ECE315 Introductory Microprocessor Laboratory

Lab 1

Creating Schematic Symbols and PCB Footprints

1. Introduction

An embedded system is a collection of integrated circuits, sensors, and other components that are chosen to accomplish a specific task. These components are soldered onto a printed circuit board that interconnects the components as specified by the system designer.

You have all probably taken other laboratory courses where you design a circuit using a breadboard and through hole parts. This is a great way for students to learn key concepts and take measurements used to validate their designs. Industrial design of embedded systems is much different. Instead of using bread boards, we use a printed circuit board (PCB) to help ensure reliability and manufacturability that cannot be accomplished with a breadboard. This course will introduce you to some of the tools and practices used to design a PCB.

When designing an embedded system, an embedded systems engineer will use a schematic editor to generate a set of schematics for a design. The schematics define how each component is used and how components are interconnected.

This semester, you will be modularized Wack-A-Mole game using the Infineon [CY8C4045AZI-S413 PSoC4 microcontroller](#) and various other components. This design will be fairly straightforward, but it will help you to learn some of the basic principles of schematic design and PCB layout.

2. Altium Designer

ECE315 will require you to use Altium Designer to complete a custom printed circuit board (PCB). Altium Designer is available on any CAE PC. Additionally, you have the option to install Altium Designer on your own PC by applying for a free [student license](#) from Altium.

The instructions below will include commands that can be issued in Altium. The commands will be highlighted in **bold red letters**.

3. Base Project Download

Download the Altium base project from the Canvas Lab 1 assignment page. You should extract the contents of the ZIP file to your CAE network drive. You will use this base project for labs 1-3.

4. System Settings

Before you get started designing a project in Altium, you may want to go through and adjust some of the system level settings. Altium is highly configurable, so please be aware that different individuals may not find these settings to be optimal, but I have found that they are helpful based on the use cases for this course.

