for a new account.
- b. Once you are signed in, click on “PCB Instant Quote”



- c. Enter a board size of 100x100mm.

- d. Select that you want 5 boards fabricated (this is the minimum number of boards, but only cost a total of \$5).



The screenshot shows the 'PCB Specification Selection' interface. The 'Quantity (single):' input field, which contains the value '5', is highlighted with a red box. Other visible fields include 'Size (single):' set to '100 X 100 mm', and 'Layers:' set to '2 Layers'. There are also tabs for 'Single pieces', 'Panel by Customer', and 'Panel by PCBWay'. A note at the bottom states: '*Material model can be remarked below. HDI is available for 4-layer or more.'

- e. Click 'Save to Cart.'
- f. This will display the cost of your PCB. If the cost is under \$30, then click 'Save to Cart'. This will allow you to upload your Zip file containing your build files.
- g. You will need to repeat these steps if you are going to order additional boards in order to save on shipping costs. You/Your group will need to cover the costs of fabricating and shipping your own PCBs.

12. PCB Editor Shortcuts

Action	Short Cut
Place Track	pt
Place Polygon	pg
Place String	ps
Place Via	pv

Place Via during route	+
Rotate	Space
Exit Current Mode	Esc
Modify Properties of Selected Object	Tab
Zoom	Mouse Scroll Wheel
Pan	Right Mouse Button
Highlight a Net	CTRL + Left Click on the net
Turn off Highlighting/Selection	SHFT + C
Pan	Right Mouse Button
Changing Component Layer	L Key
Single Layer Mode	SHFT + S
Change Grid Size	G Key
3D Mode	3 Key
2D Mode	2 Key