# ECE315 Introductory Microprocessor Laboratory

# Lab 2

## Schematic Capture

## 1. Introduction

In Lab 1, you created schematic symbols and PCB footprints for several components.  In this lab, you will place and connect those components in a series of schematic sheets. Schematic sheets represent the components found in your embedded system and how they are connected to one another.  Embedded systems engineers examine a product in order to select the components used in a design.  In this lab, the components have been selected for you and you will be given guidance as to how to interconnect the various components.

## 2. Placing Components for 06_MCU

You will begin placing the PSoC6 on on page 06-MCU.SchDoc.

a) Open 06-MCU by double clicking on it in the Project pane.
b) Set the grid size to be 50mils.  View→Grids→Set Snap Grid or type '**vgs**'.
c) Click the Panels tab in the lower right corner of Altium
d) Select Components
e) Components can be selected from the ece315 library.  A component is placed by double clicking on it and dragging it into the schematic.

Drag the following components to the area indicated above.  If a part needs to be rotated, press the **Spacebar** until it is in the desired position.

| Part | Description |
|------|-------------|
| CY8C4045AZI-S413 | PSoC4 Microcontroller |
| TL3300DF160Q | Tactile Switch |

f) For the remaining parts on this page, we will make use of parts supplied by the Altium Vault.  Click on "Panels" in the far right hand corner, then select "Manufacturer Part Search".  **In order to place parts from the Altium vault, you will need to register an email address with Altium Live.**

g) Place a 1uF capacitor (Part# **C1206C105K3RACTU**) on the schematic page. Type the part number into the manufacturer part search. When the part is listed in Altium, right click on the part and select "Place".

Double click on the part you placed. This will bring up the properties menu for that part. Click on the eye icon for value so that the value for the capacitor is visible. You will also want to hide the part number by clicking on the eye icon for the "Comment" until it is grayed out.
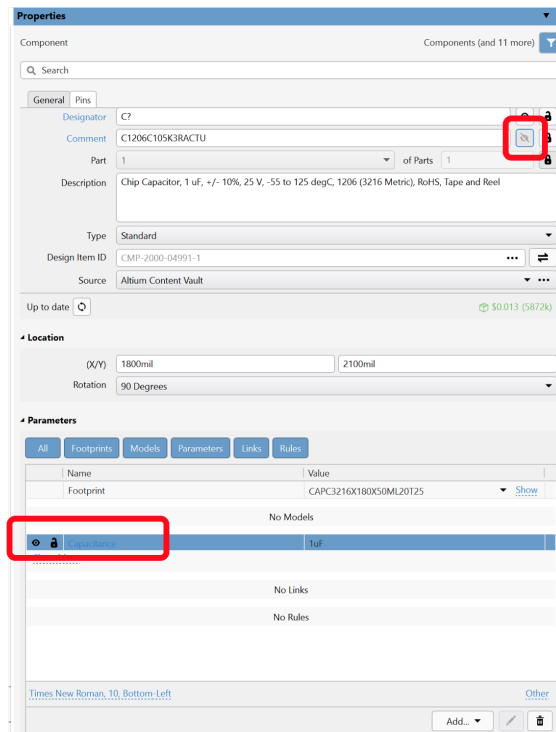


Click on the 1uF and drag it so it is just under the reference designator (C?). It should look like this when you are done.
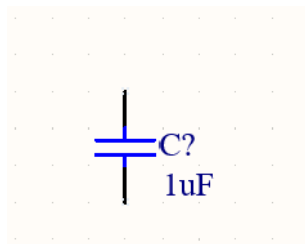
h) Copy the capacitor that you have placed (CTRL-C) and place them near the Power pins of the MCU.
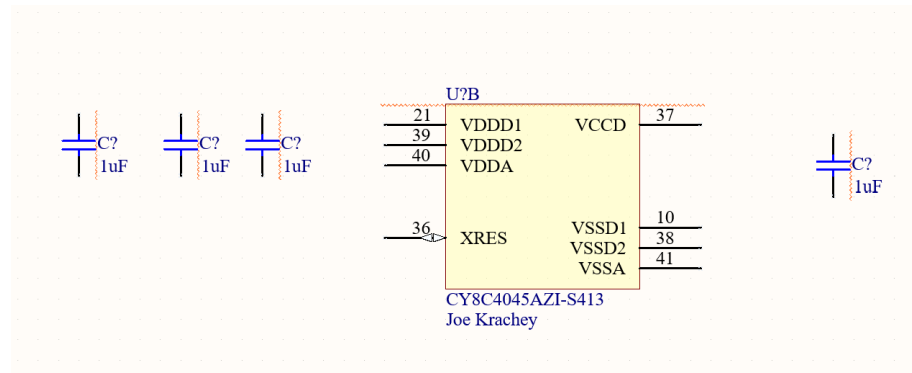