1. Extract the Altium base project from the course distribution.
2. Open the Altium project by double clicking on ECE315.PrjPcb
3. Click on the Gear Icon in the upper right hand corner of Altium
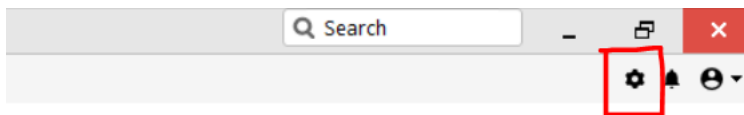
4. From the Preferences dialog, select System->View
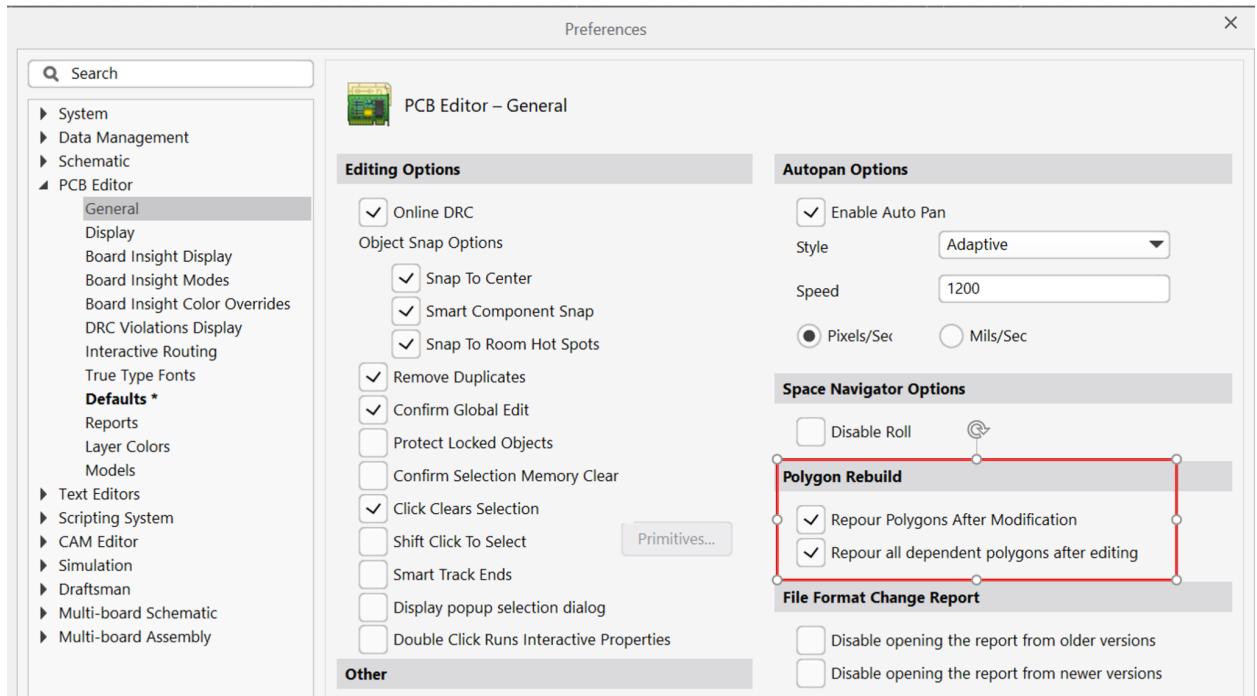5. Select the UI theme as Altium Light Grey (optional)
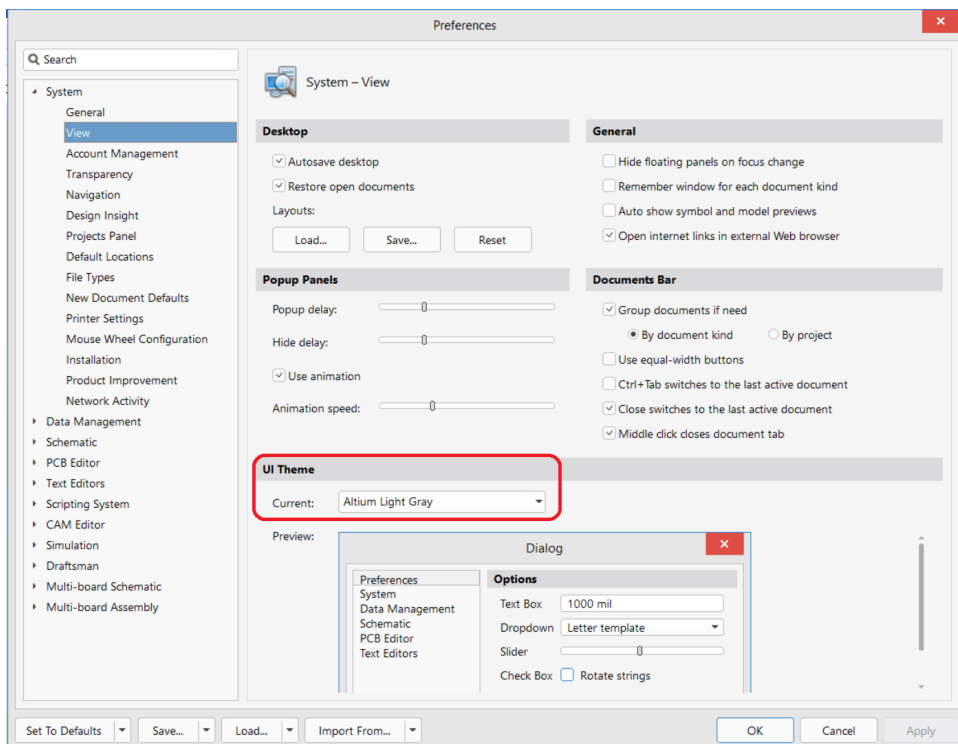
6. From the Preferences dialog, select Schematics
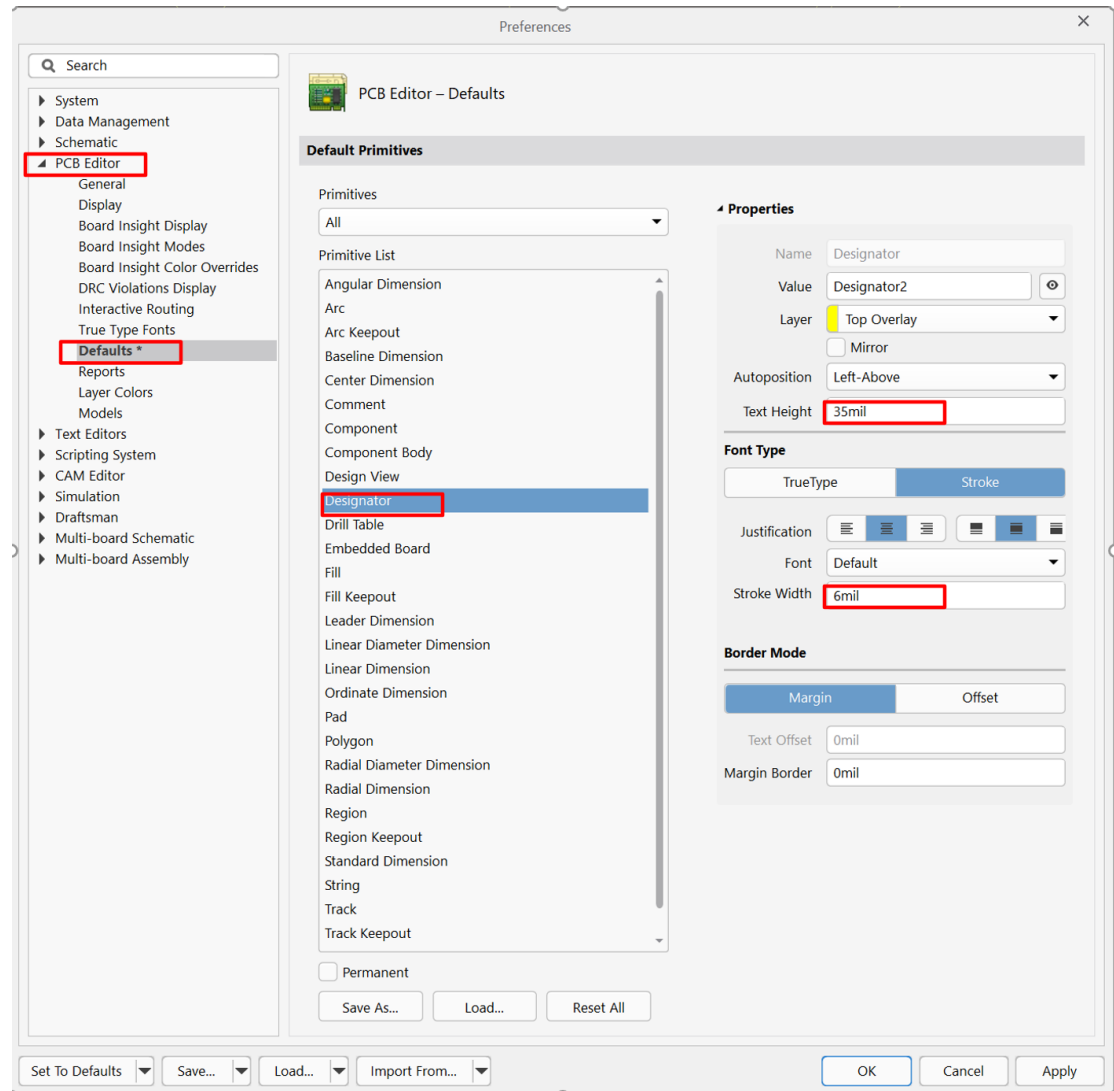7. In the Grids option, select **_Dot Grid_**
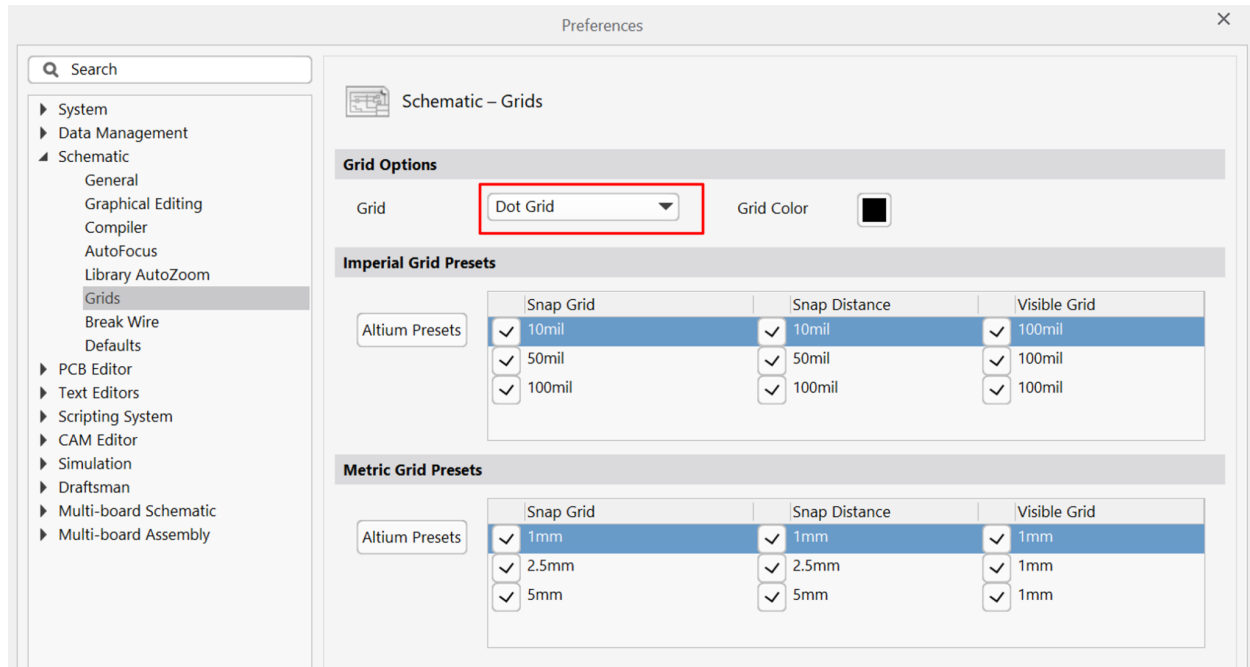
8. Now Select PCB Editor◻General

