



Electrical and Computer Engineering

ECE315

Lab 1:

Schematic Symbols and PCB Footprints

1. Pre-Lab

In this Lab 1, you will create the schematic symbols and PCB footprints for several of the components that you will use in later labs. This will require you to gather information from component datasheets and enter it correctly into Altium Designer.

2. Getting Started

1. Download ECE315.zip folder from the course website.
2. Note that there are a number of different files in the folder that are used by Altium Designer.
3. If you want to show the file extensions in a Windows folder use the following:
 - a. Start Windows Explorer, you can do this by opening up any folder.
 - b. Click **Organize**.
 - c. Click **Folder and search options**.
 - d. Click the **View** tab.
 - e. Scroll down until you notice **Hide extensions for known file types**, un-check this line by clicking the check box.
Note To hide file name extensions, check this line.
 - f. Click **OK**
4. Open the Altium Designer Project by clicking on the **ECE315.PrjPcb** PCB Project File.
5. Explore the file structure in the Projects pane. You can double click on each entity to open it up.
6. Double click on **01 Revision History and Contents.SchDoc** in the Projects pane. You will see the Revision History and a Table of Contents.
 - a. Enter your name in the **[your name here]** text box above the title.

3. Creating a Symbols for MCU and Capacitive Touch Sensor

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.

A schematic symbol represents the logical organization of the pins used to interface with a component. You will create a schematic symbols for a ST Micro STM32F303K8T6. The MCU will be programmed in a later lab and be used to control the embedded system.

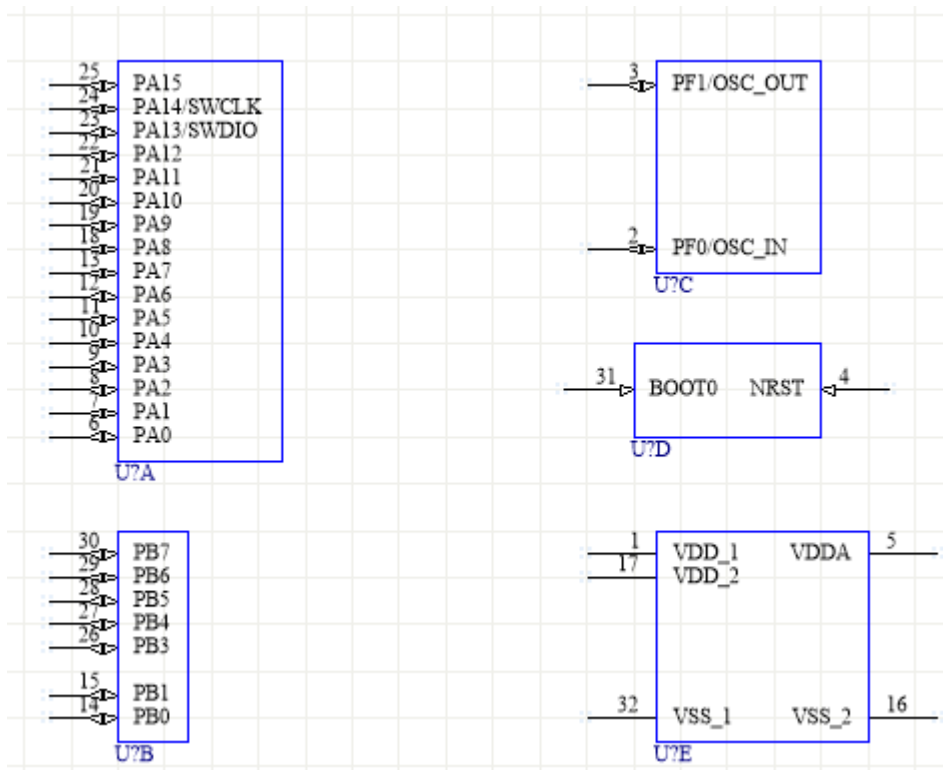
Use the following links to see how to create a schematic symbols.

[Schematic Symbols](#)

[Logically Organized Schematic Symbols](#)

NOTE: There are some pre-populated parts in the schematic and PCB libraries. Do not edit those parts until you are asked to do so.

When you have finished, your schematic symbols should similar to the images below.



4. IPC Compliant Footprint Wizard

Each schematic symbol will also need an associated footprint. The schematic symbol is used to represent the logical function of an integrated circuit. In some cases, the schematic symbol will have a logical arrangement of pins to make the schematics more readable (like we saw above, pins for certain functions are grouped together). The footprint represents the physical "pinout" (orientation of the pins to the part) of the part itself and must match the description given by the manufacturer.