i) Place a 10K 1206 resistor (Part # **RC1206FR-0710KL**) near the push button. Hide the part number and make the resistance visible in the same way you did with the capacitors.
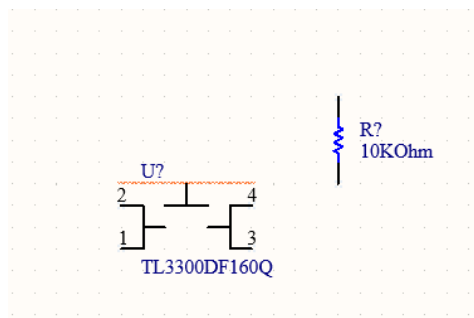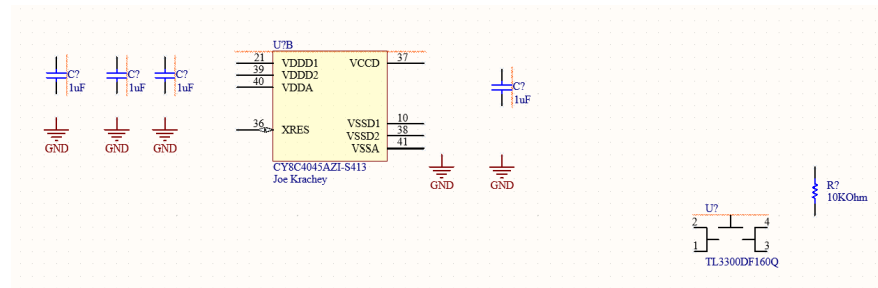


# 3. Placing Power Ports

Now that you have placed components for the MCU, you will need to place symbols that represent GND.

a) Select the Power Port Icon near the top of Altium or by typing **po**.
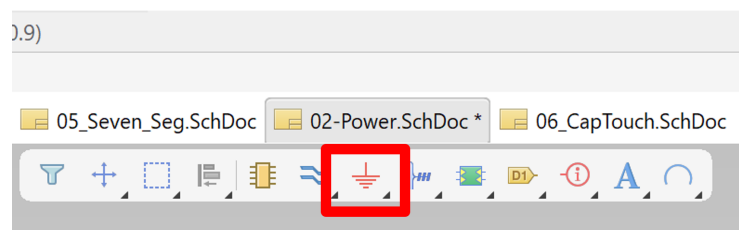
You will place the GND symbols as shown below.



b) Place a 3.3V 4 power ports as shown below.  Type **po,** then press the Tab key to set the properties of the symbol.  In the Properties pane, change the name to 3.3V and change the Style to "Arrow".  You can rotate the power ports using the SPACE bar.
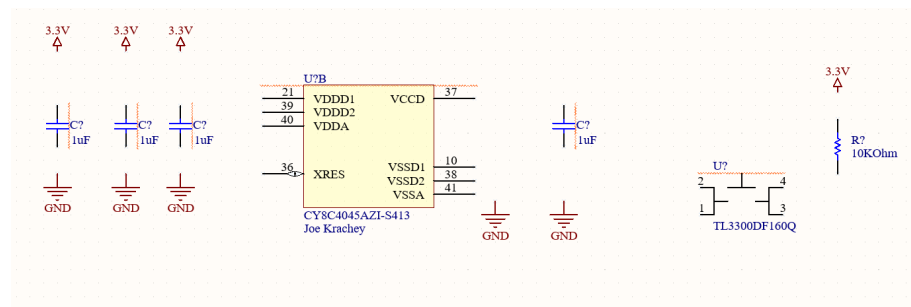


# 4. Adding Wires

Now that the required parts for the boost regulator have been placed, you will need to connect the components together.  Connections in Altium are called a **wire**.