9. Under Polygon Rebuild, place a check mark for both options.

10. PCB Editor☐Defaults☐Designator, set the Text Height to 35mil and the Stroke Width to 6mil.

11. As you work through the lab manuals, the keyboard shortcuts used in various sections of the document can also be found at the end of this document.

5. Schematic Symbols Overview

An embedded system is a collection of integrated circuits, sensors, and other components that each serve a specific purpose. These parts are soldered onto a printed circuit board that interconnects the components. In order to fabricate the printed circuit board, each component needs a schematic symbol and a PCB footprint added to Altium Designer.

When designing an embedded system, an embedded systems engineer will use a schematic editor to generate a set of schematics for a design. The schematics define how each component is used and how components are interconnected. The first step in generating a set of schematics is to define the schematic symbol for each component.

A schematic symbol is made up of the pins used to physically interface with a part. A schematic symbol may also include other non-electrical information such as the manufacturer name, manufacturer part number, supplier part number, etc.

Each pin on a schematic symbol has two characteristics: pin designator and pin name. The pin designator, or pin number, is used to map the pin of the schematic symbol to a physical pin on the component's footprint (more on footprints later).
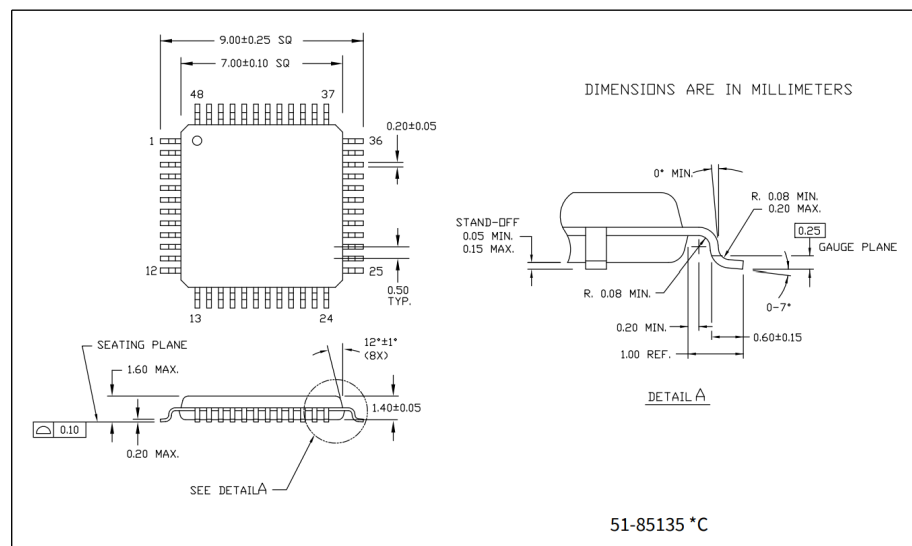
8.1   Package diagrams

Figure 6   48-pin TQFP (7 × 7 × 1.4 mm) package outline, 51-85135

Each physical pin is given a pin name. The pin name is an "arbitrary" string that is used to describe what the pin is used for. Page 15 of the MCU datasheet lists the pin numbers along with their pin names. We will use the 48-pin TQFP packages, so the first two columns of the table give us the pin numbers and pin names.

4   Pinouts

The following table provides the pin list for PSoC™ 4000S for the 48-pin TQFP, 40-pin QFN, 32-pin QFN, 24-pin QFN, 32-pin TQFP, and 25-ball CSP packages. All port pins support GPIO. Pin 11 is a No-Connect in the 48-TQFP.

Table 1   PSoC™ 4000S pin list

| 48-pin TQFP		32-pin QFN		24-pin QFN		25-ball CSP		40-pin QFN		32-pin TQFP	
Pin	Name	Pin	Name	Pin	Name	Pin	Name	Pin	Name	Pin	Name
28	P0.0	17	P0.0	13	P0.0	D1	P0.0	22	P0.0	17	P0.0
29	P0.1	18	P0.1	14	P0.1	C3	P0.1	23	P0.1	18	P0.1
30	P0.2	19	P0.2					24	P0.2	19	P0.2
31	P0.3	20	P0.3					25	P0.3	20	P0.3
32	P0.4	21	P0.4	15	P0.4	C2	P0.4	26	P0.4	21	P0.4
33	P0.5	22	P0.5	16	P0.5	C1	P0.5	27	P0.5	22	P0.5
34	P0.6	23	P0.6	17	P0.6	B1	P0.6	28	P0.6	23	P0.6
35	P0.7					B2	P0.7	29	P0.7		
36	XRES	24	XRES	18	XRES	B3	XRES	30	XRES	24	XRES
37	VCCD	25	VCCD	19	VCCD	A1	VCCD	31	VCCD	25	VCCD
38	VSSD	26	VSSD	20	VSSD	A2	VSS			26	VSSD
39	VDDD	27	VDD	21	VDD	A3	VDD	32	VDDD	27	VDD
40	VDDA	27	VDD	21	VDD	A3	VDD	33	VDDA	27	VDD
41	VSSA	28	VSSA	22	VSSA	A2	VSS	34	VSSA	28	VSSA

An example would be a pin named P0.0. The table indicates to us that pin 28 is connected to IO port pin P0.0. We will need to create a schematic symbol in Altium Designer for use in our design.

In order to create schematic symbols, you will examine the datasheet of a part to determine the number of pins along with the intended purpose of each pin. Once you have located this information, you are ready to create the symbol in an Altium .SchLib file.