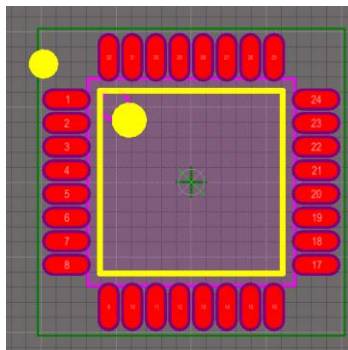
The footprint defines physical characteristics such as pin mapping, pin dimensions, and package dimensions. It is imperative that this data be entered very carefully as this is one of the biggest reasons that PCB designs don't work when you start testing them. Having the correct pin mappings ensure that 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.

Altium provides a footprint wizard for generating the most common package types. This will allow us to quickly and accurately build footprints for our parts.

The following link to create footprints for the [STM32F303K8T6](#).

[IPC Compliant Footprint Wizard](#)

When you have finished with the wizard, your footprints should look like the figure below.



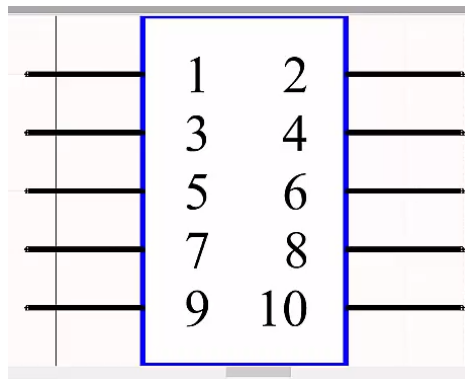
5. Drawing Parts with Custom Footprints

Many parts including connectors/headers, non-standard ICs, or wireless modules, have footprints that cannot be generated using the IPC component wizard. For these types of parts, you will have to draw your own custom footprint using the PCB library editor.

Use the following link to create a footprint for dual row, 10 position socket (61301021821) from Wurth Electronics.

[Custom PCB Footprints](#)

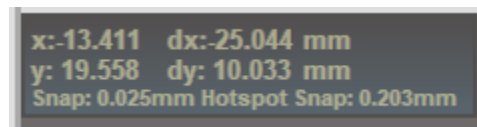
This socket will be used to connect the MCU programmer. You will also need to create a schematic symbol for the programmer. The symbol should be drawn to match the symbol found in the Custom PCB footprints video.



After you have created the schematic symbol, you will need to create the associated footprint, named CONN_ST_PROGRAMMER.

For your part, identify pin 1 by using a square pad. You will also identify pin 2 by drawing a triangle on the Top Overlay.

NOTE: Some parts are listed in the data sheet as mils and others as mm. They ARE NOT the same thing. 1 mil = 1/1000th of an inch. You can toggle between the current units of measurement by typing 'q' when you are in the PCB library editor. The upper left hand corner should display information on the X and Y coordinates so you can see what the current measurement unit is.



6. Adding Supplier Links and Footprints

The following link will show you how to add the supplier links and PCB footprints to ece315.SchLib for IC_STM32F303K8T6 and CONN_ST_PROGRAMMER. Filling out the supplier information will allow Altium to generate a build of materials or BOM that you will use to order parts for you design.

[Adding Supplier Links and PCB Footprints](#)

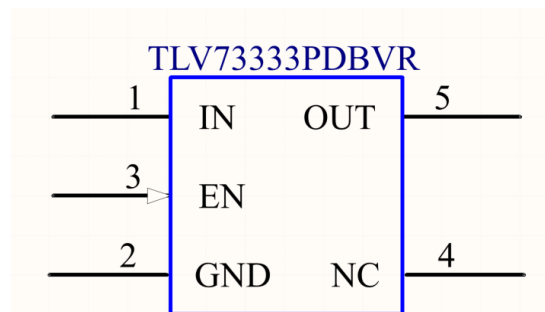
7. Pre Lab: What to Turn in

File Name	Description
Lab1_Altium.zip	Altium Project ZIP file. All parts in covered in the pre-lab [STM32F303K8T6 and 61301021821] should have completed supplier links, schematic symbols, and PCB footprints. Delete the History folder before creating the ZIP file.

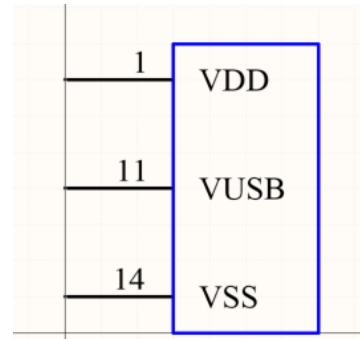
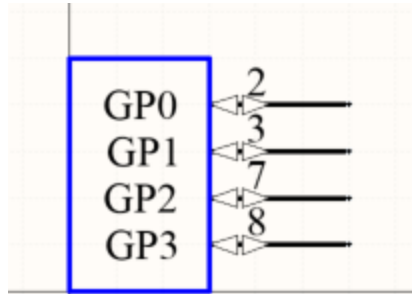
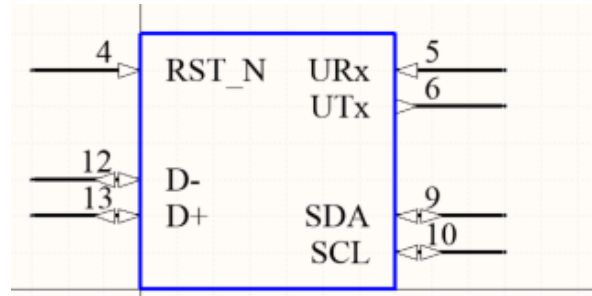
8. Lab Work

a. Create Schematic Symbols

Create a schematic symbol for a TLV73333PDBVR Linear Regulator.