a) You can place a wire by selecting the wiring tool from the command ribbon or by typing **pw** (place wire).
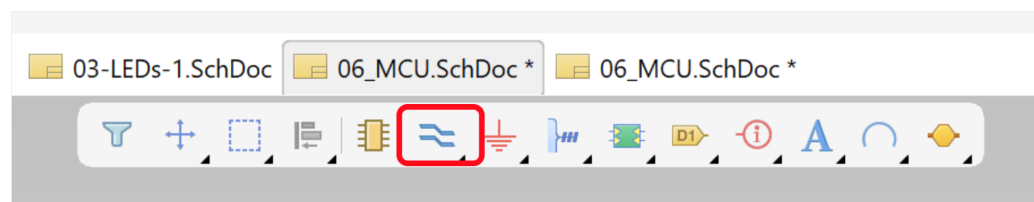


b) You can add wires by left clicking on a pin.  You can then route the wire to its desired location.  You can change the direction of a wire by 90 degrees by left clicking at the desired location.

c) Use the wiring tool to connect the capacitors to the MCU as shown below.
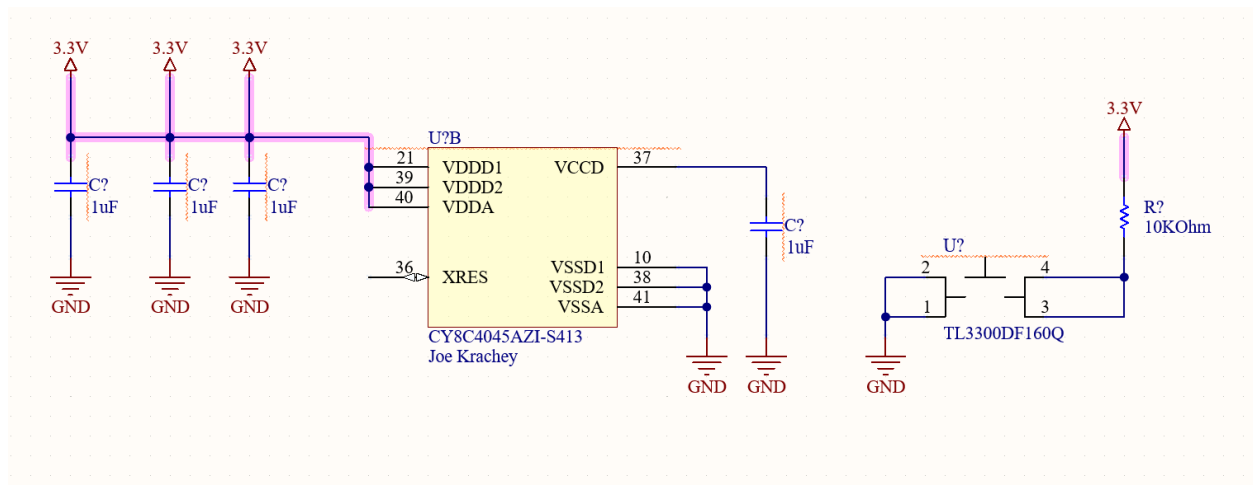


d) A good question to ask is what are the three capacitors to the right of the MCU symbol needed for. The answer is that they are bypass capacitors. Bypass capacitors help to filter out high frequency noise that can lead to the malfunction of an embedded system. **Every IC power supply pin** should have a bypass capacitor (in some cases more than 1) placed **within 5mm** of each pin to help improve system reliability.

# 5. Adding Net Names

It is an industry best practice to give all wires in a design a net name.  Naming wires allows a PCB designer to detect schematic errors during the PCB layout process.  **<u>ALL</u>** nets in your design must have a net name.