6. CY8C4045AZI-S413 Schematic Symbol

One of the parts we will use this semester is the CY8C4045AZI-S413 MCU. This integrated circuit (IC) is a 32-bit ARM Cortex-M0 microcontroller designed by Infineon.
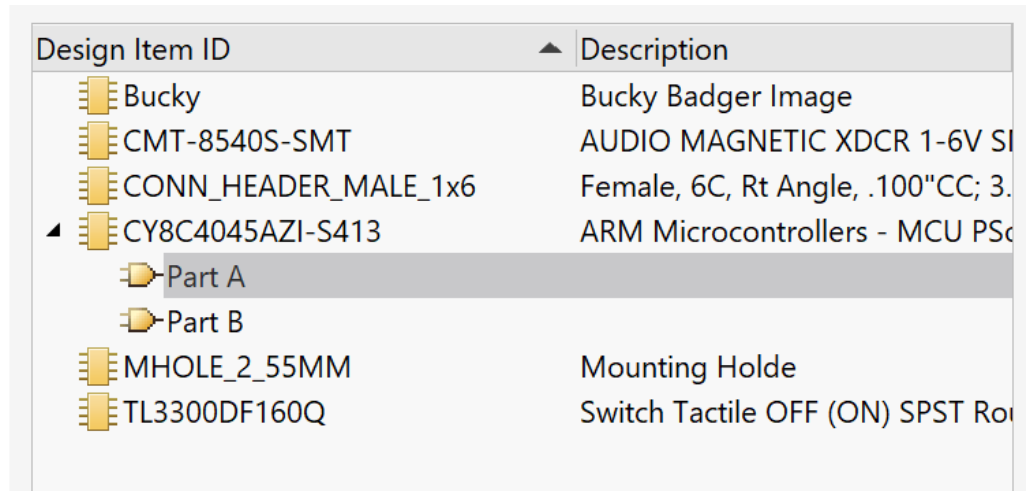
1. From the Project pane, expand the Libraries → Schematic Libraries.
2. Double click on the provided ECE315 library
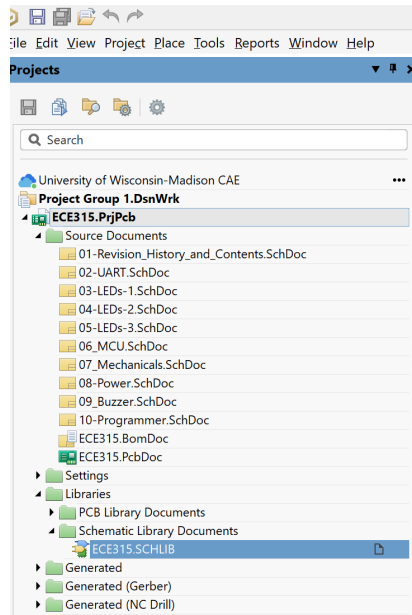
3. At the bottom left of the screen, select Sch Library. This should open a list of parts that you will use to complete your project.
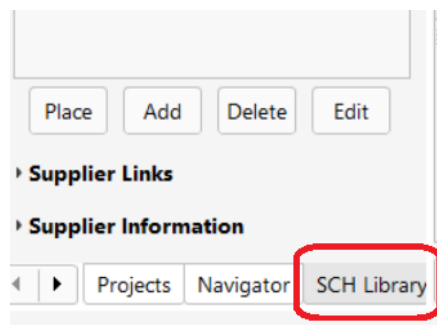
4. Double-click on the part named "CY8C4045AZI-S413" in the Design Item ID Column. This will open a blank canvas where you will create a symbol for this part.


5. Go to Tools→New Part

6. You will notice that there are two sub parts listed under the CY8C4045AZI-S413 part. We will be breaking the MCU into two different "sub" parts. Part A will be used to group all of the IO pins together. Part B will be used to group all of the power and reset pins together.

7. The datasheet for the PSoC4 MCU lists the pin numbers and pin names on page 15.

   You can type **pp** to place a pin. Before you place the pin, press the **TAB** key. This will allow you to change the properties of each pin.

   ● Change the "Designator" to be the pin number.
   ● Change the "Name" to be the descriptive name from the datasheet.
   ● Change the "Electrical Type" to I/O. Altium will auto increment the pin numbers and pin names.
   ● Group all pins of the same port together on the canvas. As an example, P0.0 through P0.7 should be located near each other.
   ● Repeat this for IO ports P0, P1, and P2. The IO pins all start with a capital P.

When you are finished placing pins, hit the **Esc** key to exit the current mode.

When you are finished, your schematic part should look similar to the image below.
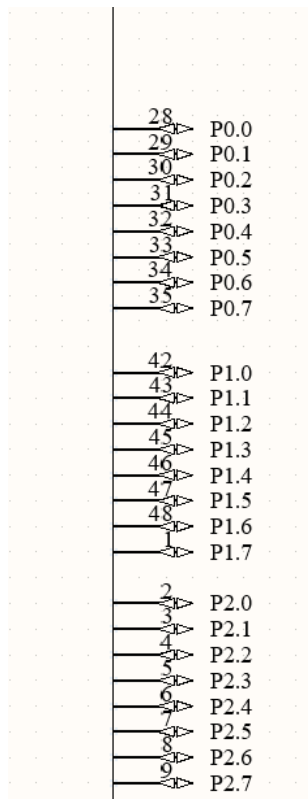
When placing the pins for P3 and P4, you will want to rotate the pins 90 degrees. You can rotate a pin/part/item in Altium by pressing the **Space** key while moving the item.
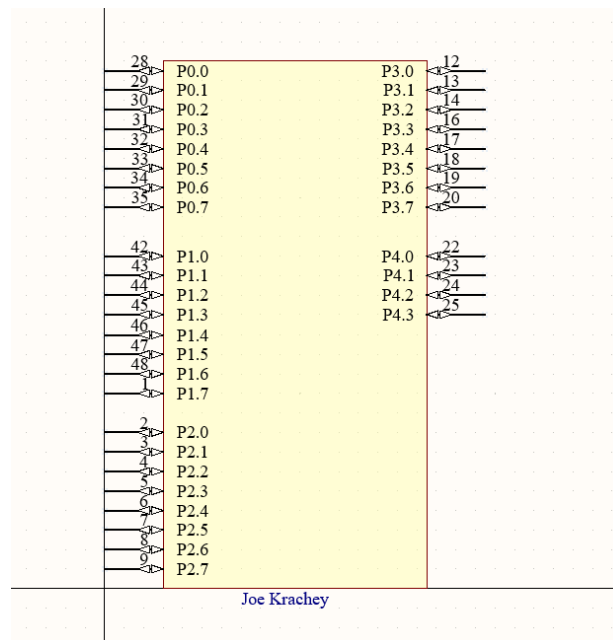
Type **pp** to place a pin. Before you place the pin, press the **TAB** key. This will allow you to change the properties of each pin.