Create a schematic symbol for a MCP2221A-I/SL USB-to-UART bridge. It should be logically organized and have the following 3 sub components.



b. Create PCB Footprints

Create a PCB footprints for the following components

- IC_ TLV73333

Create a footprint for the linear regulator using the IPC Compliant Footprint Wizard.

- IC_ MCP2221

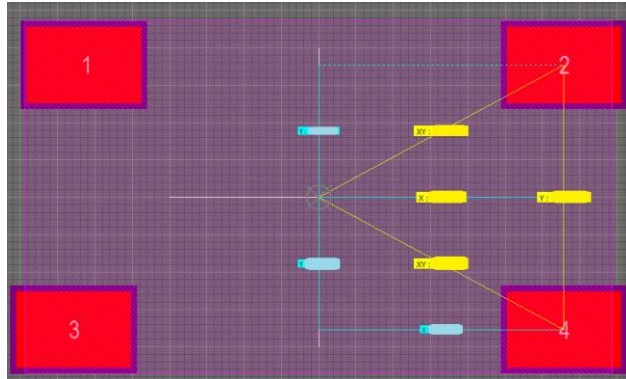
Create a footprint for the USB-to-UART bridge using the IPC Compliant Footprint Wizard. You MUST use the 14-pin SOIC version of the part.

- SW_6mm_6mm_SMT

Create a footprint for a surface mounted push button [PTS645SH50SMTR92LFS]. The schematic symbol has already been completed for you. You need to complete the footprint named SW_6mm_6mm_SMT. Add the supplied 3D footprint [PushButton_6mmx6mm.stp] to the part. You may have to rotate the shape using this dialog when inserting the 3D shape.

Once you have finished drawing your component, measure the distance between the center of the part and pads 2 and 4. Also measure the distance between the center of pads 2 and 4.

You can measure the distance by pressing CNTL-M. The measurements should be displayed in mils. Once the measurements are visible, use Windows Snipping Tool to grab a screen capture. Save the screen capture as push_button.png. The image below shows you what measurements should be displayed.



c. Add PCB footprints to the Following Schematic Symbols

- TLV73333PDBVR
- IC_MCP2221
- SWITCH_6mm_6mm_smd

d. Add Supplier Links

Add Digikey supplier links for the following components. **Be sure to select the manufacturer specified.** If there are several components that match the description, choose the cheapest option. The schematic symbol and PCB footprint have already been supplied for the resistors and capacitors.

Schematic Part	Manufacturer	Description
RES_0805_K_001.00	Yageo	RES SMD 1K OHM 1% 1/8W 0805
RES_0805_K_010.00	Yageo	RES SMD 10K OHM 1% 1/8W 0805
RES_0805_R_330.00	Yageo	RES SMD 330 OHM 1% 1/8W 0805
CAP_0805_P_004.30	AVX	CAP CER 4.3PF 100V NPO 0805
CAP_0805_U_010.00	Samsung	CAP CER 10UF 16V X5R 0805
IC_TLV73333	Texas Instruments	IC REG LINEAR 3.3V 300MA SOT23-5
IC_MCP2221	Microchip	IC USB TO I2C/UART 14SOIC
SW_6mm_6mm_SMT	ITT / C&K Components	SWITCH TACTILE SPST-NO 0.05A 12V

When you have found all your parts, complete the components.xlsx found on the course website. Fill in the manufacturer and supplier part numbers for your components.

9. Post Lab: What to Turn in

All files below inside a ZIP file named "LastnameFirstname.zip"

File Name	Description
LastName_FirstName_Lab1_Altium.zip	Altium Project ZIP file. All components should have completed supplier links, schematic symbols, and PCB footprints. Delete the History folder before creating the ZIP file.
LastName_FirstName_Lab1.pdf	Generate a PDF with the images of the following: Pushbutton footprint. The image should display the required measurements asked for in the lab document. TLV73333PDBVR Schematic Symbol TLV73333PDBVR Footprint MCP2221 Schematic Symbol MCP2221 Footprint
LastName_FirstName_components.xlsx	Completed Excel sheet with manufacturer Part Number and Supplier Part Number