a) Identify which wires connected to the MCU currently do NOT have net names.  A wire that is connected to a power port is automatically given a net name that matches the power port.
b) You can add net names by typing **pn** (put netname).
c) The **Tab** key will allow you to change the netname. Netnames are any arbitrary string, but give them a meaningful name such as **VCCD**.
d) You can place the netname by right clicking where you want the netname located.
e) When finished, press **Esc**.  Your schematic should look similar to the following image.
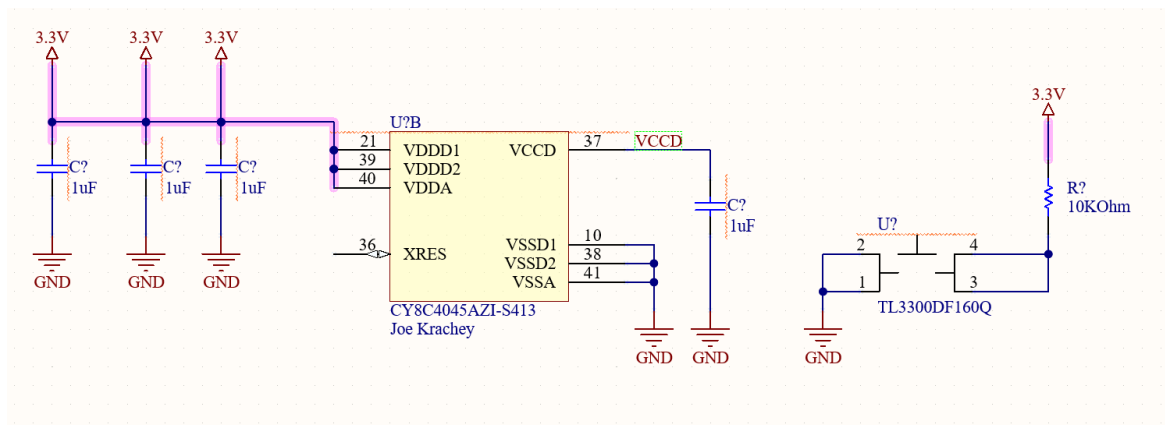


f)  An offsheet connector is a signal that can be referenced on any schematic page of the project.  Wires connected to an offsheet connector DO NOT need a netname.  The name of the offsheet connector will be assigned as a netname automatically.

Place an offsheet connector (**pc**) and change its name to BTN_IN by pressing TAB prior to placing the connector.
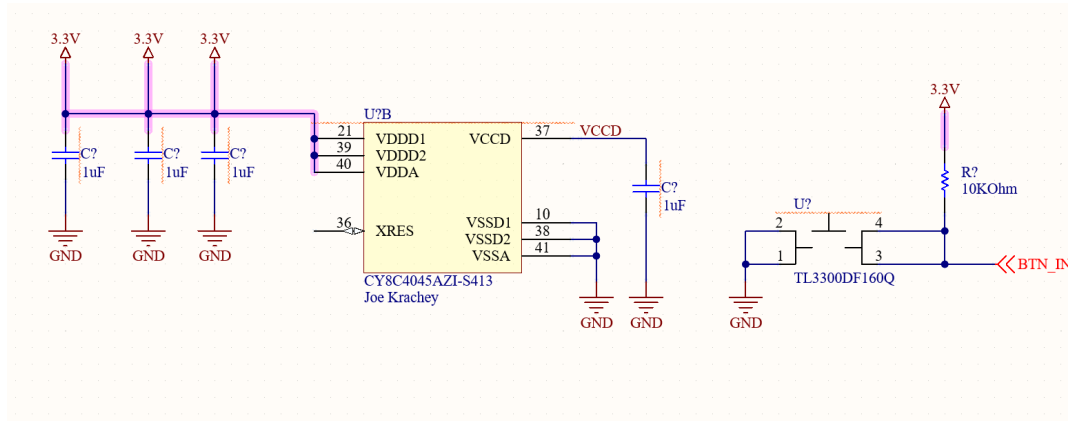