- Change the "Designator" to be the pin number.
- Change the "Name" to be the descriptive name from the datasheet.
- Change the "Electrical Type" to I/O. Altium will auto increment the pin numbers and pin names.
- Group all pins of the same port together on the canvas. As an example, P0.0 through P0.7 should be located near each other.

- Repeat this for IO ports P3 and P4.
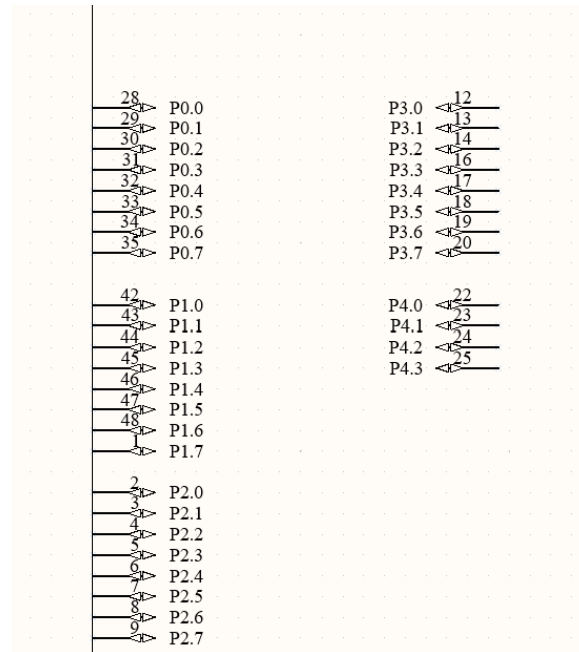
8. Add a border around your part by placing a rectangle. You can type **pr** (place rectangle). Place the rectangle so that all of your pins are touching the edges of the rectangle. After the rectangle has been placed, press **ESC**.

9. Double click on the rectangle and turn on transparency. When you're finished with part A, it should look like the image below:

10. You will now need to complete Part B of the processor. For the XRES pin, select an Electrical Type as "Input" (Before you place the pin, press the **TAB** key).
11. For the remaining pins, select an Electrical Type as "Power".
12. Add a border around your part by placing a rectangle. You can type **pr** (place rectangle). Place the rectangle so that all of your pins are touching the edges of the rectangle. After the rectangle has been placed, press **ESC**.
13. Double click on the rectangle and turn on transparency. When you're finished with part A, it should look like the image below:
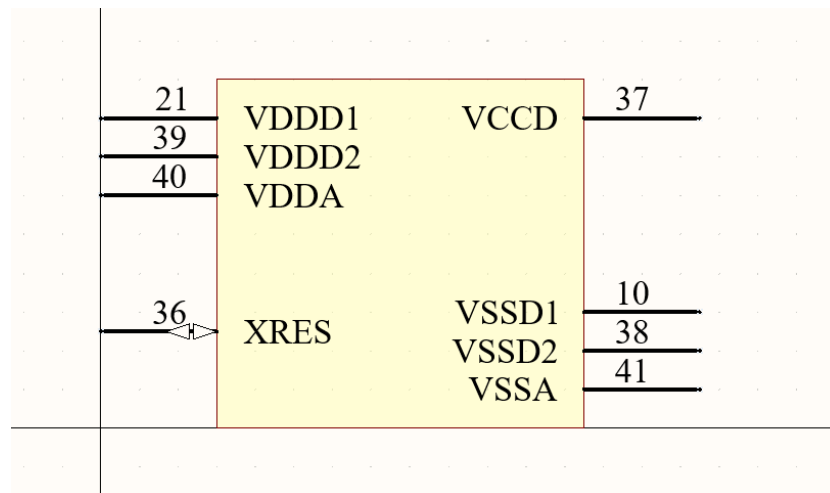
14. In the Sch Library tab in the lower left hand corner, select the CY8C4045AZI-S413 part.
15. Select the Properties Tab in the lower left hand corner.
16. Add a new Parameter (Click on the Add button) called "Drawn By". Set the value to your name and then click on the eyeball symbol until your name is visible.
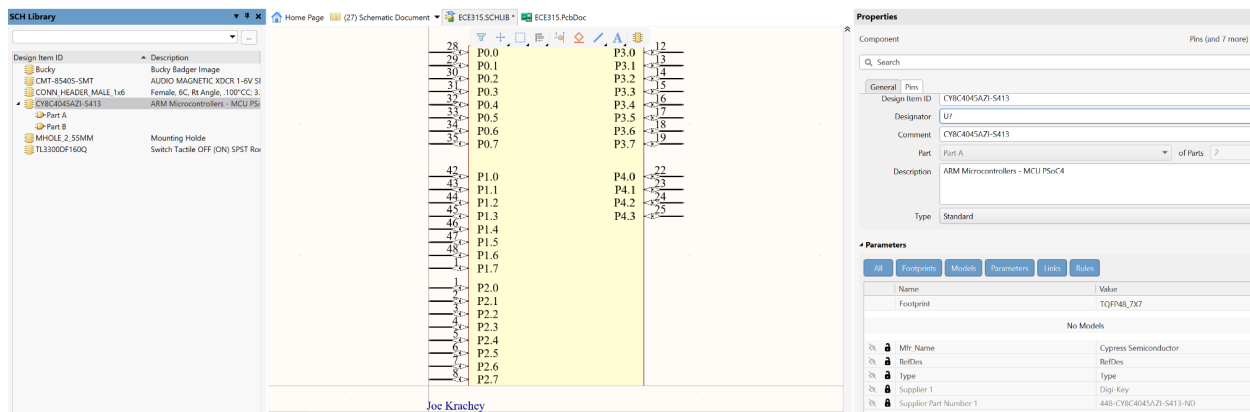
17. **Take a screen capture of your symbol and save it as Symbol_CY8C4045AZI-S413.png**
18. Save the schematic library by using the Save icon in the toolbar or by typing CTRL-S.

7. PCB Footprints Overview

Each schematic symbol needs an associated footprint. The schematic symbol is used to represent the logical function of an integrated circuit. The footprint represents the physical orientation of the pins and must match the exact mechanical information provided by the manufacturer.

The footprint defines physical characteristics such as pin mapping, pin dimensions, and package dimensions. It is imperative that this data be entered very carefully. Having the correct pin mappings ensure that your circuit is connected in the way that you expect. This information is normally supplied near the end of the data sheet. We will need to create a part that matches the supplied mechanical data.