# 6. LED Pages

The Whack-A-Mole board will need 12-LEDs.  These LEDs are a visual indication of how much time is remaining.  The image below shows how the LEDs will be arranged on the PCB.
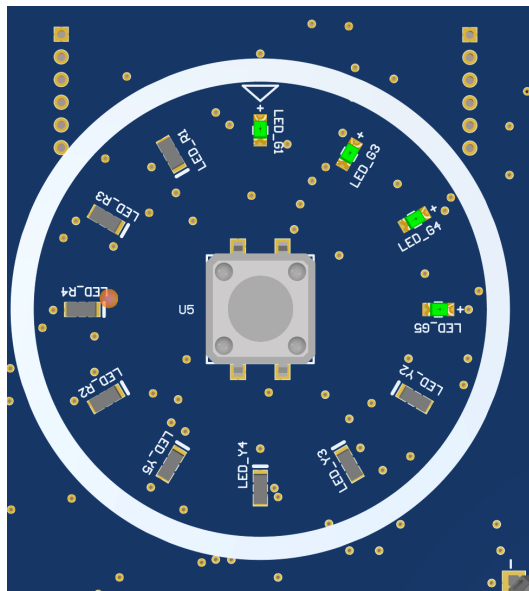
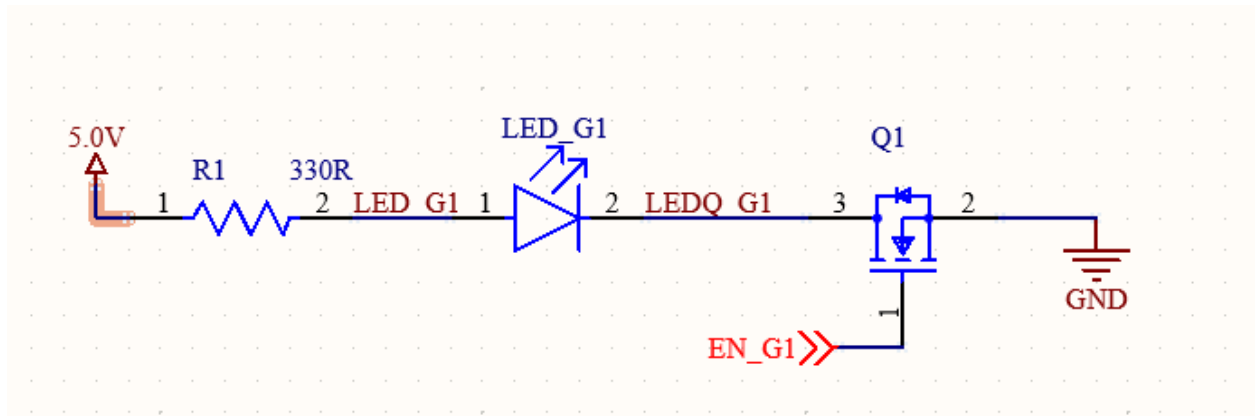Each LED is controlled by an IO pin that turns the LED On/Off. The image below shows how to correctly control the LEDs using an N-Channel MOSFET.



You will need to complete the 3 LED schematic pages using the part numbers supplied below. Each page should have 4 LEDs of the specified color.

| Part Number | Description |
| --- | --- |
| CRCW1206330RFKEA | 330Ω Resistor |
| SML-LX1206GC-TR | Green LED |
| SML-LX1206YC-TR | Yellow LED |
| SML-LX1206IC-TR | Red LED |
| 2N7002-7-F | MOSFET N-CH |

- Use the part numbers above to find the parts in the Manufacturer Part Search.
- All nets MUST have netnames. Look at the example above.
- All Off-Sheet Connectors must have a unique name.
- Make sure that the source of the FET (pin 2) is connected to GND.

If you use the naming convention above, you can quickly duplicate a circuit and have the net names auto-increment. You can do this by selecting all of the parts and netnames. Then press SHIFT and drag the selected parts.

ECE315: Introductory Microprocessors Laboratory

# 7. Power Supply

The MCU requires a 3.3V supply voltage.  This voltage will be generated using a [Linear Regulator](#).  Complete the schematic page 08-Power using the specified components below.  You can find details on how to complete the linear regulator schematics by examining the first few pages of the regulators [datasheet](#). Use the part numbers below to find the parts in the Manufacturer Part Search.  You have already used the same capacitor in 08-MCU.  You can copy and paste from that schematic sheet.

Add 3 test points to the design.  Connect 1 test point to each of the following power ports: 5.0V, 3.3V, and GND.

| Part Number | Description |
| --- | --- |
| TLV1117LV33DCYR | Linear Voltage Regulator |
| C1206C105K3RACTU | 1 uF Capacitor |
| 5019 | Test Point, 1 Position SMD |

# 8. Complete MCU Connections

All of the schematic pages for the design are completed with the exception of 08-MCU.  You will need to examine the [PSoC6 datasheet](#) to determine which MCU pins to connect the offsheet connectors to.  Use the following guidelines to connect the external devices to the MCU.

- Go to page 17 of the PSoC6 Datasheet to determine which interfaces each pin supports.
- Move the offsheet connectors so that they are close to the MCU pin they are going to connect to.  Feel free to rotate the connectors as needed.
- Connect the SWD_CLK and SWD_DATA pins first.  These pins are used to program the MCU, so they are the most important to place correctly.
- Connect UART0_TX and UART0_RX to scb0.uart capable pins.

- Connect UART1_TX and UART1_RX to scb1.uart capable pins.
- Connect BUZZER to a pin that is capable of being a tcpwm.line capable pin.
- All other offsheet connectors can be connected to any IO pin.