Many components are designed using industry standard mechanical dimensions for a component. Using standard packages will allow us to quickly generate footprints for most of the parts we will use in class.
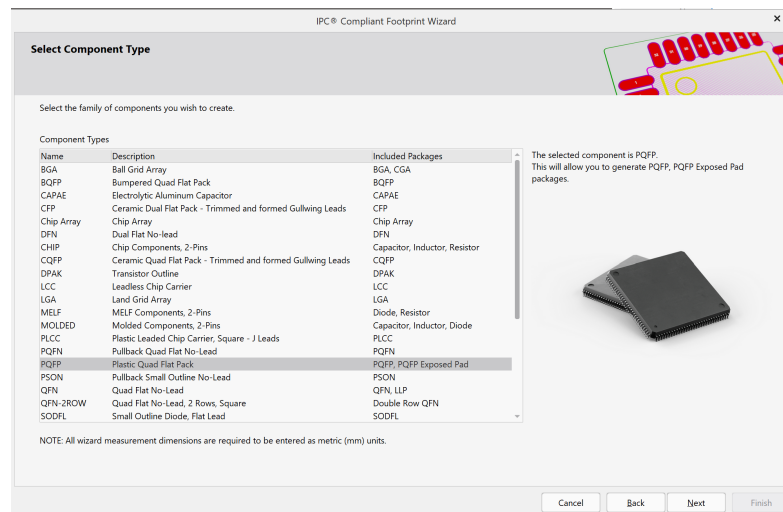
8. CY8C4045AZI-S413 PCB Footprint

Altium provides an IPC Footprint Wizard for generating standard footprints for most components. Use the instructions below to complete the footprint for the CY8C4045AZI-S413.

1. Click on the Project Tab in the lower right corner.
2. Expand Libraries, then PCB Library Documents, then ece315.PcbLib
3. Click on the PCBLib Tab in the lower right corner.
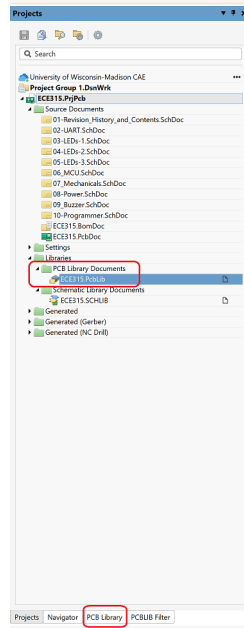
4. From the top toolbar, select Tools→ IPC Compliant Footprint Wizard
5. Select Next from the IPC Wizard.
6. Select PQFP from the list of standard packages.

7. Go to page 44 of the CY8C4045AZI-S413 datasheet. This page lists the mechanical information for the QFP48 you will use in this lab.

8. You will now need to transfer the mechanical information found in the datasheet into the IPC Footprint Wizard. Altium provides a legend of which measurement is related to each input box. You will need to find the indicated measurements from the datasheet.

   The images that follow will show you how to gather the information from the mechanical information found in the datasheet.
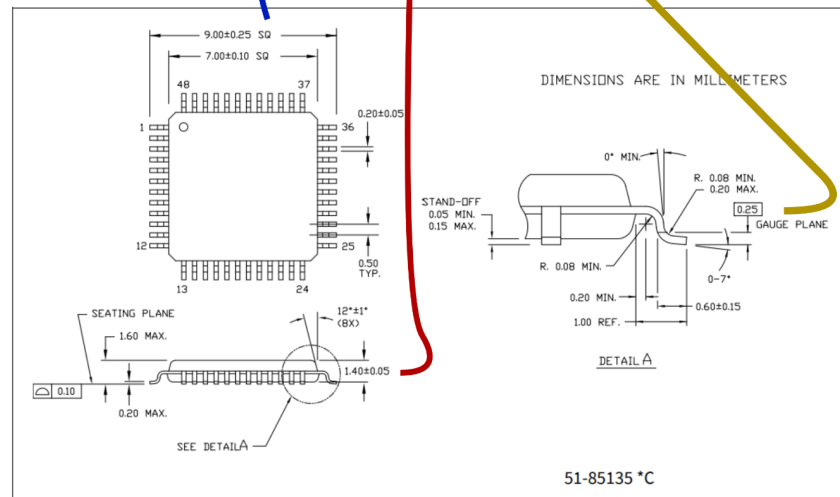
8.1 Package diagrams

Figure 6 48-pin TQFP (7 × 7 × 1.4 mm) package outline, 51-85135

9. Click on the Next button.
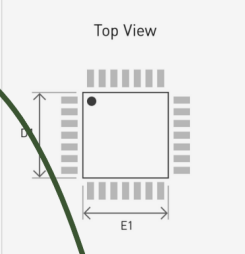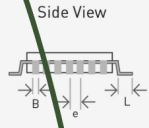10. Complete the second page of mechanical data.

IPC® Compliant Footprint Wizard

PQFP Package Pin Dimensions
Enter the required package values.

Lead Width Range (B)	Minimum	0.18mm
	Maximum	0.2mm
Lead Length Range (L)	Minimum	0.6mm
	Maximum	0.75mm
Pitch (e)		0.5mm
Body Width Range (E1)	Minimum	6.9mm
	Maximum	7.1mm
Body Length Range (D1)	Minimum	6.9mm
	Maximum	7.1mm
Number of pins (E)		12
Number of pins (D)		12

This package has 48 leads

Top View

Side View

☑ Generate STEP Model Preview   Cancel

8.1   Package diagrams

DIMENSIONS ARE IN MILLIMETERS
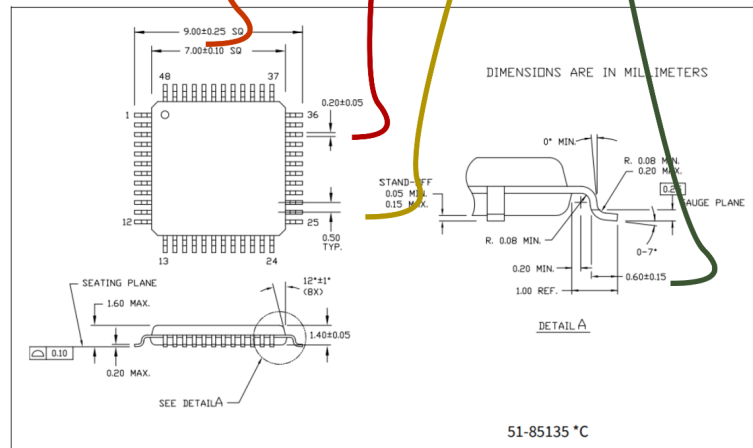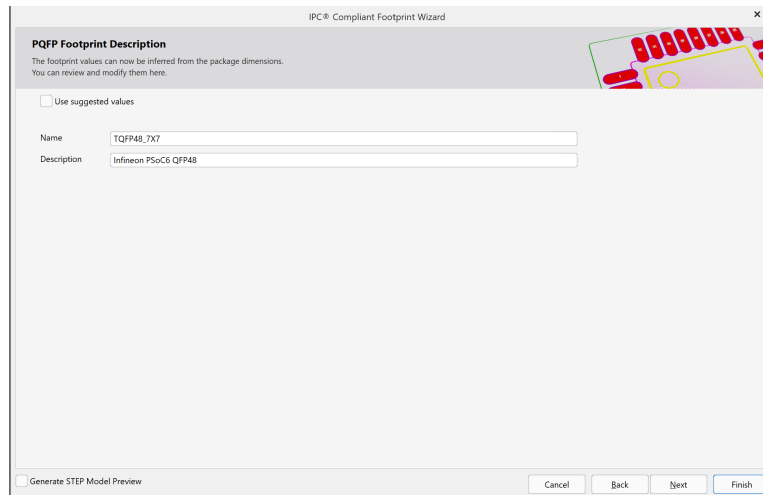
51-85135 °C

Figure 6   48-pin TQFP (7 × 7 × 1.4 mm) package outline, 51-85135

11. Click on the Next button nine (9) times until you reach the SOT23 Description Page. Change the name of the footprint to be TQFP48_7X7.

12. Click the Next button.
13. Select the option to add the footprint to the current PcbLib
14. Click on Produce 3D/Step Model

15. Click on the Next button.
16. Click on the Finish button.

17. The resulting footprint will be shown. You can **zoom** in/out in Altium by **holding down the scroll wheel on the mouse and pulling/pushing** the mouse.
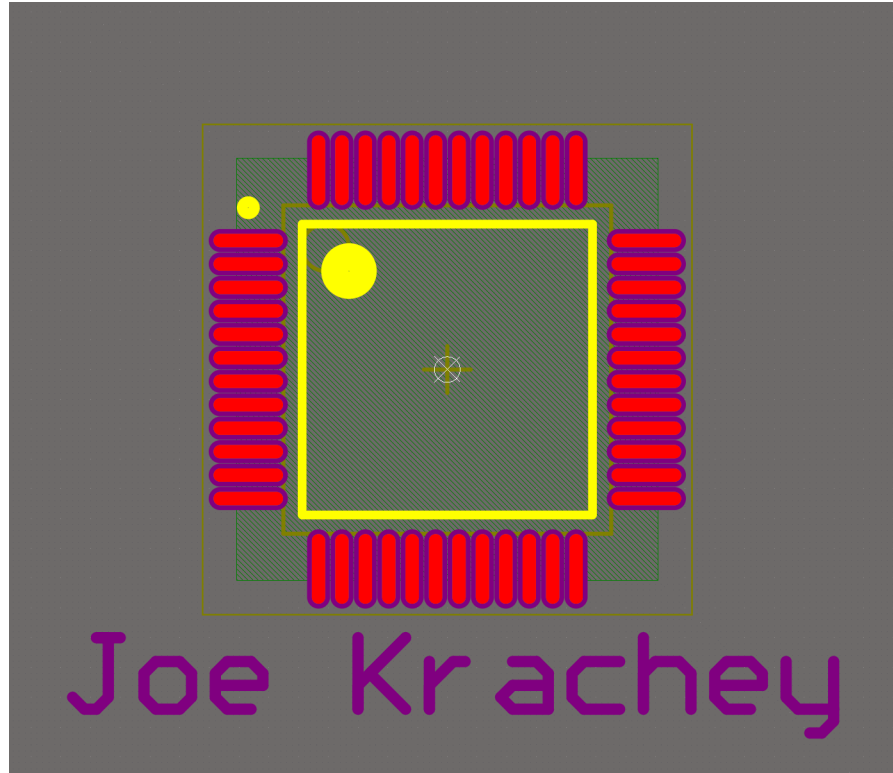
18. Place a string (**ps**) with your name under the part. The string's layer should be set as Mechanical 10. You should fill in the rest of the properties to match the image below (add your own name of course).

19. **Take a screen capture of your symbol and save it as PCB_CY8C4045AZI-S413.png**

20. Save the PcbLib.

9. Non-IPC Compliant Footprints

Some components cannot be generated using the IPC footprint wizard. In these situations, you will need to manually draw the footprint yourself. The following instructions will help you to create a footprint for a magnetic buzzer.

1. Open the CMT-8540S-SMT-TR datasheet.
2. Click on the Project Tab in the lower right corner.
3. Expand Libraries, then PCB Library Documents, then ece315.PcbLib
4. Click on the PCBLib Tab in the lower right corner.
5. Double click on the CUI_CMT_8540S-SMT footprint
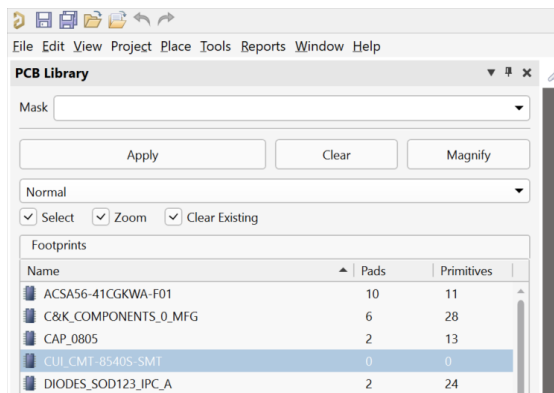
6. In the blank footprint canvas, place a pad using **pp**. Before you place the pad, hit the **Tab** key.

You will need to modify the following properties

- Designator: P
- Layer: Top Layer
- Shape: Rectangular
- X/Y 2.5mm x 2.5mm
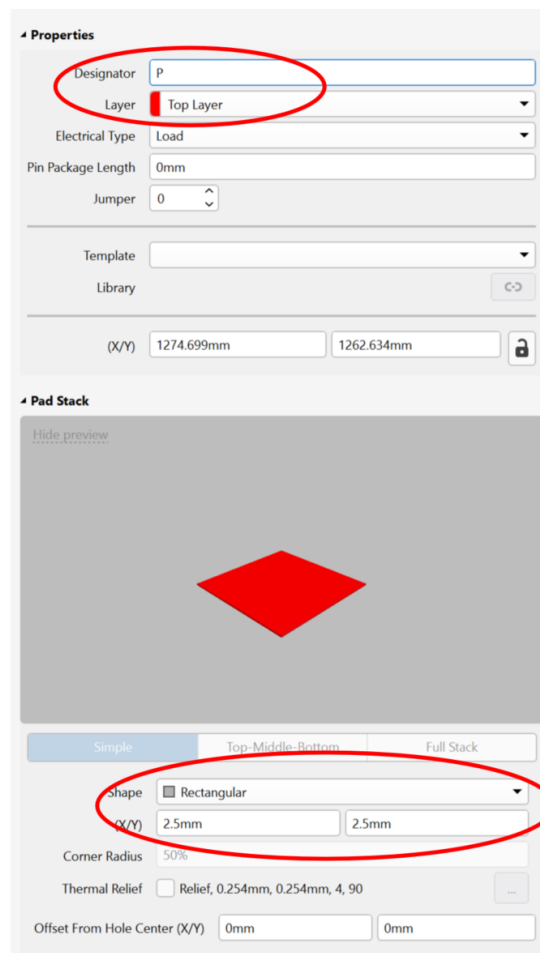
7. Place the pad below and to the right of the (0,0) axis.
8. Create a 2nd pad that is identical, but set the designator to **N**.
9. Place the pad above and to the right of the(0,0) axis.
10. Create a 3rd pad that is identical, but set the designator to **M1**.
11. Place the pad below and to the left of the (0,0) axis.
12. Create a 2nd pad that is identical, but set the designator to **M2**.
13. Place the pad above and to the left of the (0,0) axis.
14. Press **ESC** to stop placing the pads.