Any pins that are not used **MUST** be marked with a Generic No ERC marker (**pvn**).

## 9. Annotating Your Schematics

Now that the schematics have been completed, you need to uniquely identify each of the components. These unique identifiers are called reference designators. As it currently stands, each resistor is identified as "R?". We want each component to be identified with a unique string. This process is called annotation.

a) Select Tools☐Annotation☐Annotate Schematics or **taa**.

b) Select Reset All
c) Select Update Change List
d) Select Accept Changes
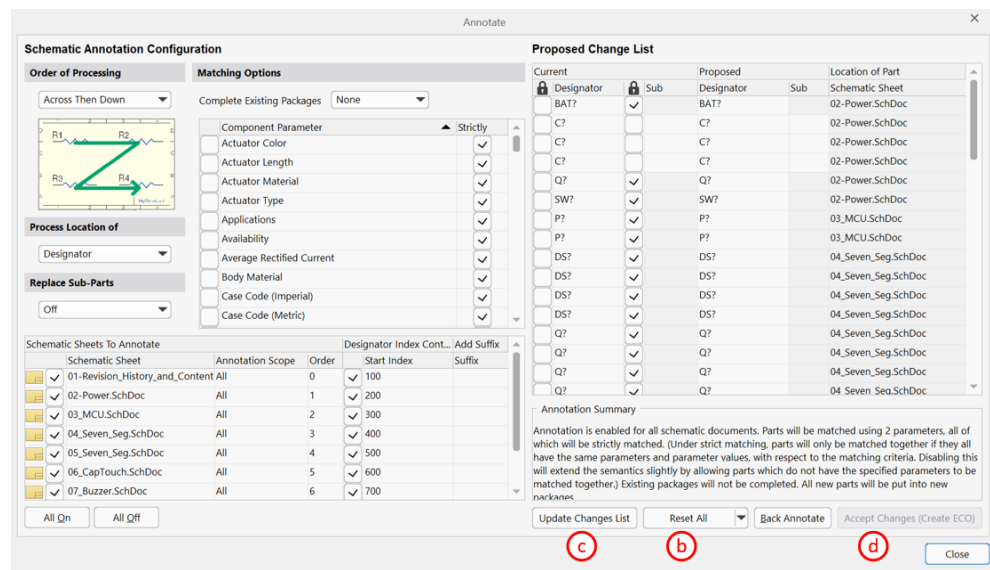
e) Select Validate Changes
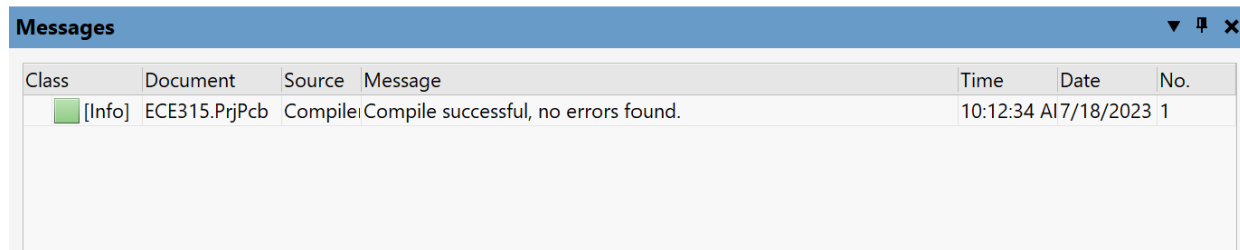f) Select Execute Changes
g) Select Close
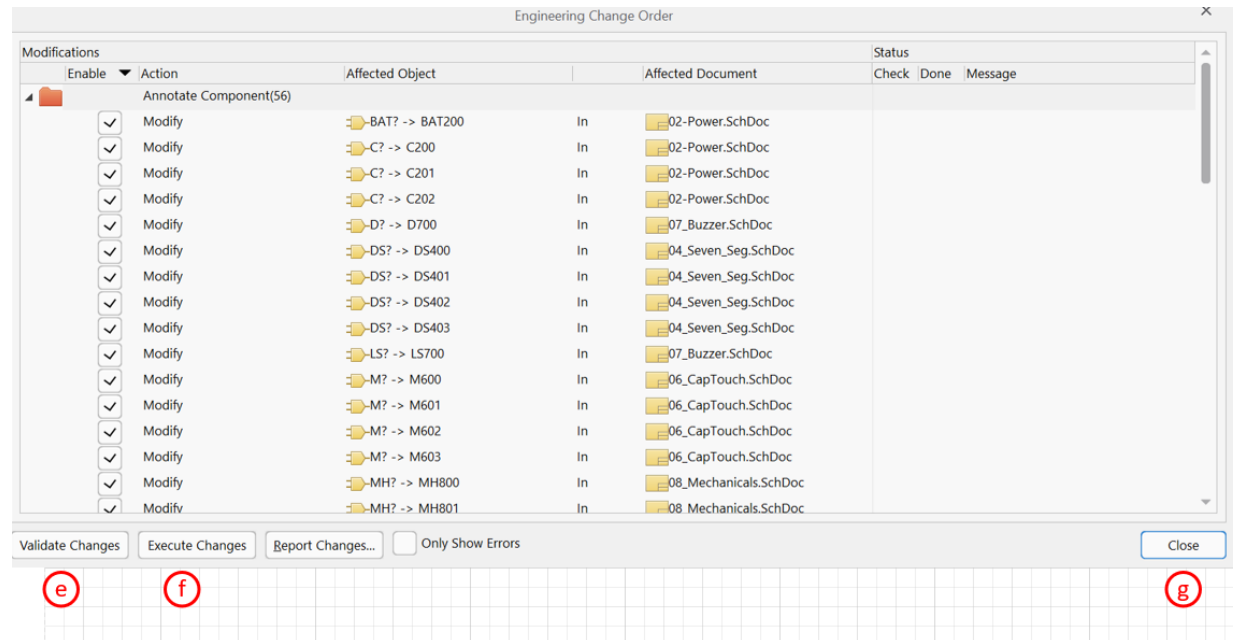