15. Press **q** to toggle between an X/Y coordinate in mils or mm. You will want to set the grid in mm.
16. You now need to place the pads in the correct locations based on the information from page 2 of the buzzer's datasheet.
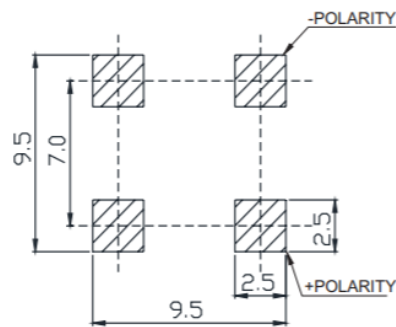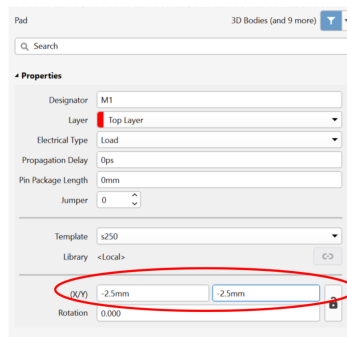

Recommended PCB Layout
Top View

We can see from this drawing that the pads should be centered at the following coordinates

P   (3.5mm, -3.5mm)
N   (3.5mm, 3.5mm)
M1   (-3.5mm, -3.5mm)
M2   (-3.5mm, 3.5mm)

Double click on each pad and set the coordinates to match these values.

17. Place a string (**ps**) with your name under the part. The string's layer should be set as Mechanical 10. You should fill in the rest of the properties to match the image below (add your own name of course).

18. Take a screen capture of your symbol and save it as PCB_CMT-8540S-SMT-TR.png

19. Save your PcbLib

20. Adding Footprints to Schematic Symbols

In order to design a printed circuit board (PCB) the schematics for a design must have footprints associated with every part in your design. The following instructions will help you to add the footprints you have created with their corresponding schematic symbols.

1. Click on the Project Tab in the lower right corner.
2. Expand Libraries, then Schematic Library Documents, double click ece315.SchLib
3. Click on the SchLib Tab in the lower right corner.
4. Select the **CY8C4045AZI-S413**
5. Under the symbol of the part, click on the Add Footprint button.
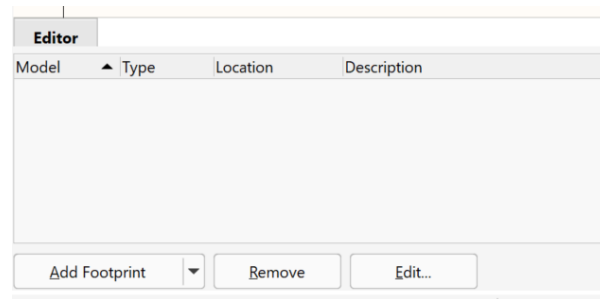
6. Select the Browse Button from the pop-up window.
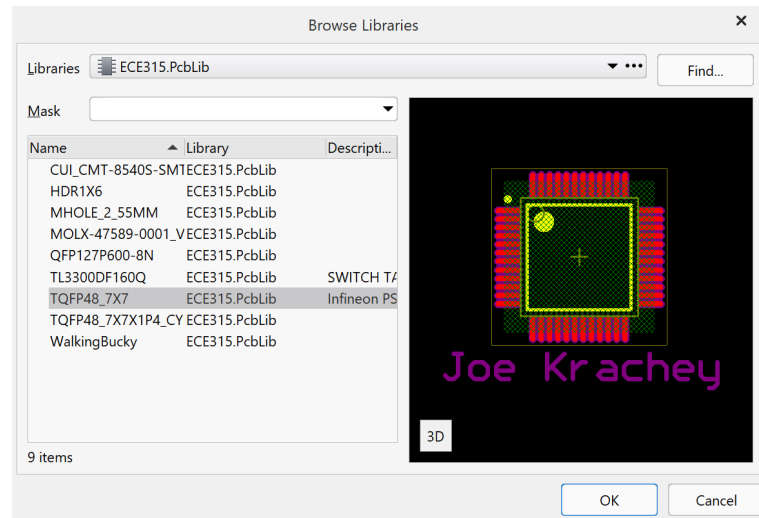
7. Select the TQFP48_7X7 footprint from the ece315.PcbLib.
8. **Repeat this process for the CMT-8540S-SMT**

21. Lab 1 Commentary

Before you can complete the schematics for a design, the components used in that design must be created in your CAD tool. This might seem like a long and tedious process (it is), but luckily most companies employ a parts librarian that creates and maintains parts. This helps an embedded systems engineer to focus their efforts and develop products rather than reading mechanical drawings from datasheets.

If you're not lucky enough to work for a company that has a dedicated parts librarian, many parts have symbols and footprints that can be downloaded for free from sites like Digikey and Octopart. Starting in Lab 2, we will make use of Altium's Manufacturer Part's Vault for all of our additional parts. This will help to speed up the design process.

22.   Schematic Library Keyboard Shortcuts

Action	Short Cut
Place Pin	**pp**
Place Rectangle	**pr**
Rotate	**Space**
Exit Current Mode	**Esc**
Modify Properties of Selected Object	**Tab**
Zoom	**Mouse Scroll Wheel**

23.   PCB Library Keyboard Shortcuts

Action	Short Cut
Place Pad	**pp**
Place String	**ps**
Rotate	**Space**
Toggle Coordinate Grid Units	**q**
Exit Current Mode	**Esc**
Modify Properties of Selected Object	**Tab**
Zoom	**Mouse Scroll Wheel**