h) Select Close

# 10. Validating the Design

All of the components have been placed and connected with wires, but it's a good idea to have Altium run some basic validation of your schematics. This helps to identify problems that you may have overlooked.

a) Execute the Validation Dialog by selecting Project☐Validate PCB Project or (**cc**).

b) In the lower left hand corner, select Panels☐Messages

The resulting dialog will display any errors that were detected. You can ignore any of the off-grid messages. If you hover over the warning, it will display the full text of the warning. In the example below, the footprints for two parts are missing. This is an error that you would need to fix before completing the lab.

| Messages | | | | | | ▼ ⊓ ✕ |
|---|---|---|---|---|---|---|
| Class | Document | Source | Message | Time | Date | No. |
| [Info] | ECE315.PrjPcb | Compiler | Compile successful, no errors found. | 10:12:34 AI | 7/18/2023 | 1 |

If you have any other warnings/errors, examine your schematics and try to fix any problems reported. If you are unsure of what the error/warning means, ask your TA.

**Take a screen capture of your messages window and save it as validate.png**

**Take a screen capture of 03_LED-GREEN and save it as 03_LED-GREEN.png.  Be sure to maximize Altium and capture the entire page.**

**Take a screen capture of 04_LED-YELLOW and save it as 04_LED-YELLOW.png.  Be sure to maximize Altium and capture the entire page.**

**Take a screen capture of 05_LED-RED and save it as 05_LED-RED.png.  Be sure to maximize Altium and capture the entire page.**

**Take a screen capture of 06_MCU and save it as 06_MCU.png.  Be sure to maximize Altium and capture the entire page.**

**Take a screen capture of 08_Power and save it as 08_Power.png.  Be sure to maximize Altium and capture the entire page.**

One thing to note is that if you update a schematic symbol or a footprint in the schematic/pcb libraries, those changes are not automatically applied to the schematic pages.  In order to update the schematic sheet(s) with your changes, you need to **right click** on the part and then choose ***Part Actions→Update Selected From Libraries***.

You then will want to run Validate☐Execute☐Close

## 11.  Schematic Editor Shortcuts

| Action | Short Cut |
|---|---|
| Place Wire | **pw** |
| Place Power Port | **po** |
| Place Offsheet Connector | **pc** |
| Place Generic No ERC | **pvn** |
| Rotate | **Space** |
| Exit Current Mode | **Esc** |
| Modify Properties of Selected Object | **Tab** |
| Zoom | **Mouse Scroll Wheel** |
| Pan | **Right Mouse Button** |
| Annotate Schematics | **taa** |
| Validate Project | **cc** |